

Working Model[®] 3D

Version 2.0

for Windows[®] 95 and Windows[®] NT

User's Manual



Information in this document is subject to change without notice and does not represent a commitment on the part of Knowledge Revolution. The software described in this document is furnished under a license agreement or non-disclosure agreement. The software may be used or copied only in accordance with the terms of the agreement. It is against the law to copy the software on any medium except as specifically allowed in the license or non-disclosure agreement. No part of this manual may be reproduced or transmitted in any form or by any means, electronic or mechanical, including photocopying and recording, for any purpose without the express written permission of Knowledge Revolution.

© Copyright Knowledge Revolution 1997. All rights reserved.
Published and printed in the U.S.A.

Knowledge Revolution, Working Model, the Working Model logo, Smart Editor, Working Model Basic, Working Model 3D, AutoMotion, AutoMotion 3D, Motion Edge, and MotionWorks are trademarks of Knowledge Revolution.

Working Model is a registered trademark of Knowledge Revolution.

Microsoft and Windows are registered trademarks of Microsoft Corporation.

Video for Windows is a trademark of Microsoft Corporation.

ACIS is a registered trademark of Spatial Technologies, Incorporated.

All other products or name brands are trademarks of their respective holders.

Working Model, Inc.
A Division of Knowledge Revolution

66 Bovet Road, Suite 200
San Mateo, California 94402
Phone: (415) 574-7777
Fax: (415) 574-7541

www.workingmodel.com

Contents

Chapter 1	Introduction.....	1-1
1.1	What Is Working Model 3D?.....	1-1
1.2	Operating Concept.....	1-1
1.3	Powerful, Easy-to-Use Features.....	1-2
1.4	Installing Working Model 3D	1-4
	Getting Started Guide.....	1-4
	Installing Working Model 3D.....	1-4
1.5	About Working Model 3D's Documentation.....	1-5
Chapter 2	Menus, Tools and Controls.....	2-1
2.1	Working Model 3D Menus.....	2-1
	The File Menu.....	2-2
	The Edit Menu.....	2-3
	The View Menu.....	2-4
	The World Menu.....	2-5
	The Object Menu.....	2-6
	The Grid Menu.....	2-7
	The Measure Menu.....	2-7
	The Window Menu.....	2-8
	The Help Menu.....	2-8
2.2	Working Model 3D Toolbars	2-9
	The Standard Toolbar.....	2-9
	The View Toolbar	2-10
	The Edit Toolbar	2-11
	The Sketch Toolbar.....	2-12
	The Run Toolbar	2-13
2.3	Object Manager.....	2-14
2.4	Tape Player Control.....	2-15
2.5	Coordinates Bar.....	2-15
2.6	Status Bar.....	2-16
Chapter 3	Coordinate System.....	3-1
3.1	Understanding the Coordinate System.....	3-1
3.2	Attaching Coordinate System to Objects	3-2
	Effects of Attaching Global Coordinate System.....	3-3

	Effects of Making Global Coordinate System Permanent.....	3-4
	Attaching Global Coordinate System to a Body.....	3-4
	Attaching Global Coordinate System to a Coord.....	3-5
	Resetting the Global Coordinate System	3-6
3.3	The Edit Grid.....	3-7
	Specifying the Edit Grid Orientation	3-8
	Properties of the Edit Grid	3-9
Chapter 4	Manipulating Views.....	4-1
4.1	Changing Views	4-1
	Head-On Views.....	4-1
	Look at Selected Object	4-2
	Perspective and Isometric Views	4-2
	Zooming.....	4-3
	Panning.....	4-4
	Rotating.....	4-5
	Undoing View Changes	4-5
	Setting and Recalling Home View.....	4-6
	Viewing All Objects in the Document.....	4-7
4.2	Viewing a Model with Multiple Windows.....	4-7
	Creating and Deleting a Window.....	4-8
4.3	Rendering Modes.....	4-8
	Wireframe Mode.....	4-8
	Shaded Mode.....	4-9
	Modifying the Background Color.....	4-11
Chapter 5	Workspace.....	5-1
5.1	Numbers and Units.....	5-1
	Numerical Display Format.....	5-3
5.2	Gravity.....	5-3
5.3	Creating the Ground Surface for Simulation.....	5-4
Chapter 6	Bodies.....	6-1
6.1	Introduction	6-1
6.2	Body Types.....	6-2
	Box.....	6-2
	Cylinder.....	6-2
	Sphere.....	6-3
	Extruded Polygon.....	6-4
	Polymesh (Polyhedron).....	6-5
6.3	Creating Bodies.....	6-5
	Edit Grid and Bodies.....	6-6
	Creating a Body on the Edit Grid.....	6-7

Chapter 7

	Creating a Body Between Coords.....	6-11
	Creating a Ground Surface.....	6-13
6.4	Manipulating Bodies	6-13
	Selecting Bodies.....	6-13
	Moving and Rotating Bodies.....	6-16
	Deleting Bodies.....	6-18
6.5	Common Body Properties	6-18
	The Properties Window.....	6-19
	Local Coordinates of a Body.....	6-20
	Changing Position and Orientation.....	6-21
	Changing Velocity.....	6-22
	Fixing a Body in the World.....	6-24
	Changing Body Geometry.....	6-25
	Changing Appearance and Body Name.....	6-26
	Changing Mass Properties (Body Properties).....	6-28
	Changing Material Properties.....	6-30
6.6	Specifying Collisions Among Bodies	6-32
	Assembling Models.....	7-1
7.1	Putting a Model Together: An Overview.....	7-1
	Coords and Constraints	7-2
7.2	Working with Coords.....	7-4
	Creating and Attaching Coords.....	7-4
	Attaching and Detaching a Coord from a Body.....	7-4
	Modifying Coord Properties.....	7-5
7.3	Creating Constraints.....	7-7
	An Example: Creating a Revolute Joint.....	7-7
	An Example: Creating a Linear Spring/Damper.....	7-10
	Creating Multiple Constraints from a Pair of Coords.....	7-12
7.4	The Create Constraint Window.....	7-13
	Constraint Type.....	7-14
	Attachment Targets.....	7-14
	Assembly Behavior.....	7-15
7.5	Applying External Forces or Torques	7-17
7.6	Modifying Constraints.....	7-18
	Understanding Common Constraint Properties.....	7-18
	Selecting Constraints.....	7-19
	Modifying Attachment Positions	7-21
	Modifying Constraint Properties.....	7-22
	Changing Constraint Type	7-24
	Modifying Constraint Appearance and Name.....	7-25
	Deleting Constraints.....	7-27

Chapter 8

7.7	Editing Constrained Bodies.....	7-28
	Modifying Configuration of a Constrained Body.....	7-28
	Dragging and Rotating Bodies.....	7-28
7.8	Splitting and Joining Constraints.....	7-30
	Constructing a Closed-Loop Assembly.....	7-33
	Changing Configuration of a Closed-Loop Assembly.....	7-35
	Impossible Configurations	7-36
	Constraints Reference.....	8-1
8.1	Introduction	8-1
8.2	List of Constraints	8-3
8.3	The Constraint Properties Window.....	8-3
8.4	Rigid Joint	8-5
	Assembly Behavior of Rigid Joint.....	8-5
	Rigid Joint Properties.....	8-7
8.5	Rigid Joint on Slot.....	8-8
	Assembly Behavior of Rigid Joint on Slot.....	8-8
	Properties of a Rigid Joint on Slot	8-10
8.6	Revolute Joint.....	8-11
	Assembly Behavior of a Revolute Joint.....	8-11
	Properties of a Revolute Joint	8-13
8.7	Revolute Joint on Slot	8-13
	Assembly Behavior Revolute Joint on Slot.....	8-14
	Properties of Revolute Joint on Slot.....	8-15
8.8	Spherical Joint.....	8-16
	Assembly Behavior of a Spherical Joint.....	8-16
	Properties of a Spherical Joint.....	8-18
8.9	Spherical Joint on Slot.....	8-18
	Assembly Behavior Spherical Joint on Slot.....	8-19
	Properties of a Spherical Joint on Slot.....	8-20
8.10	Linear Spring/Damper.....	8-21
	Assembly Behavior of a Linear Spring/Damper.....	8-22
	Properties of a Linear Spring/Damper	8-22
8.11	Revolute Spring/Damper.....	8-24
	Assembly Behavior of a Revolute Spring/Damper.....	8-24
	Properties of a Revolute Spring/Damper	8-25
8.12	Linear Actuator.....	8-27
	Assembly Behavior of a Linear Actuator.....	8-27
	Properties of a Linear Actuator.....	8-27
8.13	Revolute Motor.....	8-29
	Assembly Behavior of a Revolute Motor.....	8-29
	Properties of a Revolute Motor.....	8-30

8.14	Rope.....	8-32
	Assembly Behavior of a Rope.....	8-32
	Properties of a Rope.....	8-32
8.15	Separator.....	8-34
	Assembly Behavior of a Separator.....	8-34
	Properties of a Separator.....	8-34
8.16	Rod	8-36
	Assembly Behavior of a Rod	8-36
	Properties of a Rod.....	8-36
8.17	Force.....	8-38
	Attaching Force.....	8-38
	Properties of Force.....	8-39
8.18	Torque.....	8-39
	Attaching Torque.....	8-40
	Properties of Torque.....	8-40
Chapter 9	Running Simulations.....	9-1
9.1	Opening, Saving, and Closing Files.....	9-1
9.2	Running and Stopping a Simulation.....	9-2
9.3	Recording a Simulation.....	9-3
	Erasing Simulation History.....	9-4
	Setting Initial Condition from an Existing Simulation.....	9-5
9.4	Playing Back a Simulation	9-5
	Frame-by-Frame Playback.....	9-6
	Skipping Frames for Faster Animation.....	9-6
	Playing a Simulation Backward.....	9-7
	Moving to a Specific Frame.....	9-7
9.5	Printing Simulations.....	9-8
9.6	Controlling Simulation Accuracy.....	9-9
	Simulation Accuracy Modes.....	9-9
	Animation Step.....	9-10
	Configuration Error.....	9-10
	Integrator.....	9-10
	Tips for Faster Simulation.....	9-11
Chapter 10	Measuring Simulation Data.....	10-1
10.1	Creating Meters.....	10-1
	What Can Be Measured with Meters	10-1
10.2	Customizing Meters.....	10-3
	Switching Between Digital and Graph.....	10-3
	Modifying Meter Position and Size	10-4
	Adjusting Scale of Meter Axes	10-5

	Modifying Meters to Display Customized Properties.....	10-8
	Meters and Unit Labels.....	10-9
10.3	Printing Meter Data.....	10-9
10.4	Deleting Meters.....	10-11
Chapter 11	Import and Export.....	11-1
11.1	Available Import/Export Options.....	11-1
	Import.....	11-1
	Export.....	11-1
11.2	Importing ACIS Files.....	11-2
	Reading ACIS Files.....	11-2
	Tuning Facet Resolution	11-4
11.3	Importing Working Model 3D Files.....	11-6
11.4	Exporting Video for Windows	11-7
	Video Compression Options	11-8
11.5	Exporting Meter Data.....	11-9
	Export Options	11-11
	Printing the Tabulated Meter Data.....	11-12
Chapter 12	Using Formulas.....	12-1
12.1	Units and Formulas.....	12-1
	How Working Model 3D Handles Unit Conversions	12-2
12.2	Customizing Constraints	12-3
	Creating a Torque-Limiting Velocity Motor.....	12-3
	Constraint that Breaks at Certain Load.....	12-4
12.3	Customizing Meters.....	12-6
	Changing X-axis.....	12-6
	Finding Patterns in Nonlinear Systems.....	12-7
	Using Meters as Variables.....	12-9
Appendix A	Technical Information.....	A-1
A.1	Numerical Methods.....	A-1
A.2	Time Step and Performance	A-4
	Choosing a Proper Time Step.....	A-4
	Variable Time Step.....	A-4
A.3	How Working Model 3D Bounds Errors.....	A-5
	Relative Acceptable Error.....	A-6
	Absolute Acceptable Error.....	A-7
	Acceptable Error	A-7
	Choosing an Acceptable Error for Your Simulation.....	A-8
A.4	Accuracy Dialog and Simulation Parameters.....	A-9
	Integrators.....	A-9
	Animation Step.....	A-11

	Configuration Error.....	A-11
	Integration Time Step.....	A-12
	Simulation Error Tolerances	A-12
	Warnings	A-13
A.5	Simulating Collisions	A-14
	How Working Model 3D Simulates Collisions.....	A-14
	Computing Collision Forces.....	A-16
	Collision Force in Meters.....	A-17
A.6	Hints for Controlling Simulation Accuracy	A-17
	Preventing Unstable Simulations.....	A-17
	Fine-tuning the Simulation Accuracy	A-18
	Correcting Systems that Appear to Gain Energy.....	A-19
	Chaotic Systems in Working Model 3D	A-19
A.7	Technical References.....	A-19
	On Numerical Methods.....	A-20
	On Mechanics.....	A-20
Appendix B	Formula Language Reference.....	B-1
B.1	Overview	B-1
B.2	Scalars and Vectors	B-1
B.3	Identifiers.....	B-2
B.4	Mathematical Functions	B-6
B.5	Simulation Functions.....	B-12
B.6	Operators	B-14
B.7	Predefined Variables	B-18
Appendix C	Keyboard Shortcuts and Troubleshooting.....	C-1
C.1	Keyboard Shortcuts.....	C-1
C.2	Troubleshooting.....	C-2
	Simulations with Polyhedra Run Very Slow.....	C-2
	Simulation with Excessive Body Overlap.....	C-3
	A Coord Does Not Attach Correctly to a Body Surface.....	C-3
	Menu Items are Disabled.....	C-3
	Index.....	I

x

A Word from Working Model, Inc.

We would like to thank you for purchasing one of our products. At Working Model, Inc., we are committed to providing you with quality products which are easy to use. The product you have purchased is the result of thousands of hours of work by dedicated professionals.

To provide quality, we are always listening to our most important source of feedback: customers like you. We would like to know how you use our product and how we can better serve you. We are constantly improving the Working Model family of products, and look forward to your comments and suggestions. Please feel free to call me directly if I can be of service.

Working Model, Inc.
A Division of Knowledge Revolution
66 Bovet Road, Suite 200
San Mateo, CA 94402
(415) 574-7777

A handwritten signature in dark ink that reads "Dave Baszucki". The signature is written in a cursive, flowing style.

David Baszucki, President and CEO
Knowledge Revolution

C H A P T E R 1

Introduction

1.1 What Is Working Model 3D?

Knowledge Revolution's Working Model 3D is a three-dimensional dynamics simulation program for desktop computers running the Windows® 95 or Windows® NT operating systems. Working Model 3D combines advanced motion simulation technology with sophisticated, easy-to-use editing capabilities and serves as a complete, professional tool for engineering simulation.

The dynamics simulation engine provides a translation of real world Newtonian mechanics to desktop computers, while the simple yet powerful graphical user interface makes it easy to experiment with various scenarios and situations.

1.2 Operating Concept

The operating concept of Working Model 3D is simple. Define a set of rigid bodies and constraints (e.g., motors, springs, and joints). With a click of the mouse, Working Model 3D uses its powerful simulation engine to put your model in motion.

Working Model 3D's versatile Properties window allows you to fine-tune a simulation with its tabs and multi-purpose pages that provide access to mass properties, geometry, constraint characteristics, and attachment conditions. You can create meters to plot the data that is taken during a simulation. You can design a model in your favorite CAD program and export it to Working Model 3D. This User's Manual will explain how to use these features.

1.3 Powerful, Easy-to-Use Features

Simulation Engine

The simulation engine of Working Model 3D is designed for both speed and accuracy. It calculates the motion of interacting bodies using advanced numerical analysis techniques. The engine allows the construction of complex systems, and computes their motion under a variety of constraints and forces. In addition to user-imposed constraints such as spring dampers or joints, the engine has the capability to simulate multi-body interactions such as collisions, gravity, and external load conditions. The engine is fully configurable, and every aspect of a simulation from the time step (fixed or variable) to the integration technique can be specified by the user.

Modeling Objects

Each body has a set of physical properties, including mass, coefficient of restitution, coefficient of friction, moments of inertia, positions, and velocities. Rotational Constraints include rigid joints, revolute joints, spherical joints, and slot joints (rigid, revolute, spherical). Linear constraints include rods, ropes, separators, springs, and dampers. The properties of these constraints (e.g., a spring constant) are user definable by numeric or equation input.

In addition, forces, torques, actuators, and motors are available. Properties can be defined by numeric values or equations.

Smart Constraint Joining

Attach a point coordinate (Coords) to an object, position it exactly, and then join it to another “coord” to create joints—the objects self-assemble. Furthermore, you can graphically position objects on the screen, or use a set of numerical entry tools to specify exact configurations.

Automatic Collision Detection

Collisions are simulated automatically, such as back-lash in gears, parts on conveyors, accident reconstruction, cams/followers, and wedges.

Simulation Control

Integrator Control—Users can choose from Euler or Kutta-Merson integration techniques with fixed or variable time-stepping. Users can adjust error/accuracy levels and animation step.

Equation Parsing—A mathematical parsing language allows the user to define formulas to control parameters of bodies, constraints, and forces.

Simulation Forces—Run simulations in a gravity field or turn gravity completely off.

Run Control—Reset, single step, or pause the simulation at any time. Save the simulation histories to a file.

Unified Modeling, Simulation, and Analysis—No separate pre-processing, analysis, and post-processing phases. Everything is unified in one design/simulation space—the way you think and work.

Measurement & Analysis

Real-Time Graphs—Measure time, velocity, acceleration, positions, momentum, orientation, angular velocity, angular acceleration, force, torque, friction, and contact. Enter equations to create custom measurements. Show any value on any axis. Scale graphs to any value.

Unit Control—Simulate and measure in virtually any unit system, including SI, English, CGS, or custom combinations. Change unit systems at any time.

Point Coordinate “Coord” Measurements—Attach a “coord” to any part of a body and measure position, velocity, acceleration, orientation, angular velocity, or angular acceleration.

Data Exchange

ACIS Import—Import ACIS-based (.sat) CAD models and assemblies.

Video Export—Export simulations to Video for Windows (.avi) formats.

Data Export—Export graph data to tab-delimited text files for use in spreadsheets.

Model Viewing

Rendering—View models in shaded, wire-frame.

Representation—View models in perspective or isometric.

Dynamic Viewing—Dynamic manipulation of parts and views.

1.4 Installing Working Model 3D

This section includes instructions for installing Working Model 3D and the sample files on a hard disk.

Getting Started Guide

Before you install Working Model 3D, please read the enclosed *Getting Started with Working Model 3D*. You will find useful information about installation, as well as the information that was not included in this manual, such as last-minute changes that may have taken place with Working Model 3D.

Installing Working Model 3D

Working Model 3D comes with a setup program that will install the application and sample files on your hard disk.

NOTE: Please refer to the Getting Started with Working Model 3D Guide for the latest installation information.

1. Insert the Working Model 3D CD-ROM.

The CD-ROM automatically shows the welcome screen.

2. Follow the instructions given by the setup program.

Working Model 3D is now installed. Be sure to store the CD-ROM in a safe place.

1.5 About Working Model 3D's Documentation

Start with the Tutorial or Online Workshops

Working Model 3D includes the following documentation:

- the *User's Manual* you are currently reading
- *Getting Started Guide with Working Model 3D*
- *Tutorial Guide*

Online help is also available in Working Model 3D's Help menu:

- *Working Model 3D Help*
- CAD integration tutorials.

To learn how to use Working Model 3D most effectively, begin with the *Tutorial Guide* or the online workshops in Working Model 3D Help. The exercises give you step-by-step instructions to guide you through construction of sample models, thereby exposing you to Working Model 3D's most frequently used features. Also, consulting the *User's Manual* frequently will help you learn more about Working Model 3D.

Task-Oriented Organization

Knowing that most people do not have time to read manuals from cover to cover, we designed the User's Manual and online help as a source of "how-to's" for most common features of Working Model 3D.

The rest of the book is organized as follows:

Chapter 2, "Menus, Tools and Controls", provides an overview of the menu items and toolbar icons, as well as controls available in Working Model 3D's application window.

Chapter 3, "Coordinate System", discusses how Working Model 3D employs various coordinate systems. In addition, the chapter explains how to use the Edit Grid feature effectively.

Chapter 4, "Manipulating Views", shows you how you can use the powerful tools available in Working Model 3D to "navigate" through your model and add realism to it by changing the rendering mode.

Chapter 5, "Workspace", explains how to change a property that affects the entire simulation, such as the unit system and gravity setting.

Chapter 6, “Bodies”, serves as a reference for all the body types available in Working Model 3D. You can find instructions to create them, change their properties, and specify the collision property among them.

Chapter 7, “Assembling Models”, explains step-by-step how to put a model together by connecting bodies with constraints. The chapter also introduces *Coords* (point elements with a coordinate system attached) that serve as a basis to define constraints.

Chapter 8, “Constraints Reference”, is a reference section for all types of constraints available in Working Model 3D. The chapter contains the complete information on their properties and behaviors.

Chapter 9, “Running Simulations”, shows you how to open, save, and close files, as well as various ways to run and play back your simulations.

Chapter 10, “Measuring Simulation Data”, introduces meters. Meters are powerful tools in Working Model 3D to measure simulation data in numerical or plot format.

Chapter 11, “Import and Export”, explains how you can import files into Working Model 3D, or export data and animation to another file.

Chapter 12, “Using Formulas”, introduces you to the formula language, a powerful tool to customize your simulation. The chapter also shows you several examples using the formula language.

Appendix A, “Technical Information”, provides in-depth coverage of how Working Model 3D computes simulations.

Appendix B, “Formula Language Reference”, provides references on Working Model 3D’s formula language synopsis.

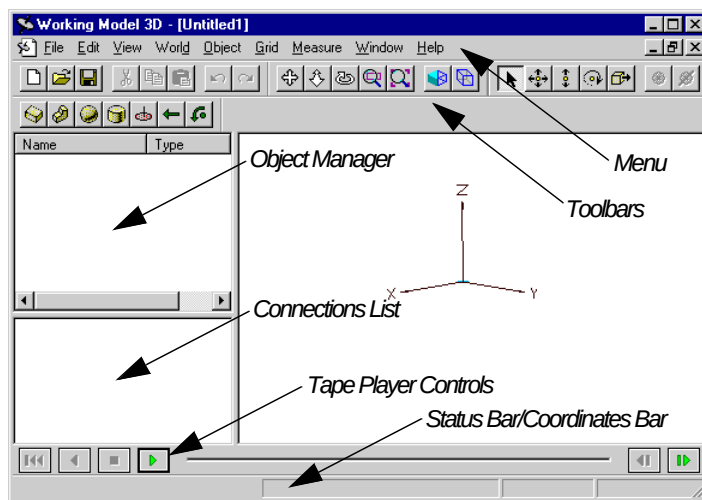
Appendix C, “Keyboard Shortcuts and Troubleshooting”, provides references on keyboard shortcuts and troubleshooting tips.

CHAPTER 2

Menus, Tools and Controls

The Working Model 3D application window appears as shown in Figure 2-1. This chapter discusses menus, tools, and controls that are located at the top of the Working Model 3D document window. For discussion regarding the modeling environment, please refer to later chapters.

Figure 2-1
Working Model 3D Window



2.1 Working Model 3D Menus

Working Model 3D provides a standard menu bar with pull-down menus. The following menu commands are available.



The File Menu

New creates a new, blank Working Model 3D document.

Open opens a previously created Working Model 3D document. You can have multiple documents open at once (subject to available memory on your system).

Close closes the currently active document. If there are changes that need to be saved, you will be prompted.

Save saves the active document to disk. Previously saved documents are updated.

Save As lets you save a copy of an existing document under a new name.

Document Properties presents a dialog box for you to list the document's creator, company, and any comments you want to add to describe the file. This dialog box also shows the version of Working Model 3D in which the file was created.

Import presents a dialog box for importing external Working Model 3D files into the workspace. See Chapter 11, "Menus, Tools and Controls" for more information.

Export Meters presents a dialog for exporting Working Model 3D meter data. The menu item is dimmed when no meter exists in the current document. See "Exporting Meter Data" on page 11–9 for more information.

Export Video presents a dialog for exporting Working Model 3D simulation history to an AVI file (Video for Windows format). See for more information.

Print presents the Print dialog, allowing you to print your simulations or the current meter. See "Printing Simulations" on page 9–8 and "Printing Meter Data" on page 10–9 for more information.

Print Preview presents the Print preview window where you can print the current file and zoom in or zoom out.

Print Setup presents the Print Setup dialog for specifying the printer destination or other controls such as printing to a network printer.

Exit exits Working Model 3D.

Edit	
Undo Cut	Ctrl+Z
Redo	Ctrl+Y
Cut	Ctrl+X
Copy	Ctrl+C
Paste	Ctrl+V
Delete	Delete
Select All	Ctrl+A
Properties	Alt+Enter

The Edit Menu

Undo reverses the last action performed in the simulation. The menu item shows the last action taken (the figure on the left shows an example with undoing Cut), or shows “Can’t undo” if the action is irrecoverable. Working Model 3D supports multiple undo, so you can reverse several actions by selecting Undo again and again.

Redo repeats the last action performed in the simulation. If you had just performed a cut, and then undid it using the Undo command, for instance, the menu item would show as “Redo Cut.”

Cut removes the selected objects from the document and places them on the Clipboard.

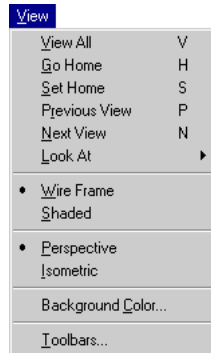
Copy places a copy of the selected objects on the Clipboard.

Paste pastes the contents of the Clipboard into the active simulation.

Delete deletes the selected objects from the simulation without placing the contents onto the Clipboard.

Select All selects all of the objects in the active simulation window.

Properties opens the Properties window, which allows you to set or examine an object’s features and attributes, including **Appearance**, **Position** (Pos), **Velocity** (Vel), **Material**, **Geometry**, **Central Inertia**, and **Center of Mass Offset** (CM).



The View Menu

View All automatically scales and pans the view so that all the objects in the document (bodies, Coords, constraints, and Edit Grid) become visible in the document window.

Go Home sets the view angle to the Home view angle. See “Setting and Recalling Home View” on page 4–6 for more information.

Set Home sets the current view angle as the Home view angle.

Previous View restores the view angle prior to the most recent change. The menu item is not active if you have not changed the view angle at least once. You can retrieve up to 10 previous view changes.

Next View restores the view that was restored by Previous View. The menu item is not active unless you chose the Previous View menu at least once.

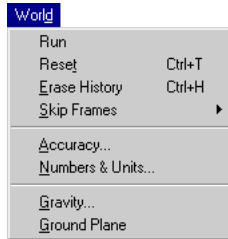
Look At allows you to quickly view the model for several preset views. The menu item provides a submenu with six view angle options that let you choose a “head-on” view (from positive and negative directions of the x-, y-, and z-axes), as well as an option that zooms in on an object selected in the Object Manager list. See “Head-On Views” on page 4–1 or “Look at Selected Object” on page 4–2 for more information.

Wire Frame and Shaded menu items allow you to choose the rendering mode of the current window. See “Rendering Modes” on page 4–8 for more information.

Perspective and **Isometric** menu items switch the projection mode of the current window. See “Perspective and Isometric Views” on page 4–2 for more information.

Background Color presents the Windows color pick dialog to choose the background color of the current document. See “Modifying the Background Color” on page 4–11 for more information.

Toolbars allow you to select the toolbars you want to have displayed as you work.



The World Menu

Run/Stop starts and stops the simulation. The menu only applies to the currently active document (i.e., the document that appears foremost).

Reset stops your simulation and then returns it to the initial conditions (frame zero).

Erase History erases the currently stored simulation history regardless of the current frame number. In effect, the current position and velocity of all objects becomes the set of initial conditions for the next simulation.

NOTE: You cannot undo the Erase History command.

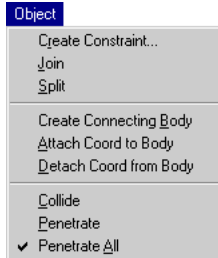
Skip Frames lets you specify various playback rates for your simulations. Skipping more frames provides a faster playback on a previously calculated simulation. The menu item presents a submenu with **1** (i.e., displays every frame), **2** (displays every other frame), **4** (displays one every fourth frame), **8** (you get the idea), and **16**.

Accuracy invokes the Accuracy dialog which allows you to control various simulation parameters. See “9.6 Controlling Simulation Accuracy” for more information.

Numbers & Units presents a dialog to control the unit system and number display format. See “5.1 Numbers and Units” for more information.

Gravity presents the dialog to control the gravity settings. See “5.2 Gravity” for more information.

Ground Plane toggles the existence of the ground plane for the simulation. See “Creating the Ground Surface for Simulation” on page 5-4 for more information.



The Object Menu

Create Constraint invokes the Create Constraint dialog which allows you to create constraints (such as joints) to construct a model.

Join recovers the selected constraint that has been split.

Split temporarily disables the selected constraint and allows individual bodies to be moved separately.

Create Connecting Body creates a cylinder body between two selected Coords.

Attach Coord to Body attaches selected Coords to the selected body, without moving any of the Coords. See “Attaching and Detaching a Coord from a Body” on page 7-4 for more information.

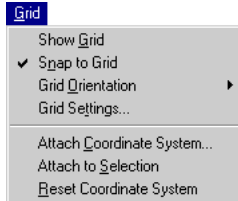
Detach Coord from Body detaches selected Coords from the body or bodies to which the Coords are currently attached.

Collide specifies that selected bodies would collide among one another, when they come into contact.

Penetrate specifies that selected bodies would penetrate one another when they come into contact.

Penetrates All specifies that selected body or bodies would penetrate all other bodies when they come into contact.

NOTE: Not all the items are available to every object. For example, the **Collide** menu is only available when at least two bodies have been selected.



The Grid Menu

Show Grid toggles the Edit Grid on and off.

Snap To Grid toggles the grid snap feature.

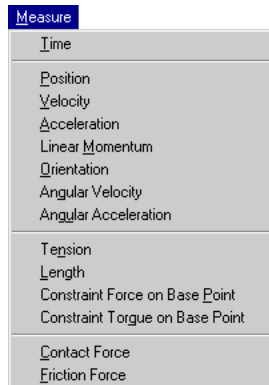
Grid Orientation specifies the orientation of the Edit Grid. The menu item provides a submenu with the following options: **XGrid**, **YGrid**, **ZGrid**, or the **Automatic**. The **Automatic** option automatically determines the plane that appears most convenient for you to edit models, and attaches the Grid orientation accordingly.

Grid Settings invokes the Grid Settings dialog.

Attach Coordinate System invokes the Coordinate System dialog.

Attach to Selection attaches the global coordinate system to the selected Coord or the selected face of a body.

Reset Coordinate System resets the global coordinate system to the world coordinates. The menu item is not active unless the current global coordinate system is currently attached to an object.



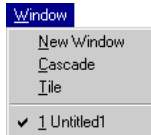
The Measure Menu

The Measure menu allows you to create meters to measure properties of a Working Model 3D simulation. See Chapter 10, “Measuring Simulation Data” for more information.

Time creates a meter that measures time.

[List Measurements] The rest of the menu lists the valid measurements for any selected object. The figure shown on the left, for example, lists valid measurements when you select a single body.

When you select two bodies, the Measure menu changes so that you can measure forces that act between a pair of bodies, such as the contact force.



The Window Menu

New Window opens a new window to view the current model. See “Viewing a Model with Multiple Windows” on page 4–7 for more information.

NOTE: The New Window simply opens another window to view the same model with a different angle. Use the File->New to open an entirely new document.

Cascade arranges any open windows in a cascade, with all the titles visible, so that you can have access to more information and options.

Tile arranges any open windows so their contents are at least partially visible on the screen.

[List of Currently Open Document Windows] A list of windows is appended to the bottom of the Window menu.

The Help Menu

The Help menu provides access to Working Model 3D’s online help and tutorials. See the *Getting Started Guide* for more information on troubleshooting and getting technical support.

2.2 Working Model 3D Toolbars

The Working Model 3D toolbars provide access to the tools you use to create and work with simulations.

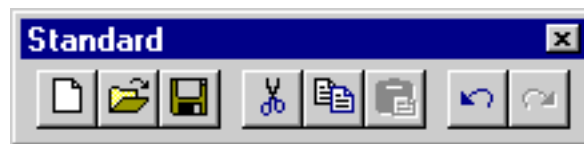
Any action you take with the tools affects the currently active document. If you have multiple windows open, the effect on one window is immediately replicated on all other windows.

To use a tool, click once on the tool icon; after using it once, it will revert back to the Select (Arrow) tool. To keep using the tool, double-click it, and it will not revert to the Select tool.

The individual toolbars are very flexible—you can leave them “docked” at the top of the document window, or you can move them to anywhere on your screen. You can also use the View menu’s Toolbars command to show or hide them as you choose.

The Standard Toolbar

Figure 2-2
Standard Toolbar



The **New** button creates a new, blank Working Model 3D document using the current default settings.



The **Open** button opens a previously created Working Model 3D document. You can have multiple documents open at once (subject to available memory on your system).



The **Save** button saves the active document to disk. Previously saved documents are updated.



The **Cut** button removes the selected objects from the document and places them on the Clipboard.



The **Copy** button places a copy of the selected objects on the Clipboard.



The **Paste** button pastes the contents of the Clipboard into the active simulation.



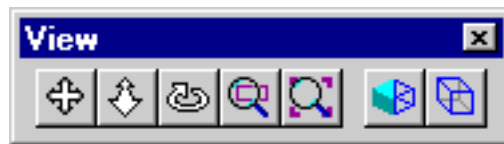
The **Undo** button reverses the last action performed in the simulation.



The **Redo** button repeats the last action performed in the simulation.

The View Toolbar

Figure 2-3
View Toolbar



The **Pan** tool allows you to pan the view parallel to the view plane. Click the tool, and drag on the document window while holding down the mouse button to pan. Alternatively, you can press the F2 key to access the tool. See “Panning” on page 4-4 for more information.



The **Zoom In/Out** tool is used to move the camera in or out of the view plane. Alternatively, you can press the F3 key to access the tool. See “Zooming” on page 4-3 for more information.



The **Rotate** tool allows you to rotate the view direction. Click the tool, and drag on the document window while holding down the mouse button to rotate the view angle. Alternatively, you can press the F4 key to access the tool. See “Rotating” on page 4-5 for more information.



The **Box Zoom** tool allows you to zoom into a rectangle.



The **ViewAll** tool zooms out to view the entire world, allowing you to see the whole document. Click the tool to immediately see everything. This command is also available in the View menu as View All.



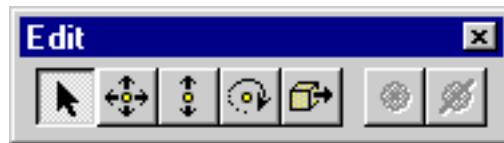
The **ToggleWireframe** tool lets you go back and forth between wireframe and shaded mode. This function is also available in the View menu, with Shaded and Wireframe being the two choices which are here combined in one button.



The **ToggleIsometric** tool lets you go back and forth between Perspective and Isometric mode. This function is also available in the View menu, with Perspective and Isometric being the two choices which are here combined in one button.

The Edit Toolbar

Figure 2-4
Edit Toolbar



The **Select** tool is used to select an object or a group of objects. See “Moving and Rotating Bodies” on page 6–16 for more information. Once you have selected a body, the tool switches to box-select mode if you move the mouse.

When an object is selected, the body is drawn with a thicker frame (wireframe mode) or with an adorning box (shaded mode). Constraints are shown with a bounding box, and coords show axes and a box around the base.



The **Drag** tool is used to select and drag an object or a group of objects parallel to the Edit Grid. See “Moving and Rotating Bodies” on page 6–16 for more information.



The **Drag-Z** tool is used to select an object or a group of objects, or to drag selected objects perpendicular to the Edit Grid. See “Moving and Rotating Bodies” on page 6–16 for more information.



The **Rotate** tool is used to select and rotate a body or a set of bodies. See “Moving and Rotating Bodies” on page 6–16 for more information.



The **Resize** tool is used to select and resize a body or a set of bodies. See “Moving and Rotating Bodies” on page 6–16 for more information.



The **Join/Create Constraint** button is used to bring up the Create Constraint window to construct constraints between selected objects. See “The Create Constraint Window” on page 7–13 for more information.



The **Split** button is used to disable constraints temporarily. See “Splitting and Joining Constraints” on page 7–30 for more information.

The Sketch Toolbar

Figure 2-5
Sketch Toolbar



The **Box** tool is used to create boxes. See “Creating Bodies” on page 6–5 for more information.



The **Extruded Polygon** tool is used to create extruded polygons. See “Creating Bodies” on page 6–5 for more information.



The **Sphere** tool is used to create spheres. See “Creating Bodies” on page 6–5 for more information.



The **Cylinder** tool is used to create extruded cylinders. See “Creating Bodies” on page 6–5 for more information.



The **Coord** tool is used to create and attach Coords. See “Working with Coords” on page 7–4 for more information.



The **Force** tool is used to create and attach external forces. See “Applying External Forces or Torques” on page 7–17 for more information.



The **Torque** tool is used to create and attach external torques. See “Applying External Forces or Torques” on page 7–17 for more information.

The Run Toolbar

Figure 2-6
Run Toolbar



The **Reset** button is used to bring a simulation back to its initial conditions (the first frame).



The **Stop** button is used to stop a simulation. After stopping the simulation, you can click the Run button to resume the simulation or click the Reset button to reset the simulation.



The **Run** button starts the simulation on the currently active document.

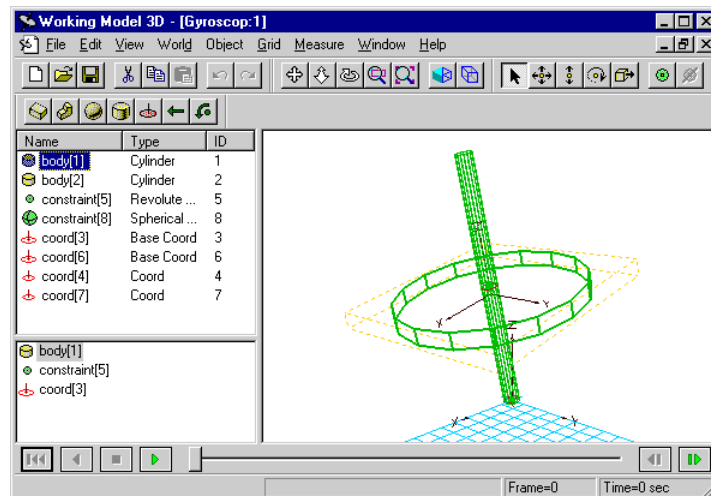
2.3 Object Manager

The two panes on the left side of the document window make up the Object Manager (shown in Figure 2-7). The Object Manager provides a convenient way to view information about the objects and connections in your model, and to quickly and accurately select objects.

The top pane, or Object list, shows all the objects in the open document. When you select an object, the bottom pane, or Connections list, displays all of the other bodies, Coords and Constraints attached to the selected object.

When you select an object in the Object Manager, you can use the right mouse button to display a menu with commands relevant to that particular object. The selected object is also highlighted in any open drawing windows to help you identify it.

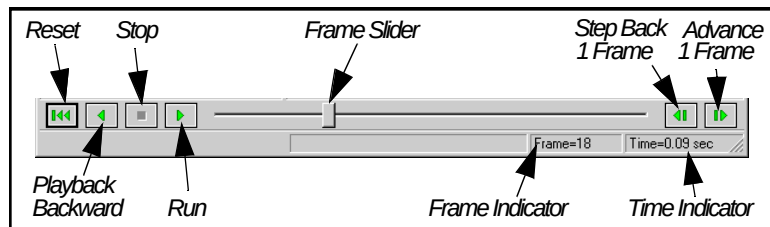
Figure 2-7
Object Manager



2.4 Tape Player Control

While running a simulation, Working Model 3D shows the advancing frame number and time elapsed in the Tape Player (shown in Figure 2-8), located at the bottom of the application window. The Tape Player controls allow you to run the recorded simulation forward and backward, to run and pause a simulation, and to advance or retrace the simulation frame by frame.

Figure 2-8
Tape Player Control

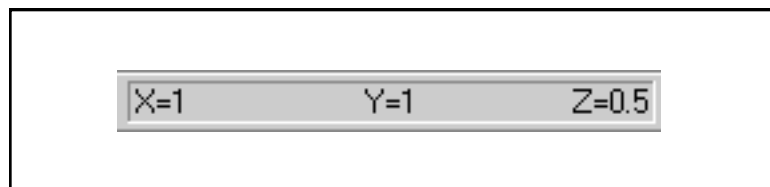


See “Playing Back a Simulation” on page 9-5 for more about the Tape Player controls.

2.5 Coordinates Bar

The Coordinates Bar (shown in Figure 2-9), located at the bottom of the application window, shows the current body position, body dimensions and other information, depending on what you are doing in Working Model 3D. The Coordinates Bar is only visible when a body or Coord is being dragged, rotated, or resized.

Figure 2-9
Coordinates Bar

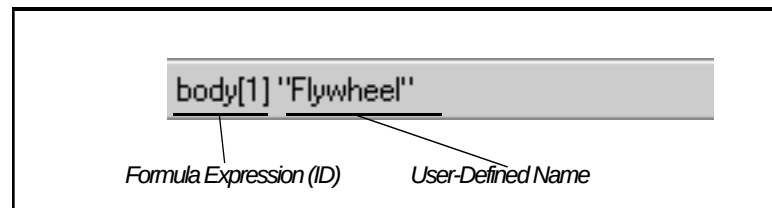


2.6 Status Bar

The Status Bar, (shown in Figure 2-10) located at the bottom left of the application window, displays information about the object currently beneath the mouse pointer.

- For bodies, the status bar displays the formula expression for the body (e.g., `body[1]`), and the body type or the user-defined name (if one is assigned).
- For constraints, the status bar displays the formula expression for the constraint (e.g., `constraint[2]`), and the constraint type or the user-defined name (if one is assigned).
- For Coords, the status bar displays the formula expression for the Coord (e.g., `coord[5]`) and the user-defined name, if assigned.
- For meters, the status bar displays the formula expression for the meter (e.g., `output[3]`).
- For tools (on the toolbar, as shown in Figure 2-2), the status bar provides a brief explanation.

Figure 2-10
Status Bar for a Body



Formula Language Expressions

Working Model 3D provides a powerful formula language that allows you to customize the behavior and characteristics of your model through various functions and control-flow statements. The status bar assists you in devising your formula language expressions, as you can easily identify the objects and their corresponding formula expressions. For more information on the formula language, please refer to Chapter 12, “Using Formulas”.

Assigning Custom Names

You can also define custom names for the objects so that you can quickly identify parts and components in your model. Please see “Changing Appearance and Body Name” on page 6–26 for bodies, as well as “Modifying Constraint Appearance and Name” on page 7–25 for constraints.

CHAPTER 3

Coordinate System

This chapter describes how Working Model 3D defines the coordinate system and represents the configuration of objects in a document.

3.1 Understanding the Coordinate System

Working Model 3D represents position and orientation of objects using the Cartesian (x, y, and z) coordinate system and body XYZ angle.

Each document has one global coordinate system (the global origin and orthogonal xyz axes). Within a document, individual bodies and Coords have a coordinate system attached to them, and Working Model 3D represents the configuration of these objects in the following fashion:

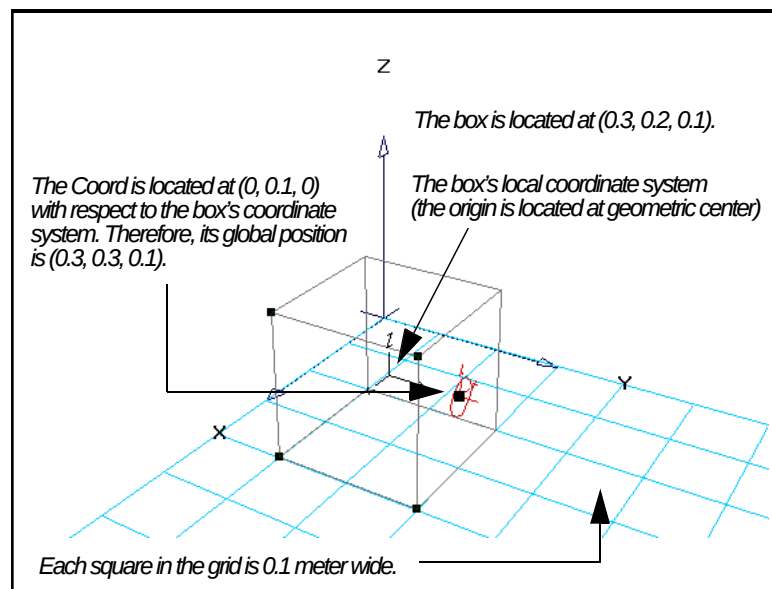
Objects and Coordinates

Object Type	How Its Configuration Is Represented	How the Coordinate System Is Attached
Body (box, cylinder, sphere, polyhedra)	The position of the body's geometric center is represented in terms of the global coordinate frame. The orientation is expressed in body XYZ angles.	The coordinate system is attached at the body's geometric center. The orientation of the system remains fixed with respect to the body.
Coords	The configuration is represented in terms of the body's frame to which the Coord is attached (or global if unattached).	The origin of the coordinate frame coincides with the Coord's position.

Objects and Coordinates

Object Type	How Its Configuration Is Represented	How the Coordinate System Is Attached
Extruded Polygon	The position of the body's geometric center is represented in terms of the global coordinate frame. The orientation is expressed in body XYZ angles	The coordinate system is attached to the first vertex when sketched. The orientation of the system remains fixed with respect to the body.

Figure 3-1 below also shows the relationship between an object and the coordinate system.

Figure 3-1*Objects and Coordinate Systems*

3.2 Attaching Coordinate System to Objects

Working Model 3D allows you to attach the global coordinate system to any object to facilitate the design effort. You can either:

- Attach the coordinate system to a body, or
- Attach the coordinate system to a Coord.

World and Current Coordinate System

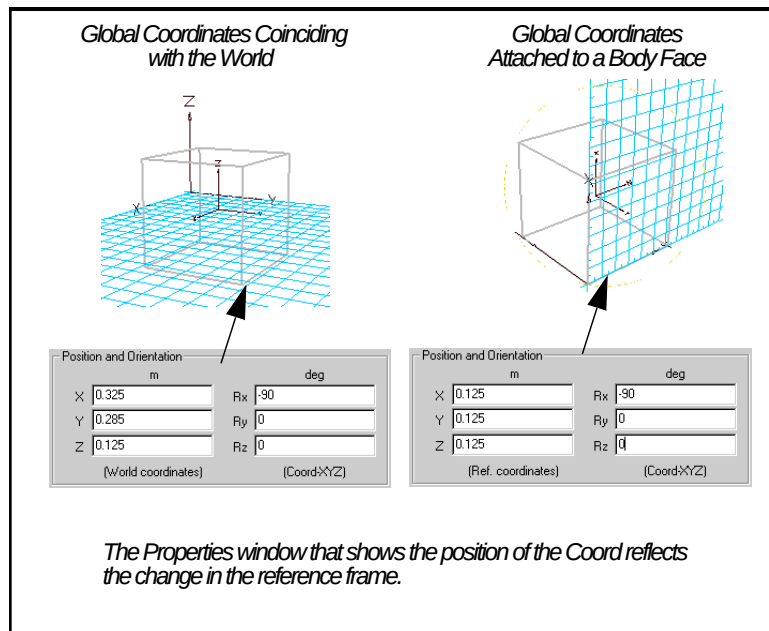
No matter where you attach the global coordinate system, Working Model 3D “remembers” *World* coordinates, or the *original* global coordinate system. On the other hand, the *global coordinate system* may be a coordinate system that is temporarily attached to a body or a Coord, and is referred to as a current coordinate system.

Effects of Attaching Global Coordinate System

When you attach the global coordinate system to a body or a Coord:

- The Properties window shows the position and orientation of a body in the current global coordinate system.
- The Properties window shows position and orientation of Coords in the current global coordinate system (see Figure 3-2).

Figure 3-2
Effects of Attaching Global
Coordinate System to Body Face



On the other hand, the following functionality remains intact regardless of the coordinate system customization:

- The direction in which the gravitational force acts remains fixed with respect to the World. The gravity, if turned on, *always* acts on every body in the negative z direction of the World coordinate system.
- The View menu shortcuts (see “Head-On Views” on page 4–1) act on the World coordinate axes. For example, the “Front” shortcut key *always* sets the viewing direction to the negative x-direction of the World coordinate axes.
- Formula language expressions pertaining to the configuration of bodies and Coords (such as `body[3].p.x` and `coord[5].v.y`) *always* return the value in World. See Appendix B, “Formula Language Reference”, for more information on formula language expressions.

Effects of Making Global Coordinate System Permanent

You can change the world coordinate system by making a change to the global coordinate system permanent. If you do so:

- The global coordinate system will be permanently transformed as specified. You will not be able to undo this action.
- The configuration of all objects will reflect this change.
- The xyz components of force and torque constraints will stay the same, and you may need to reenter those values in the new global frame.

Attaching Global Coordinate System to a Body

You can attach the global coordinate system to a body so that you can position other bodies relative to the body coordinates.

► To attach the global coordinate system to a body or a face:

1. Select the body to which you wish to attach the coordinate system.

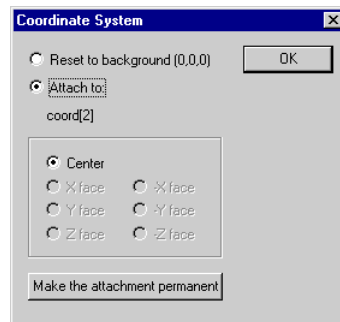
NOTE: You can select the body itself as it is represented in the document, or select its name in the Object Manager, which runs along the left side of the document window.

The selected body is shown with a bounding box when in Shaded mode, and with a thicker frame when in Wireframe mode.

2. Choose Attach Coordinate System in the Grid menu.

The Coordinate System dialog appears (Figure 3-3).

Figure 3-3
Coordinate System Dialog



3. Choose the body's center or the face to which you would like to attach the coordinate system.

The coordinate system is immediately shown in the position you choose. For example, if you choose “-Z Face”, the global coordinate system is attached to the surface of the bounding box that intersects the negative portion of the body's z-axis.

4. Click OK when you are finished.

Attaching Global Coordinate System to a Coord

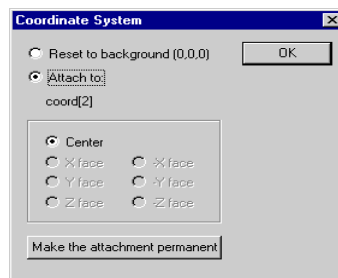
You can attach the global coordinate system to a Coord so that you can position other objects with respect to the Coord.

► **To attach the global coordinate system to a Coord:**

1. Select the Coord to which you wish to attach the coordinate system in the document window or in the Object Manager list.
2. Choose **Attach Coordinate System** in the Grid menu.

The Coordinate System dialog appears (Figure 3-4). Note that all the radio-button options except “Center” are dimmed.

Figure 3-4
Coordinate System Dialog



3. Click the **Attach To (name of coord)** radio button to attach the coordinate system to the coord.
4. Click **OK**.

The global coordinate system coincides with that of the Coord; i.e., the coordinate origin becomes the position of the Coord, and the global coordinate axes are aligned with the Coord's axes.

Resetting the Global Coordinate System

► **To reset the global coordinate system to World coordinates:**

1. Choose **Reset Coordinate System** in the Grid menu.

The global coordinate system is reset to the original (the World coordinate system).

If the menu item is dimmed, the coordinate system has already been reset to the World.

NOTE: If you have made the attachment permanent, essentially changed the World Coordinate System, you will not be able to Reset to any previously defined World Coordinate System!

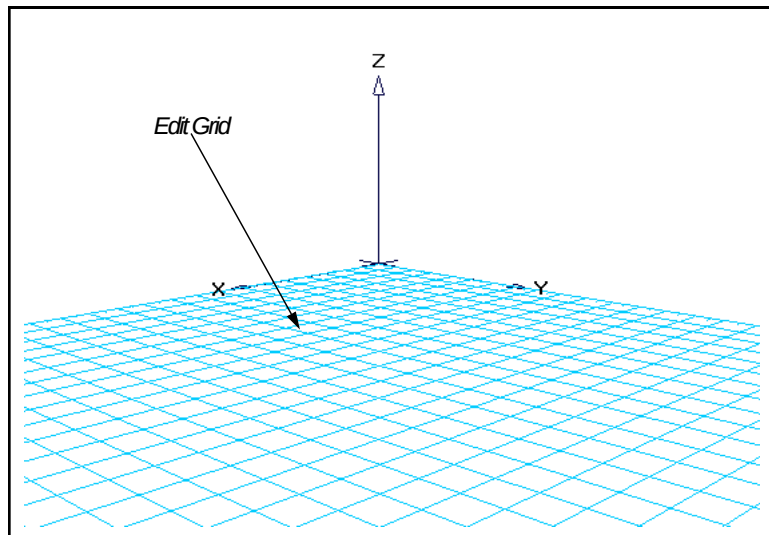
3.3 The Edit Grid

In Working Model 3D, you can sketch and move bodies using the Edit Grid. The Edit Grid is turned off (not shown) by default—it appears as you start to create an object, and disappears as you complete the object. You can use the Grid menu's Show Grid command to keep the Grid visible as you work.

The Edit Grid is rendered as a light-blue color mesh (Figure 3-5), and it is always aligned with a plane orthogonal to one of the axes in the global coordinate system. If you customize the global coordinate system, then the Edit Grid also follows the new coordinate system.

This section explains how you can specify and customize the Edit Grid. For sketching bodies using the Edit Grid, see "Creating Bodies" on page 6-5.

Figure 3-5
Edit Grid



Specifying the Edit Grid Orientation

The Edit Grid is always aligned with one of the x-, y-, or z-planes of the global coordinate system.

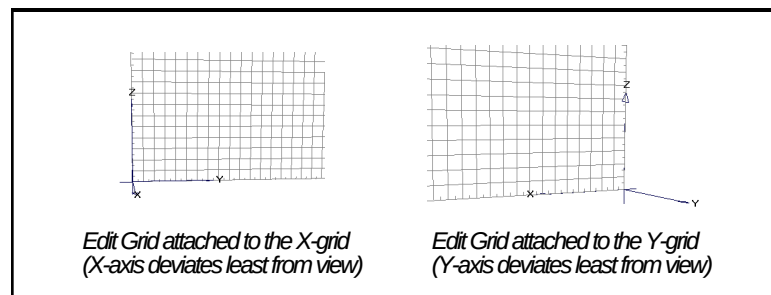
Edit Grid and Viewing Direction

By default, Working Model 3D automatically chooses one of the three planes that is most convenient for you. In particular, the Edit Grid is chosen so that its normal vector deviates *least* from the viewing direction. A *viewing direction* is the direction in which the three-dimensional model is projected on the viewing screen.

In this default mode, the Edit Grid shifts from one plane to another as you rotate the view using the Rotate View tool. Two instances shown in Figure 3-6 illustrate the cases where the Edit Grid is aligned with x- and y-axes, because these axes deviate least from the viewing direction.

Figure 3-6

Edit Grid and View Direction



Locking the Edit Grid

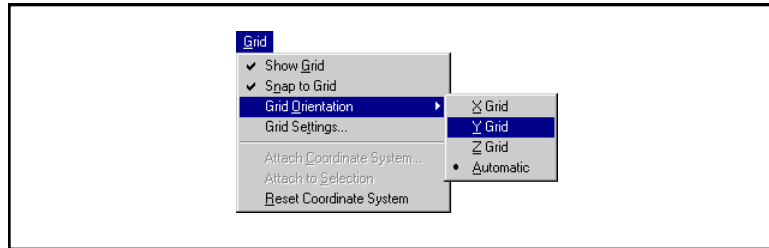
You can “lock” the Edit Grid to be one of the x-, y-, or z-planes permanently regardless of the viewing direction.

► **To lock or unlock the Edit Grid:**

1. **Choose Grid Orientation in the Grid menu.**

A submenu appears with three options: X Grid, Y Grid, Z Grid, and Automatic (Figure 3-7).

Figure 3-7
Grid Orientation Submenu



2. Choose X-Grid, Y-Grid, or Z-Grid from the submenu.

The x-grid means the grid perpendicular to the x-axis. Similarly, y-grid and z-grid correspond to the zx- and xy-planes, respectively.

► **To restore the automatic grid feature:**

1. Choose Automatic from the Grid Orientation submenu.

NOTE: The orientation of the Edit Grid has no effect on the direction in which gravity acts, or on the View menu's shortcut keys. These properties are based on the fixed, World coordinate system. See "Effects of Attaching Global Coordinate System" on page 3-3 and "Effects of Making Global Coordinate System Permanent" on page 3-4 for more information.

Properties of the Edit Grid

The Edit Grid, accessed with the Grid menu's Grid Settings command, has the following properties:

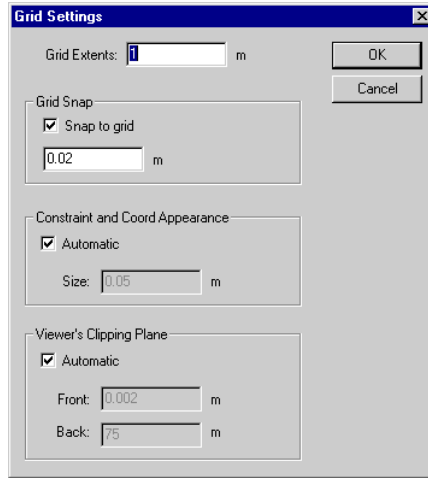
- Grid Extents
- Snap to Grid
- Grid Snap Size
- Constraint and Coord Appearance
- Viewer's Clipping Plane

► **To change these properties:**

1. Choose Grid Settings in the Grid menu.

The Grid Settings dialog appears (Figure 3-8).

Figure 3-8
Grid Settings Dialog



Each setting provides the following functionality:

Grid Extents

Grid Extents defines the width of the Edit Grid when displayed on the screen. Working Model 3D displays the Edit Grid as a square, as (0, 0, 0) as one of its bounding vertices. The Edit Grid is “spread” up to the value specified in Grid Extents. The quantity is associated with the current unit system (see “Numbers and Units” on page 5–1).

For example, if the number 20 is entered and the current distance unit is in meters, then the Edit Grid spreads from (0, 0, 0) to (20, 20, 0) (in meters), provided that the Edit Grid is locked to the z-plane (see “Specifying the Edit Grid Orientation” on page 3–8).

Working Model 3D automatically adjusts the zooming rate of your document depending on the specified magnitude of the Grid Extents setting, so that the entire Edit Grid fits in the window. For example, if you expanded the Grid Extents to 100 meters from the default of 1 meter, the Edit Grid expands in terms of its size, although the change is not as apparent on the document window, because Working Model 3D “steps back” the view so that the Edit Grid fits in the window.

Working Model 3D allows the Grid Extents of up to 1.0×10^9 meters (or equivalent distance in the current unit system).

Snap to Grid

The Snap to Grid feature forces the mouse pointer to “snap” at intervals set by the Grid Snap Size (see below). With this feature, you can re-size, create, or drag bodies and Coords precisely.

You can also toggle this feature directly by choosing **Snap to Grid** in the **Grid** menu.

Grid Snap Size

Grid Snap Size specifies the minimal interval to which objects snap on the Edit Grid. The quantity is associated with the current unit system (see “Numbers and Units” on page 5–1).

Working Model 3D ensures that the Grid Extents are at least ten times as large as the Grid Snap Size. If you enter a value in Grid Snap Size that is greater than one tenth of Grid Extents, Working Model 3D automatically increases Grid Extents immediately.

NOTE: The “coarseness” of the meshed lines drawn on the Edit Grid is *not* affected by Grid Snap Size. Working Model 3D automatically controls the density of these meshed lines and changes them according to the window size, zoom rate, and the size of the Grid Extents.

Constraint and Coord Appearance

The Constraint and Coord Appearance section allows you to choose whether the size of constraints and Coords in the drawing window will be set automatically or manually. If you prefer, you can set the size by entering a number in the Size box.

Viewer’s Clipping Plane

The Viewer’s Clipping Plane setting allows you to set whether or not it is Automatic, and to set the Front and Back measurements. These measurements will be set in whatever distance units you have chosen.

C H A P T E R 4

Manipulating Views

Working Model 3D provides an extensive array of interactive controls to manipulate how the model is presented on your screen. You can modify viewing directions, change rendering modes, and create multiple windows that can be controlled independently. This chapter shows how to manipulate the viewing options in Working Model 3D.

4.1 Changing Views

Working Model 3D provides several “preset views” in addition to interactive controls.

Head-On Views

You can quickly specify the viewing directions at any time by pressing a single key to access six “head-on” viewing directions as shown in the table below. The menu commands to invoke the preset view directions are also available in the Look At submenu in the View menu.

Views and Shortcut Keys

Name	Viewing Direction	Key Shortcut
Front View	From positive x-axis	X
Back View	From negative x-axis	Shift X
Right View	From positive y-axis	Y
Left View	From negative y-axis	Shift Y
Top View	From positive z-axis	Z
Bottom View	From negative z-axis	Shift Z

NOTE: The viewing directions are based on the World coordinate system and independent of the current Edit Grid orientation. For details on coordinate systems, please see "Attaching Coordinate System to Objects" on page 3–2.

Look at Selected Object

You can quickly view a specific object by selecting it in the Object Manager list. When an object is selected, you can click the right mouse button and choose Zoom to Selection in the content-sensitive menu. Alternatively, you can choose the Look At submenu in the View menu, then choose Selection.

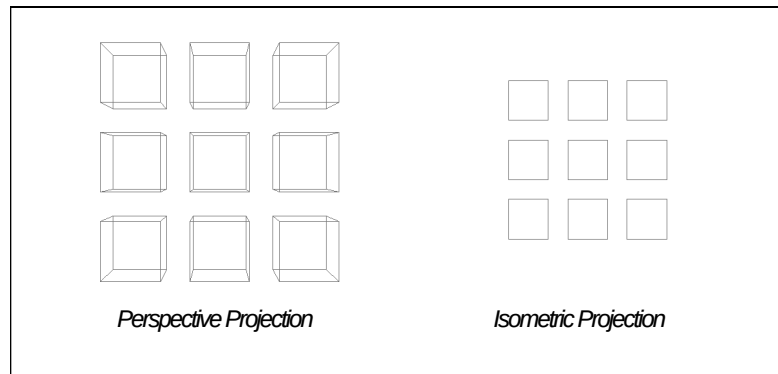
Perspective and Isometric Views

Working Model 3D provides two types of 3D projection modes: *perspective* and *isometric*. You can switch between these modes by:

- Choosing Perspective or Isometric in the View menu. The mode selected will show a bullet beside it.
- Clicking the Toggle Isometric tool on the View toolbar.
- Clicking the right mouse button in the background area and choosing Perspective or Isometric from the menu that appears.

Figure 4-1 shows a comparison between the two projection modes when you view an array of 9 cubes.

Figure 4-1
Perspective and Isometric Projections



Perspective Projection

In perspective mode, Working Model 3D presents the 3D world on the flat viewing screen by projecting the view from a single vanishing point. Visual depth information is richer in perspective mode than in isometric mode, but detecting the precise geometric relationship may be difficult. For example, parallel lines in the real world may not appear parallel in perspective projection.

Isometric Projection

Isometric projection¹ simply projects each object parallel to the viewing window. You can think of the isometric projection as a special case of perspective projection, where the vanishing point is located at infinity. Depth information is poor compared to the perspective projection, but you can easily verify geometric accuracy; parallel lines are projected to be always parallel in isometric projection.

Zooming

Zooming a view will enlarge the objects, coordinate axes, and the Edit Grid. In isometric mode, zooming simply changes the scale of the rendered model. In perspective mode, zooming is equivalent to moving the camera closer or further away from objects; therefore, incident angles between lines will vary as you zoom in or out.

¹. Also called *flat projection* or *orthographic projection*.

Three zooming operations are possible: zooming in and out, box zooming and viewing all. The tool used for zooming in and out is pictured below, alongside the description of this activity.

► **To change the view by zooming:**



1. Click the Zoom In/Out tool on the View toolbar, or press F3 on the keyboard.
2. Hold the mouse button down while the mouse pointer is in the document window.
3. While holding down the mouse button, move the mouse pointer **DOWN** to zoom in (increase the magnification) or move it **UP** to zoom out (decrease the magnification).

Working Model 3D zooms the entire view. The point where you clicked the mouse button remains fixed while the view is zoomed in and out.



Boxzooming allows you to select a portion of the document with a box, and then zoom to whatever you have encompassed within the box.



View All (or zoom to extents) allows you to see all the objects in your document; for more information, see “Viewing All Objects in the Document” on page 4–7.

Panning

Panning a view means simply moving the viewpoint sideways or up and down. In isometric rendering mode, panning will not affect the appearance of the model at all. In perspective mode, you will notice that the glancing angle changes as you pan the camera. See “Perspective and Isometric Views” on page 4–2 for more details.

► **To change the view by panning:**



1. Click the Pan tool on the View toolbar, or press F2 on the keyboard.
2. Hold the mouse button down while the mouse pointer is in the document window, and drag the mouse around to pan the view.

Rotating

Rotating the view gives the most flexibility in changing the view, giving you a complete 360° view from all directions.

► **To rotate the view:**



1. **Click the Rotate Around tool on the Edit toolbar, or press F4 on the keyboard.**

As you move the mouse pointer, a circle in dotted line appears on the screen. The circle is centered on the object closest to the pointer.

2. **Click in the document window, hold the mouse button, and drag the mouse to rotate the view.**

Working Model 3D rotates the view as you drag the mouse, keeping the line-of-sight fixed.

If you click the object within the circle, the view rotates around the object at the point you clicked. If you click outside the circle, the object(s) within the circle do a “barrel-roll”, snapping to the up position.

Undoing View Changes

Working Model 3D allows you to “undo” and “redo” the change in view angles.

► **To undo and redo changes in the view:**



1. **Open a Working Model 3D window, and rotate the view using the Rotate Around tool.**
2. **Zoom the view using the Zoom In/Out tool.**
3. **Choose Previous View in the View menu (as a shortcut, you can simply press P on your keyboard).**

Note that the viewing angle reverts back to the state prior to zooming.

4. **Choose Previous View again.**

The view angle reverts back again to the state prior to rotating.

5. **Choose Next View in the View menu (as a shortcut, you can simply press N on your keyboard).**

The viewing direction is now identical to the one set after step 1 above.

6. **Choose Next View again.**

You are back to the state after step 2.

You can thus press the P and N keys to go back and forth in the “stack” of the past view angles. In a sense, the Previous View command works much like “Undo”, while the Next View command works like “Redo”. Working Model 3D can remember up to ten past view angle changes.

Setting and Recalling Home View

After a series of rotations, zooms, and pans, you may find it difficult to remember where you started. You may have once set a “favorite” viewing direction and zoom rate best suited for presentation of your model. The “Home view” feature in Working Model 3D allows you to quickly set and go back to your favorite or most frequently used viewing direction.

► To set the Home view:

1. **Use the Pan, Zoom, or Rotate Camera tools to arrange the viewing direction that suits you.**
2. **Choose Set Home in the View menu.**

The current viewing direction and zoom rate are stored as the Home view.

► To recall the Home view:

1. **Choose Go Home in the View menu.**

The Home view is stored until you choose Set Home again. When you save the document, the Home view is stored in the document as well. Therefore, you can access the same Home view again when you open the document at a later time.

Viewing All Objects in the Document

As you navigate the model with the flexible camera tools, you may end up “being lost”; for example, you may zoom too much or pan in the area of the document in which no or a small number of parts are visible.

Working Model 3D provides a convenient “panic button” feature that allows you to view all the objects in the current document.

► ***To view everything in the current document:***

1. **Choose View All in the View menu, click the View All button on the toolbar, or press V on the keyboard.**

Working Model 3D immediately pans and scales the model so that all the objects (bodies, Coords, constraints, and Edit Grid) become visible on the screen.

4.2 Viewing a Model with Multiple Windows

Working Model 3D allows you to view a single model with multiple windows, and you can control the viewing direction and rendering mode of each window independently. Modifications on one window are immediately reflected on all other open windows.

Multiple windows provide you with more visual information of the model. For example, you can open one window with the wireframe mode to allow the complete, transparent view of the model, and you can leave another window open with the shaded mode as a realistic preview of the model (see “Rendering Modes” on page 4–8 for more information).

Creating and Deleting a Window

You must have a file open before you create multiple windows.

► ***To create a new window:***

1. **Choose New Window in the Window menu.**

A new window appears.

► ***To delete a window:***

1. **Click the button at the top-right corner of the window.**

NOTE: Closing all windows of a document closes the file itself. When you are about to close the last open window of a document, Working Model 3D prompts you to save changes to the file.

To close the file immediately, simply choose Close from the File menu.

► ***To organize multiple windows:***

1. **Choose Title in the Window menu.**

4.3 Rendering Modes

Working Model 3D features a full set of rendering functions to help you understand the model better, and also add substantial realism to the virtual prototype created on your screen.

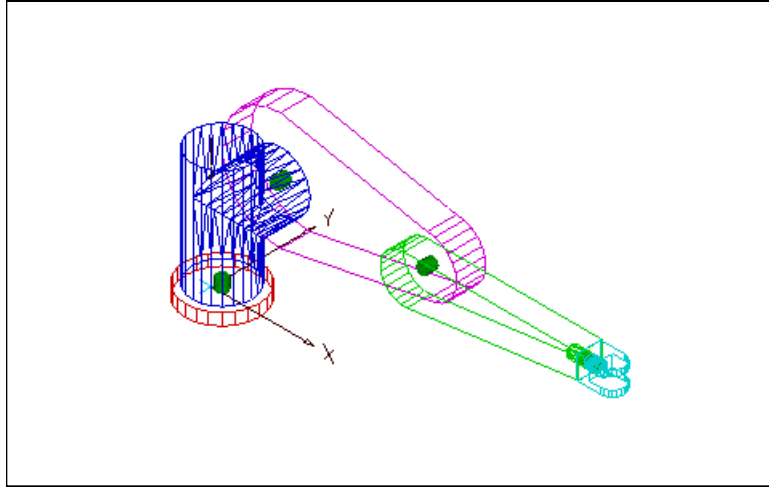
Working Model 3D can render the model in two modes—wireframe and shaded—the latter being the more realistic.

Wireframe Mode

Simple wireframe is the default mode and the fastest way to render models. The model consists of outlines, as shown in Figure 4-2. The constraint icons are visible in the wireframe mode (you can hide them

through the Appearance page of the Properties window for each constraint. See “Modifying Constraint Appearance and Name” on page 7–25 for more information).

Figure 4-2
Wireframe Mode

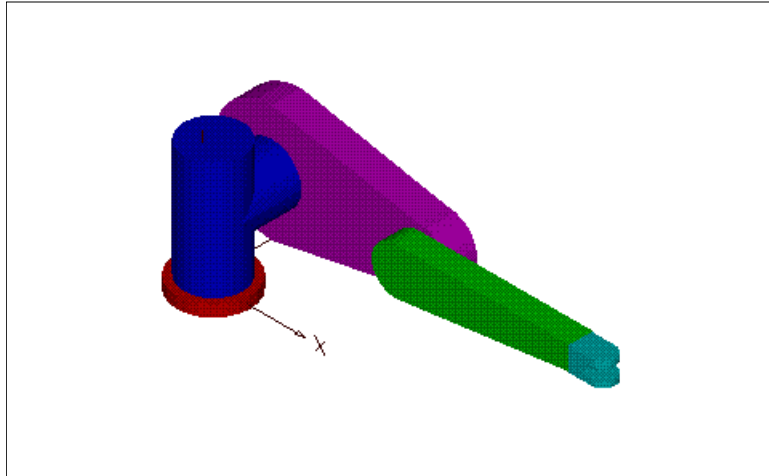


NOTE: For optimal performance, spheres appear as polyhedral approximations in the wireframe mode. However, the simulation engine of Working Model 3D treats them as true spheres, and computes their collisions and mass properties as such.

Shaded Mode

In the shaded mode, Working Model 3D computes the luminosity of each object using the locations of the light sources as well as surface properties of each object in the model. As shown in Figure 4-3, the shaded mode is more realistic than wireframe rendering.

Figure 4-3
Shaded Mode



For each object surface, Working Model 3D computes the relative angle with the light sources and determines the luminosity. For spheres, the appearance of the surface is computed so that it is smooth and continuous across the entire object. “Changing Appearance and Body Name” on page 6–26 provides more information on how to change the surface properties (rendered appearance) of each body.

In the shaded mode, constraint icons may not be visible; use the wireframe mode to view them.

***Notes on the Light Sources
Used in Shaded Mode***

Shaded mode uses two fixed light sources for optimum visual effects as follows:

- One “bright” light beams toward the object, as if from a flashlight you are holding pointing at the object from your point of view. The light moves as you change your viewpoint. The light is parallel and the distance from the light source is irrelevant in shading.
- The other light source, called *ambient* light, adds light to every surface, regardless of its surface’s normal vector.

Modifying the Background Color

By default, a Working Model 3D document has a white background. You can customize this background color (subject to your system's graphics limitations).

► ***To change the background color:***

1. **Choose Background Color in the View menu.**

The standard Windows color selection dialog appears.

2. **Choose a color, and click OK.**

The background color is changed accordingly.

CHAPTER 5

Workspace

5.1 Numbers and Units

Each new simulation document that you create defaults to the SI/metric unit system. The following units are the default settings:

Quantity	Default Units
Time	seconds
Distance	meters
Rotation	degrees
Mass	kilograms
Force	Newtons
Power	watts

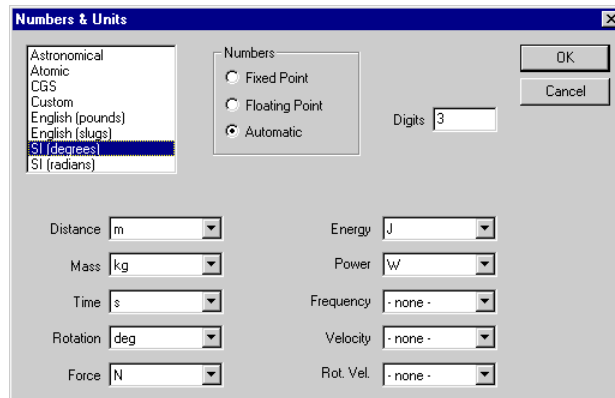
Working Model 3D performs all internal calculations in metric units. However, Working Model 3D can display the results in various unit systems. You are free to choose the unit system most appropriate to your simulation so you can enter and monitor values in the unit system.

► **To specify a unit system:**

1. **Choose Numbers & Units from the World menu.**

The Numbers & Units dialog appears as shown in Figure 5-1.

Figure 5-1
Numbers & Units Dialog



2. Select a general unit system from the Unit System list.
3. Choose how numbers will be displayed by clicking Fixed Point, Floating Point, or Automatic.

See “Numerical Display Format” on page 5–3 for information on these modes.

4. To customize units for individual quantities, change the settings in the individual unit settings pop-up menus.
5. Click OK to close the dialog.

The unit system change is applied to the document immediately.

The behavioral and quantitative aspects of your simulation remain unchanged throughout the unit system change, but the numbers representing the quantities are affected, reflecting the change. For example, suppose you changed the distance unit from meters to inches and the time unit from seconds to minutes. Then the part that was 1.5 meters long would become 59.055 inches (= 1.5 meters) long, and a 120-second period of a vibration system would become 2 minutes (= 120 seconds).

Numerical Display Format

The Numbers & Units dialog provides the following display options:

- **Fixed Point** format displays all numbers with a fixed number of digits to the right of the decimal point.
- **Floating Point** format displays all numbers in an exponential format of the form 1.23e4.
- **Automatic** lets Working Model 3D decide whether to display numbers in fixed or floating point formats, optimized for clarity.

The following table shows examples of how a number appears, depending on which option (fixed, floating, or automatic) you select.

Figure 5-2
Numerical Formats

Number	Fixed (1 digit)	Float (1 digit)	Automatic (1 digit)
0.000123	0.0	1.2e-4	0.0001
333.3333	333.3	3.3e2	3e+2

5.2 Gravity

Working Model 3D provides the following options for the gravitational force.

- Vertical Gravity
- None (no gravitational force)

Vertical Gravity

The *Vertical Gravity* option activates the gravitational force to act on every body in negative z-direction. You need to enable the Vertical Gravity option when you simulate a mechanism that needs to take gravity into account.

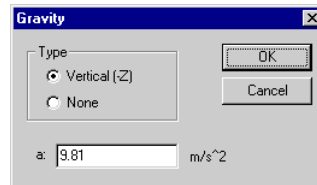
The force always acts in the negative z-direction of the World coordinate system, and the magnitude of the force is proportional to the mass of the body (i.e., $F = mg$). By default, Working Model 3D sets the gravitational acceleration to match that of the Earth. Since Working Model 3D allows you to change the value of the constant, you can simulate the lunar surface environment, for example, by decreasing the value of the gravitational constant.

► **To control the gravitational force:**

1. **Choose Gravity in the World menu.**

The Gravity dialog appears (Figure 5-3).

Figure 5-3
The Gravity Dialog



2. **Choose the type of gravitational force appropriate for your simulation.**

See above for an explanation of the force options.

3. **If appropriate, change the value of the constant.**
4. **Click OK.**

NOTE: If you want gravity to act in a direction other than the negative Z-axis, consider reorienting the global coordinate system. Please see Chapter 3 “Coordinate System” for more information.

5.3 Creating the Ground Surface for Simulation

When the vertical gravity option is active, you may find it convenient to have a ground surface of the simulation.

► **To create the ground surface for simulation:**

1. **Choose Ground Plane in the World menu.**

The ground surface appears, with its top surface aligned with the z plane.

Ground Plane is nothing more than an anchored box (see “Fixing a Body in the World” on page 6–24). It can collide with other bodies, and it has position, orientation, as well as all the other mass properties.

Working Model 3D generates the Ground Plane according to the following rules:

- The default orientation of the Ground Plane is aligned with the z-plane of the World coordinates. Even if the current coordinate system is attached to a body or a Coord, the Ground Plane always appears so that its top surface is aligned with the z-plane (see Figure 5-5).
- Working Model 3D automatically sizes the Ground Plane so that its surface area is slightly greater than the xy extent of all the bodies in space when they are projected onto the xy plane (see Figure 5-4). The thickness of the Ground Plane also depends on the xy extent of all the bodies in space.

Figure 5-4
How the Size of the Ground Plane Is Determined

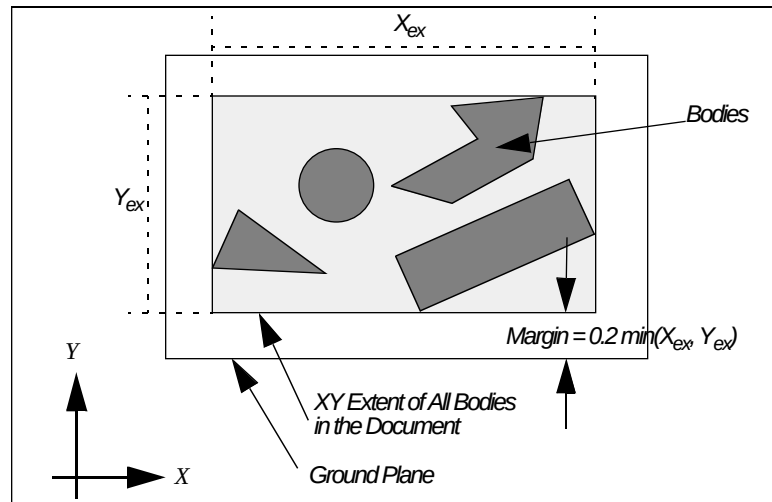
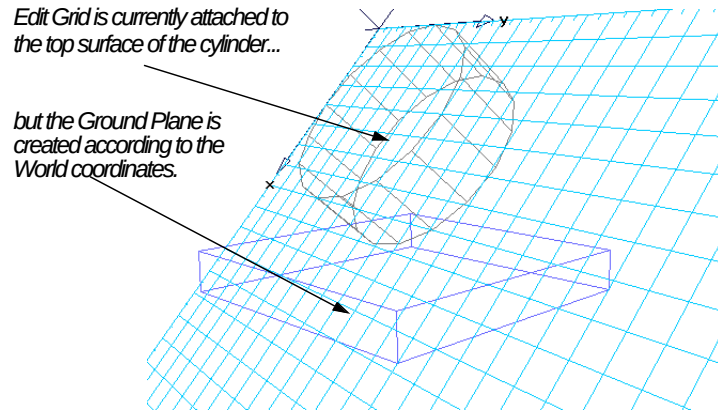


Figure 5-5
The Ground Plane and Current Coordinates



Resizing the Ground Plane Automatically

As you add more bodies in your model, you may wish to resize the ground plane so that it covers all the bodies. Since Working Model 3D sizes the Ground Plane whenever you activate it, you can simply turn the Ground Plane off and on again. The Plane reappears with the revised dimension to accommodate new bodies.

CHAPTER 6

Bodies

6.1 Introduction

What Is a Body?

In Working Model 3D, bodies are the primary elements of simulation models. When you construct a physical prototype, you would start with bars, wheels, and links, then put them together with constraints such as joints and springs. You would be following the very same pattern to construct a simulation model in Working Model 3D.

Depending on their geometry, a body in Working Model 3D can be classified as a *box*, *cylinder*, *sphere*, *extruded polygon*, or *polymesh*. Each body has a number of parameters that define how it behaves when you run a simulation. Some parameters are initially set to default values. For example, the density of every mass object is initially set to 1000 kg/m^3 .

When you run a Working Model 3D simulation, bodies behave according to the properties and constraints you specify. For example, bodies may fall under gravity, collide with other bodies, and/or freely slide along a slot.

Rigid Bodies

In dynamics terms, bodies in Working Model 3D are called *rigid*. Rigid bodies do not deform or break apart during simulation, even though a physical object (the subject of simulation) may actually do so. If you want to simulate *flexible* bodies—such as a bending beam or a rubber tire—you need to connect a series of rigid bodies with appropriate constraints to approximate the flexible body.

Putting Things Together

If you are ready to put together a model, jump to Chapter 7, “Assembling Models”, for instructions on how to put bodies and constraints together.

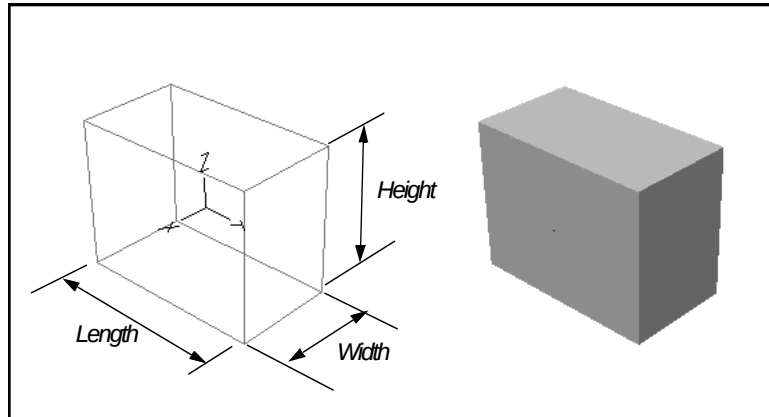
6.2 Body Types

This section shows all types of bodies available in Working Model 3D. Each section defines a body type and its dimension parameters. See the section on "Creating Bodies" on page 6-5 for more information.

Box

The geometry of a *box* can be completely described by its width, height, and depth. A box has six faces and each face is perpendicular to the adjoining face. Figure 6-1 is an example of a box.

Figure 6-1
Box



Cylinder

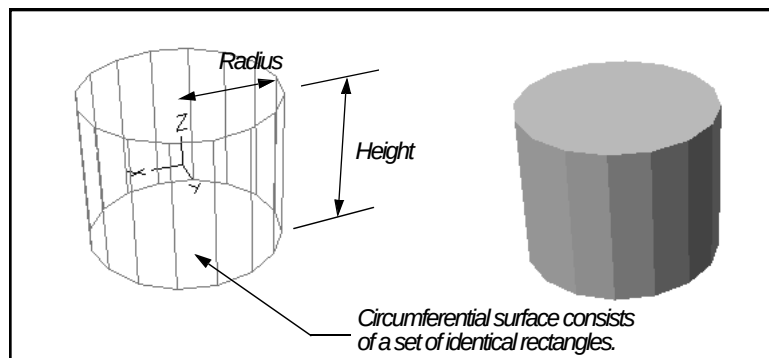
In Working Model 3D, cylinders are approximated as polyhedrons. The end faces are approximated as regular polygons instead of circles, and the circumferential surface consists of a set of identical rectangles. Therefore, the geometry of a cylinder can be completely described by the base radius, height, and the number of vertices used to approximate the end polygons. The base radius indicates the distance from the geometric center of the base polygon to its vertices; i.e., the radius draws a circle that bounds the base polygon.

The default cylinder in Working Model 3D has 16 sides (Figure 6-2); therefore, the end faces are regular polygons with 16 vertices, and the circumferential surface consists of 16 identical rectangles.

You can increase or decrease the degree of approximation through the Geometry page of the Properties window for the cylinder (see “Changing Body Geometry” on page 6–25).

Working Model 3D computes a cylinder’s integral properties (such as the mass moment of inertia) as a true cylinder. Also, when you attach Coords on a cylinder’s surface, the relative position of each Coord is calculated as if it were attached to the true cylinder’s surface. Consequently, a Coord may appear “floating” away from the cylinder’s polyhedral surface when rendered.

Figure 6-2
Cylinder



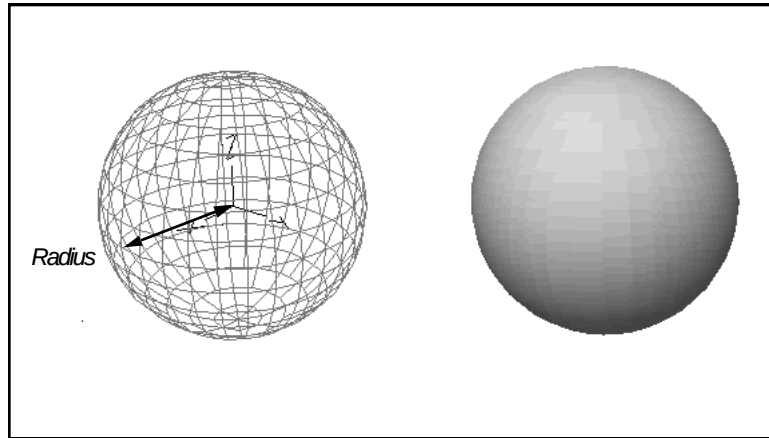
Sphere

The geometry of a *sphere* can be completely described by its radius.

For the optimum rendering performance and clarity, spheres are described by a polyhedral approximation in wireframe mode as shown in Figure 6-3.

On the other hand, Working Model 3D’s simulation engine treats all spheres as true spheres. For example, its integral properties (such as the mass moment of inertia) and collision detection are computed as true spheres.

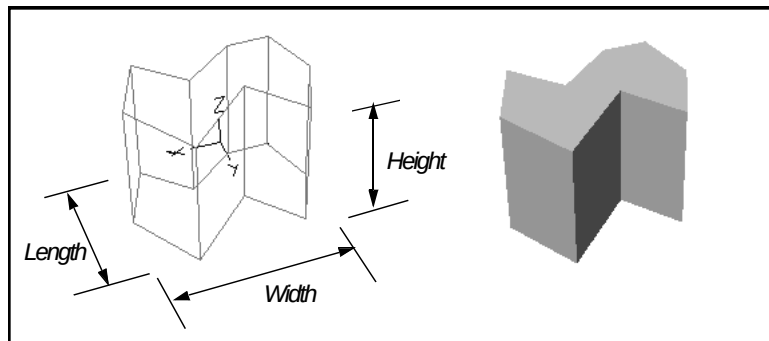
Figure 6-3
Sphere



Extruded Polygon

The geometry of an extruded polygon can be completely described by its base geometry (a polygon) and height.

Figure 6-4
Extruded Polygon

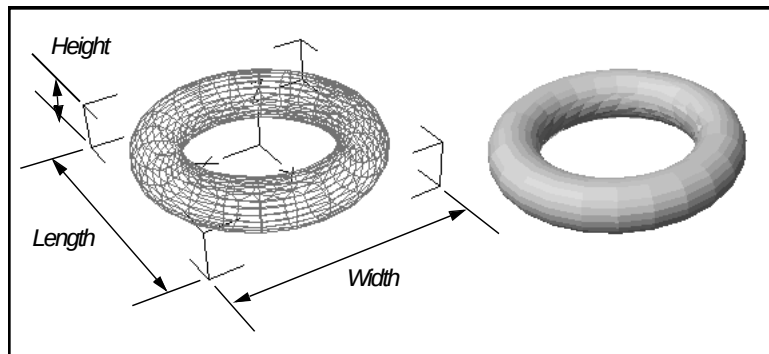


Working Model 3D allows you to sketch the base geometry in a fashion similar to a sketching program, followed by the extrusion. Once the construction is completed, you can edit the dimensions of the bounding box (length, width, and height) through the Geometry page of the Properties window.

Polymesh (Polyhedron)

A *polymesh* in Working Model 3D is what is commonly called a polyhedron. A polymesh is a solid with a closed boundary surface composed of planar polygons. In principle, a polymesh in Working Model 3D includes any closed-surface polyhedron with non-zero volume. Figure 6-5 shows an approximated (faceted) toroid as an example of a polymesh.

Figure 6-5
Polymesh Example (a Toroid)



NOTE: Working Model 3D does not provide a facility to create a polymesh directly. Instead, you can import the polymesh geometry from drawing files created by CAD programs. Once imported, polymeshes can be treated as any other rigid bodies available in Working Model 3D. Working Model 3D's simulation engine is also fully capable of computing necessary properties, including collision detections, for any polymesh bodies. Please see Chapter 11, "Import and Export", for more information.

The geometry of a polymesh can be completely described by a set of vertices and face descriptions (i.e., a set of vertices for each face). A box and extruded polygon can be considered as special cases of a polymesh.

6.3 Creating Bodies

Working Model 3D lets you manipulate 3D objects from your flat screen. In order to simplify the interface, Working Model 3D uses the Edit Grid as part of the drawing tool.

Edit Grid and Bodies

In Working Model 3D, you sketch and move bodies using the Edit Grid. The Edit Grid is rendered as a light-blue color mesh (see Figure 6-6), and it is always aligned with a plane orthogonal to one of the axes in the global coordinate system. If you customize the global coordinate system, then the Edit Grid also follows the new coordinate system. Please see "The Edit Grid" on page 3–7 for more information.

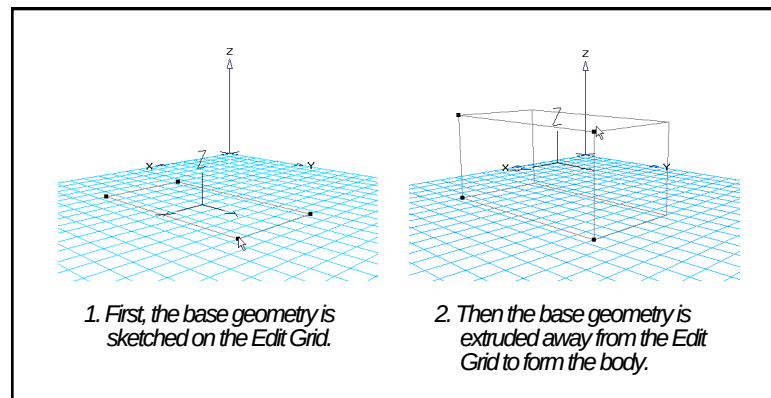
The Edit Grid is turned off by default, appears as you begin to draw a new body, and disappears as you complete the new body. If you want to always see the Edit Grid, choose Show Grid in the Grid menu.

Creating Bodies

The Edit Grid serves as a base plane to create a body. For example, as shown in Figure 6-6, a body's base geometry is first drawn on the Edit Grid, followed by the extrusion of the geometry.

Figure 6-6

Box Being Created on the Edit Grid



Dragging Objects

Working Model 3D also uses the Edit Grid when you move a body. When you use the Drag tool to drag a body, the body is translated *parallel* to the Edit Grid.

On the other hand, when you use the Z-Drag tool, a body is translated *perpendicular* to the Edit Grid.

The section "Manipulating Bodies" on page 6–13 provides more information on how you can manipulate bodies.

Creating a Body on the Edit Grid

The Edit Grid serves as a base plane to create a body. Working Model 3D allows you to construct a body as if you were drawing shapes in a drawing application.

Creating a Box



► To create a box using the Edit Grid:

1. Click the Box tool on the Sketch toolbar.

The Edit Grid appears.

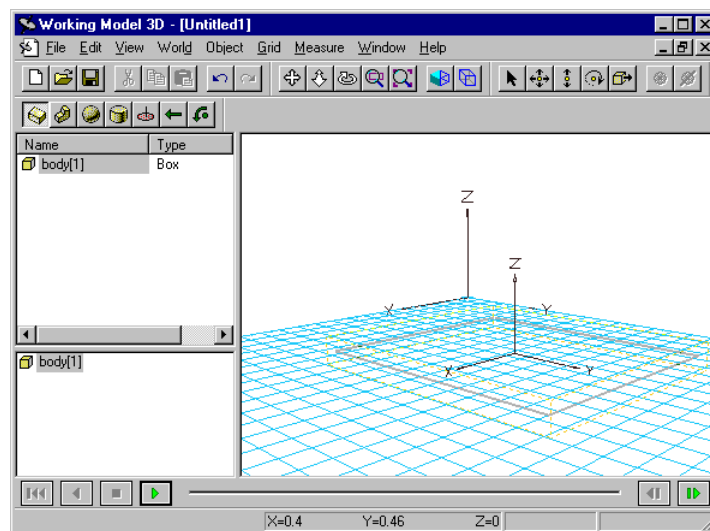
2. Click once on the Edit Grid.
3. Drag to extend the base rectangle.

Note that the first click point specifies a corner of the base rectangle of the box.

4. Click again when the base rectangle reaches the desired size.

The status bar at the bottom of the application window (Figure 6-7) shows the current dimension in the current unit system.

Figure 6-7
Status Bar Showing Dimensions



5. To extrude the base rectangle, drag away from the Edit Grid.

Again, the status bar shows the current height (distance from the Edit Grid) in the current unit system.

6. Click again when the box reaches the desired dimensions.

You can now modify the body properties, such as mass and geometry, using the Properties window.

Creating a Cylinder



► **To create a cylinder using the Edit Grid:**

1. Click the Cylinder tool on the Sketch toolbar.

The Edit Grid appears.

2. Click once on the Edit Grid.

3. Drag to extend the base circle.

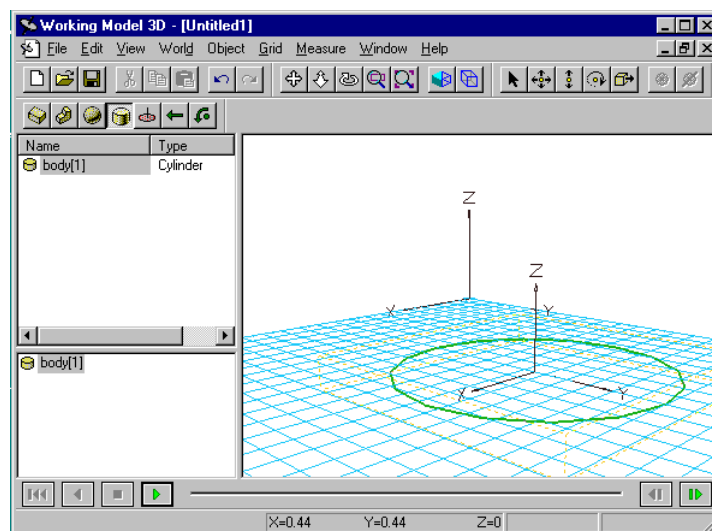
The first click point specifies the center of the base circle.

4. Click again when the base circle reaches the desired size.

The status bar at the bottom of the application window (Figure 6-8) shows the current dimension in the current unit system.

Figure 6-8

Status Bar Showing Dimensions



5. **To extrude the circle, drag away from the Edit Grid.**

Again, the status bar shows the current height (distance from the Edit Grid) in the current unit system.

6. **Click again when the cylinder reaches the desired height.**

You can now modify the body properties, such as mass and geometry, by double-clicking it in the document window or the Object Manager to open the Properties window.

Creating an Extruded Polygon



- **To create an extruded polygon using the Edit Grid:**

1. **Click the Extruded Polygon tool on the Sketch toolbar.**

The Edit Grid appears.

2. **Click once on the Edit Grid.**

The first mouse click marks one of the vertices of the base polygon.

3. **Drag to extend the line segment of the base polygon.**

4. **Repeat steps 2 and 3 above and continue creating the base polygon.**

NOTE: Do not let edges cross!

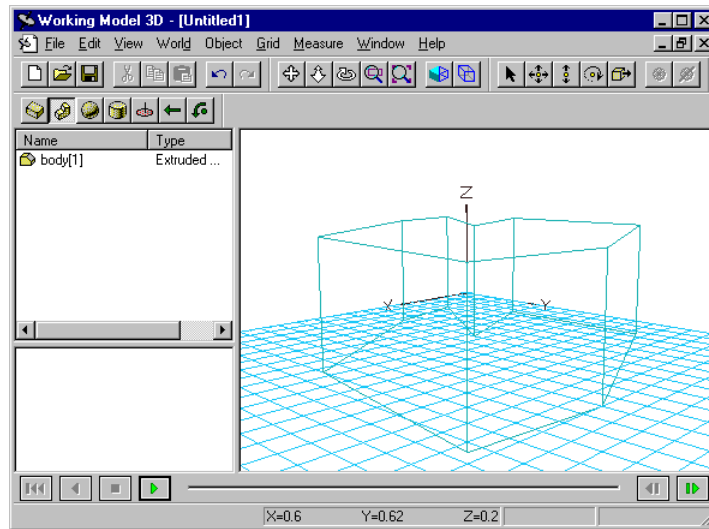
The status bar shows the dimension of the rectangle that bounds the base polygon.

5. **Press the space key to close the polygon.**

Alternatively, you can click the first point of the base polygon, or double-click.

6. **To extrude the base polygon, drag away from the Edit Grid.**

Figure 6-9
Status Bar Showing Dimensions



The status bar shows the current height (distance from the Edit Grid) in the current unit system.

7. **Click again when the extruded polygon attains the desired dimensions.**

NOTE: After you have sketched the body, you can double-click anywhere on it to display the Properties window, which you can use to set the Geometry and other characteristics. See “The Edit Menu” on page 2–3.

Using Grid Snap

You can use the Grid Snap feature to assist you in creating bodies precisely. To control the Grid Snap Size, choose **Grid Settings...** in the **Grid** menu. See “Properties of the Edit Grid” on page 3–9 for more information.

Importing CAD Geometry

Working Model 3D provides a tool to import CAD drawings via ACIS files. The imported geometry is treated as a polyhedron in the simulation (see “Polymesh (Polyhedron)” on page 6–5 for more information). For instructions on importing ACIS files, please see “Importing ACIS Files” on page 11–2.

Creating a Body Between Coords

While modeling a simple linkage mechanism, you may have precise definitions of the attachment points of the linkage members. In such cases, you can quickly construct a body between the two attachment points.

Working Model 3D allows you to specify a pair of distinct Coords and automatically create a cylinder between them, attaching both endpoints with spherical joints. Coords are point elements in Working Model 3D that serve as constraint endpoints. See Chapter 7, “Assembling Models” for more information on Coords.

► **To create a cylinder body between two Coords:**



1. **Click the Coord tool on the Sketch toolbar.**

The Edit Grid appears.

2. **Click on the Edit Grid to create a Coord.**
3. **Repeat to create a second Coord, so that you have two endpoints of the cylinder.**
4. **Select the two Coords.**

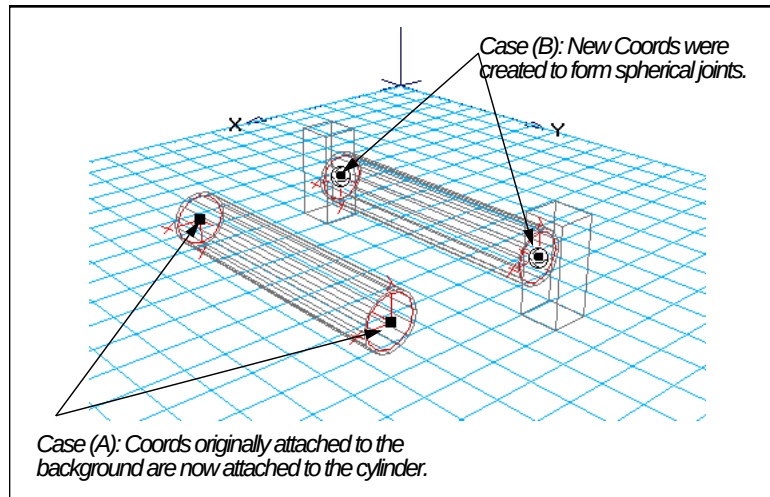
You can select two Coords simultaneously by selecting one Coord first, then holding down the Control (CTRL) key and selecting the other.

5. **Choose Create Connecting Body in the Object menu.**

A cylinder body appears between the two Coords. Figure 6-10 illustrates two cases that vary depending on the endpoint condition. See below for more information.

Figure 6-10

Creating a Cylinder Between Two Coords



Working Model 3D determines endpoint conditions of the cylinder based on the following criteria:

- If the endpoint Coord was originally attached to the background, the Coord will be attached to the new cylinder (as shown in Case (A) of Figure 6-10).
- If the Coord was originally attached to another body, a new Coord will be created with the identical position and orientation and attached to the new cylinder. Working Model 3D automatically connects the two Coords with a spherical joint (as shown in Case (B) of Figure 6-10).

In either case, the position and orientation of the original Coords remain intact.

The cylinder's height is the distance between the two Coords, while the default radius of the cylinder is one tenth of the height.

Once the body is created, you can change its geometry and/or change the constraint conditions. For example, you can change the two spherical joints to be a rigid joint and a revolute joint on each end. For information on changing the constraint type, see "Modifying Constraints" on page 7–18.

Creating a Ground Surface

Working Model 3D can automatically generate a “ground surface” that serves as the base unit for your simulation model. See "Creating the Ground Surface for Simulation" on page 5–4 for more information.

6.4 Manipulating Bodies

You can not only select, drag, rotate delete bodies, but you can also cut, copy, and paste bodies as you would in a drawing program. When you copy a body, Working Model 3D will name the new object name as “copy of”, followed by the original name.

Selecting Bodies

You can select bodies in several ways: point-select, box-select, or by selecting them in the Object Manager that runs along the left side of the document window.

Point-Select

You can select bodies by a simple mouse click. As you move the mouse and hover over objects in the document window, Working Model 3D draws a dotted box around objects. When you click the mouse, the object surrounded by the dotted box is selected (see “Edit Grid and Bodies” on page 6–6 for more information).

Once you select an object, the feedback you receive depends on the type of object selected:

- Bodies are drawn with a thicker lines (wireframe mode) or with an adorning box (shaded mode).
- Coords are displayed with their axes and a base square.
- Constraints are shown with a bounding box.
- Point-to-point constraints are drawn with thicker lines.

► To select a body:



1. Click the Select (Arrow), Drag, Drag-Z, or Rotate tool in the Edit Toolbar.

Each tool has different functions, but you can point-select an object using any of these tools.

2. **Locate the mouse pointer over the object you wish to select, and click it, or click the object you want to select in the Object Manager list.**

When you select a body (or constraint) in the Object Manager, the body itself is highlighted in the model, while the lower pane of the Object Manager shows all the bodies, constraints or coords attached to it.

The body reveals its local coordinate axes and the edges of the body thicken.

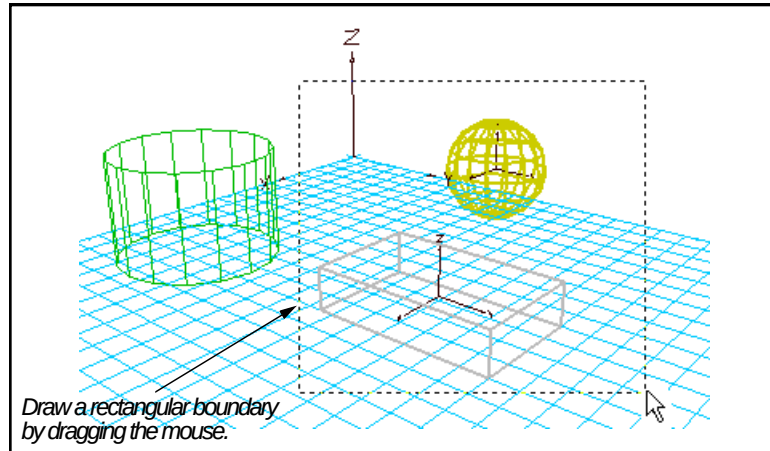
NOTES:

- If you wish to select an object that is completely hidden behind another, try changing the view first (see “Changing Views” on page 4–1) to bring the hidden object up front, or use the box-select method (discussed below). You can also select it in the Object Manager list.
- If you wish to select multiple bodies using point-select, successively select one object after another while holding down the Control (CTRL) key. You may find it easier to use box-select if the bodies are close together.

Box-Select

Select multiple bodies easily by using box-select to draw a rectangular bound on the screen. This procedure lets Working Model 3D select all bodies that fall completely within the bounds (see Figure 6-11). You may find the box-select method appropriate for very small objects that may be difficult to point-click.

Figure 6-11
Box-Selecting Multiple Bodies



► **To box-select bodies:**



1. Click the **Select** tool on the toolbar.
2. Holding the mouse button down, drag to draw a rectangular boundary. Resize the boundary to cover all the objects you wish to select.
3. Release the mouse button to complete the selection.

Alternatively, you can also use the **Drag**, **Drag-Z**, or **Rotate** tool to draw the rectangular bounds.

The box-select selects not only bodies but also other objects such as **Coords** and **constraints**. If you end up selecting more objects than you intended, selectively eliminate these objects from the selection by holding down the control (**CTRL**) key and point-selecting each object you want to omit.

Likewise, if you wish to add more objects to the ones currently selected, you can hold down the **CTRL** key, and either box-select or point-select additional objects.

NOTE: You can also quickly select and deselect objects in the Object Manager list.

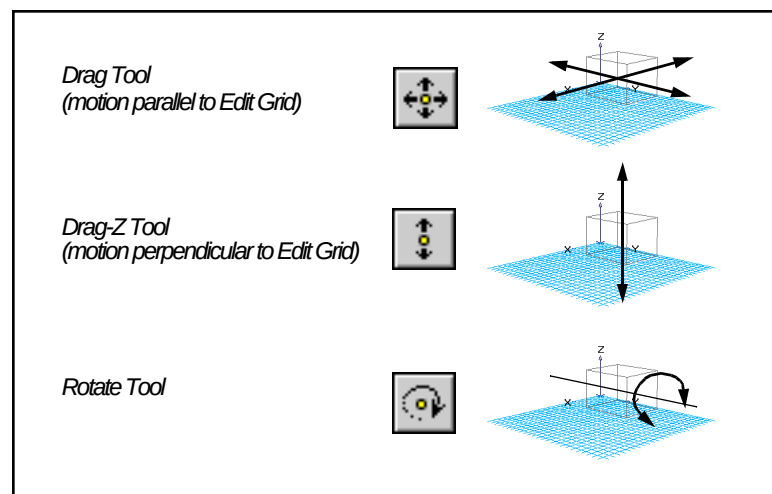
Moving and Rotating Bodies

You can move a body with the mouse in two ways—Translation (linear motion without rotation) or Rotation.

NOTE: If you wish to translate or rotate a body with a precise numerical specification, the Properties window provides precise control. Please refer to "Common Body Properties" on page 6–18 for more information.

Working Model 3D provides the following set of tools to accomplish these motions, as shown in Figure 6-12.

Figure 6-12
Motion Tools for Working Model 3D



Translating Bodies

In Working Model 3D, you can translate the body parallel or perpendicular to the Edit Grid of the current coordinate system (see "Attaching Global Coordinate System to a Body" on page 3–4).

► **To move a body parallel to the Edit Grid:**



1. Click the Drag tool in the Edit toolbar.
2. Click the body to be moved, and drag to the desired location.

The body moves parallel to the Edit Grid.

► **To move a body perpendicular to the Edit Grid:**



1. Click the Drag-Z tool in the Edit toolbar.
2. Click the body to be moved, and drag to the desired location.

The body moves perpendicular to the Edit Grid.

Rotating Bodies

You can rotate a body with one of its coordinate axes fixed.

When you select a body with the Rotate tool, Working Model 3D determines the axis of rotation depending on where you clicked. The body will rotate around an axis that is perpendicular to the selected face.



► **To rotate a body:**

1. **Click the Rotate tool in the Edit toolbar.**

Be careful not to select the Rotate Around tool in the View toolbar.

2. **Click the desired spot on one of the body's faces.**

The crosshair indicator appears to show the plane of rotation and the axis.

3. **Drag to rotate.**

The body rotates on the axis, around the crosshair.

NOTE: See the status bar for information about which bodies are selected by the circle, or those at the center of the rotation.

Deleting Bodies

When you delete a body, the Coords that were attached to the body retain its global position, but the Coords are attached to the background thereafter.

► **To delete a body:**

1. **Select the body by clicking on it.**
2. **Press the Delete key.**

The body disappears, and any Coords attached to the body are attached to the background.

6.5 Common Body Properties

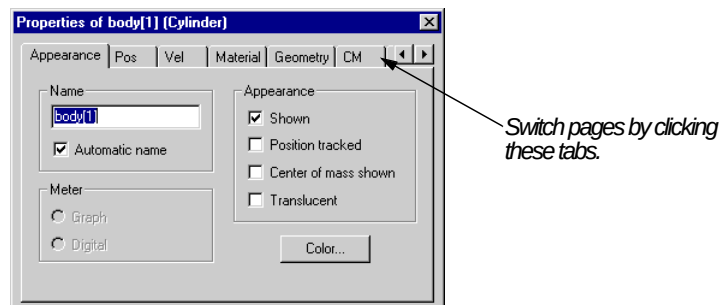
This section describes properties that are common to all the body types available in Working Model 3D. See "Body Types" on page 6–2 for an overview of available body types in Working Model 3D.

The Properties Window

The Properties window provides multiple pages that provide easy access to a variety of controls for each body. You can switch from one page to another with the tabs at the top of the window.

- The **Appearance** page provides control over the body's color, as well as giving you the option to name the body or to have them named automatically. Visibility is controlled here, with options to Show, Show the Center of Mass, and Track History. Finally, Meter controls allow you to choose Graph or Digital.
- The **Position** page provides control over the body's initial position and orientation.
- The **Velocity** page provides control over the body's initial velocities (linear and angular).
- The **Material** page provides control over the body's mass, density, and volume, as well as allowing you to set the coefficient of restitution and the coefficient of friction.
- The **Geometry** page provides control over the body's width, length and height.
- The **Centrality** page provides control over the body's moment of inertia
- The **CM** page provides control over the center of mass offset, allowing you to set parameters for X, Y and Z, as well as whether or not to show the center of mass.

Figure 6-13
Properties Window



► **To open a Properties window for a body:**

1. **Select the body.**
2. **Either (a) double-click the body in the document window, (b) double-click the body in the Object Manager list, or (c) choose Properties from the Edit menu.**
3. **Click the tab of the properties you want to set.**

Note that the Properties window is a *floating* window. You can place it anywhere on your screen, and the window stays frontmost and remains open (until you close it) while you are using Working Model 3D.

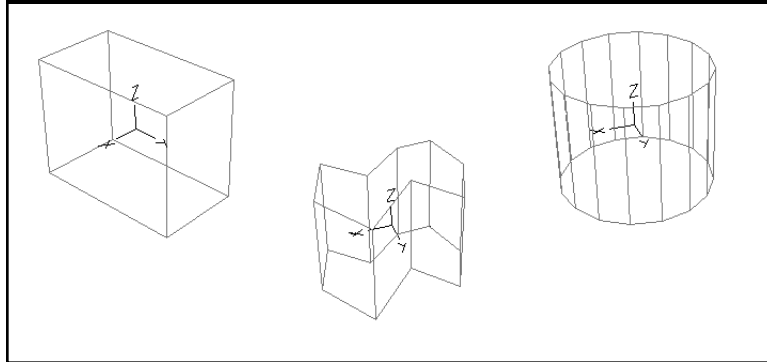
Local Coordinates of a Body

Each body in Working Model 3D has its own local coordinate system. The origin of the frame is defined at the geometric center of the body, and the frame has x-, y-, and z-axes. The coordinate axes appear when the body is selected (Figure 6-14).

NOTE: You can display the coordinate axes permanently by enabling the “Center of mass shown” option on the Appearance page; see “Changing Appearance and Body Name” on page 6–26.

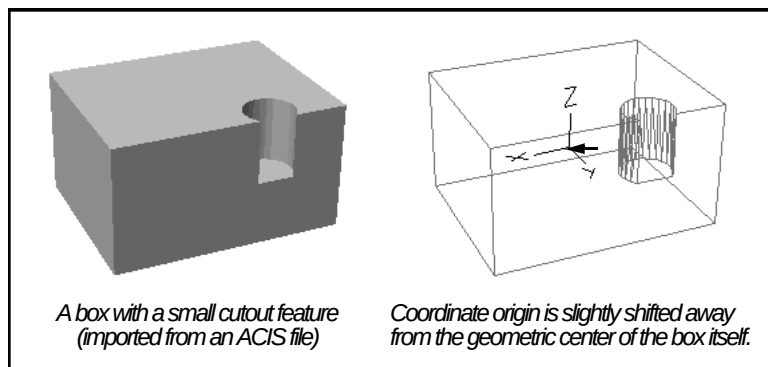
You will likely find the local coordinates to be a convenient (and important) property when you define positions of points or other bodies. For example, Working Model 3D expresses the angular velocity of a body in the axes of the local coordinate system.

Figure 6-14
Local Coordinate System for a
Body



Working Model 3D determines the coordinate origin to be the geometric center, including imported polyhedra (see "Importing ACIS Files" on page 11–2). For example, even if a body almost appears to be a true right-angle parallelepiped (known as a “box” in Working Model 3D), the body’s coordinate origin is shifted—due to the cutout feature—from what would be otherwise the geometric center of the box (see Figure 6-15).

Figure 6-15
Coordinate Origin for “Irregular”
Bodies



Changing Position and Orientation

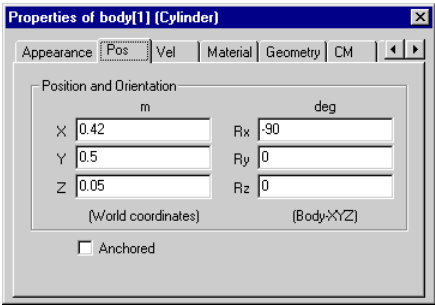
The Properties window provides access to the position and orientation of the body.

► To change the position and orientation of a body:

1. Open the Properties window and select the Position (Pos) tab.

The page appears as shown in Figure .

Figure 6-16
Properties Window (Position Page)



2. Enter the desired values.

Each control is associated with the current unit system.

The following table shows the relationships between a particular property and the coordinate system on which the measurement is based.

Position Page Properties and Descriptions

Property	Descriptions
Position (X, Y, Z)	Position of the body's geometric center (the origin of the local coordinate system) expressed in global coordinates
Orientation (Rx, Ry, Rz)	Orientation of the body frame expressed in body XYZ angles (see Figure 6-18)

Changing Velocity

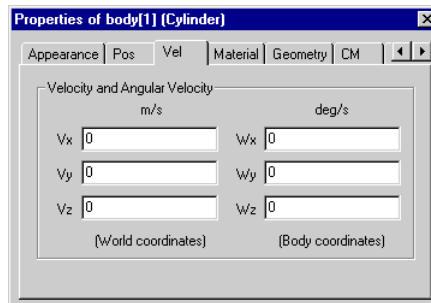
The Properties window provides access to the velocity of the body.

► **To change the velocity of a body:**

1. Open the Properties window and select the Velocity (Vel) tab.

The page appears as shown in Figure 6-17.

Figure 6-17
Properties Window (Velocity Page)



2. Enter the desired values.

Each control is associated with the current unit system.

The following table shows the relationships between a particular property and the coordinate system on which the measurement is based.

Velocity Properties and Coordinate System

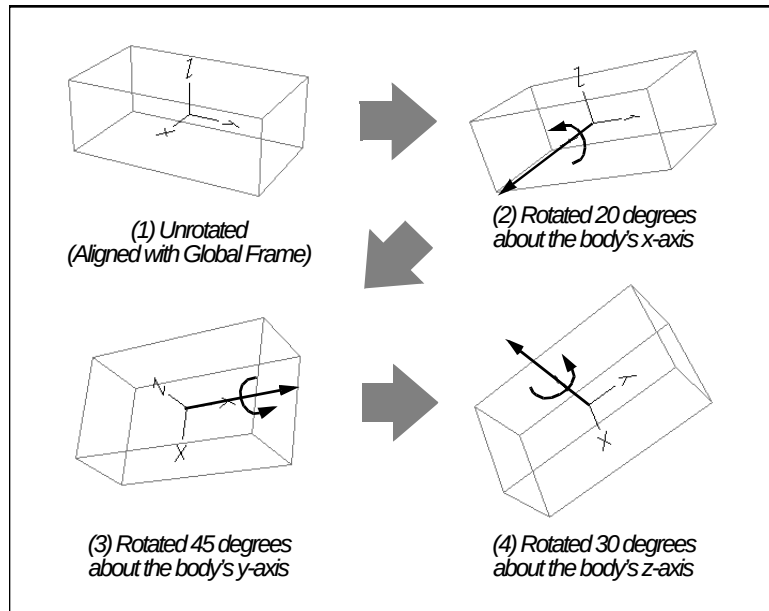
Property	Descriptions
Linear Velocity (V_x , V_y , V_z)	Linear velocity of the body's geometric center (the origin of the local coordinate system) expressed in global coordinates
Angular Velocity (w_x , w_y , w_z)	Angular velocity of the body frame, expressed in the local coordinate axes

XYZ Angles

Working Model 3D describes the orientation of a body in *body XYZ angles*. The XYZ angles express the orientation of a frame in a series of three rotations. Each rotation occurs around the x-, y-, and z-axes *of the body's coordinate system* in sequence. (Note that, in the body XYZ angle expression, the rotations does *not* take place with the fixed global coordinate axis.)

As shown in Figure 6-18, the order of rotations plays a role in determining the goal configuration. Other body angle expressions (such as body ZYX angle) result in a different goal configuration, even if the parameters appear to be identical.

Figure 6-18
*Orientation Expressed in XYZ
 Angles*



Fixing a Body in the World

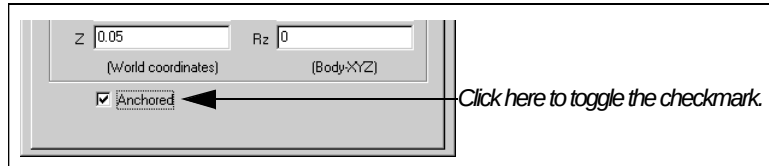
By default, Working Model 3D applies gravity to all bodies in the negative z-direction (see "Gravity" on page 5–3). Therefore, without having a body that is explicitly fixed or constrained in space, your entire model would fall.

Working Model 3D provides a simple option for every body in the model to be “anchored.” Anchored bodies remain absolutely fixed in space, completely immune to gravity or any other constraint forces, torques and collisions. If another free (unanchored) body collides with an anchored body, the free body will bounce back, provided that the coefficient of restitution is set to be greater than 0 (elastic collision).

► **To anchor a body:**

1. Select the body, open the Properties window, and select the Position tab.
2. Click Anchored (see Figure 6-19).

Figure 6-19
Anchored Checkbox



By default, a body is not anchored (except the Ground Plane; see "Creating the Ground Surface for Simulation" on page 5–4).

Changing Body Geometry

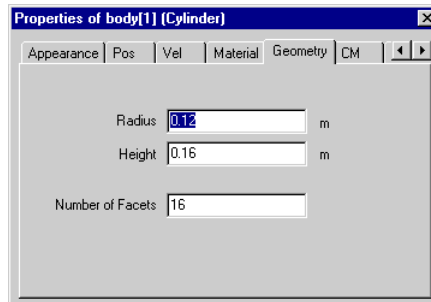
Body geometry is specific to each body type. Please refer to "Body Types" on page 6–2 for more information.

► To change the body geometry:

1. Open the Properties window and select the Geometry page.

The window appears as shown in Figure 6-20.

Figure 6-20
Properties Window (Geometry
Page) for a Cylinder



2. Enter the desired values for the appropriate control.

Depending on the body type, you can edit the following parameters in the Geometry page.

Body Types and Editable Properties in the Geometry Panel

Body Type	Editable Properties in the Geometry Page
Box	Length, Width, Height
Sphere	Radius
Cylinder	Radius, Height, Number of Sides (for approximation)
Extruded Polygon	Height, Length, Width (of the bounding box)
Polymesh (Polyhedron)	Height, Length, Width (of the bounding box)

Changing Appearance and Body Name

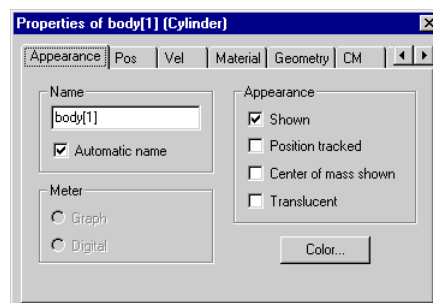
You can control the color, name, and surface appearance (parameters used for rendering in the shaded mode) through the Properties window as well.

► **To change the appearance:**

1. Select the body.
2. Open the Properties window and select the Appearance tab.

The window appears as shown in Figure 6-21.

Figure 6-21
Properties Window (Appearance Page)



3. Enter the desired values for the appropriate control.

See below for the descriptions on each control.

The Appearance page of the Properties window controls name, visibility and color.

Name

By default, Working Model 3D provides the name of the body as `body[n]`, where n is a unique integer. This default name is actually used as a variable in Working Model 3D's formula language. For details on the use of formula language, see Chapter 12, "Bodies".

The name appears in the status bar (see "Status Bar" on page 2-16) as well as in the Create Constraint dialog.

You can customize the body name to identify bodies more easily.

► **To change the body name:**

1. **Open the Properties window and select the Appearance tab.**
2. **Highlight the object name text box in the window, and type the custom name.**
3. **Press Enter.**

From then on, the custom name appears in the status bar and selection lists.

► **To switch back to the default object name:**

1. **Open the Properties window and select the Appearance tab.**
2. **Select the body.**
3. **In the Name section of the window, click the Automatic Name checkbox.**

From then on, the default name (`body[n]`) appears in the status bar and selection lists.

Visibility Options

The visibility options provide controls to:

- show or hide the body
- show or hide the center-of-mass display
- track the history

The tracking history option draws the trails of the last 150 frames of the simulation (the older frames get erased).

- make the body translucent when it's in shaded mode

When a body is translucent, you can see other bodies behind it.

NOTE: The Show checkbox only controls the visual presence of the body. Whether the body is visible or hidden, Working Model 3D takes the body into account while computing the simulation.

► **To set the Visibility options:**

1. Double-click on the body to display the Properties window.
2. Select the Appearance tab.
3. Click the appropriate checkbox under Visibility to show or hide the body; to show or hide the center of mass display; to track the history; or to make the object translucent when it is in shaded mode.

NOTE: **Color** controls the color of the body.

Changing Mass Properties (Body Properties)

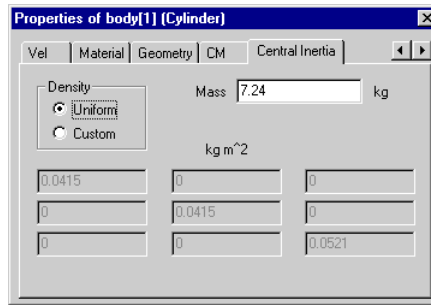
You can control the mass properties of a body through the Properties window.

► **To change the mass properties of a body:**

1. Open the Properties window and select the Central Inertia tab.

The page appears as shown in Figure 6-22.

Figure 6-22
Properties Window (Central Inertia Page)



2. Edit the desired fields.

The Central Inertia page provides the following controls:

- Mass moment of inertia matrix with respect to the body's center of mass (Central Inertia)
- Mass and density (uniform or custom)

Central Inertia

By default, Working Model 3D computes the central inertia matrix of a body assuming that the mass distribution is uniform. By clicking the Custom radio button, you can control all parameters in the matrix (Working Model 3D ensures that the matrix is symmetric).

Each edit box corresponds to the central inertia matrix as follows:

$$\begin{bmatrix} I_{xx} & I_{xy} & I_{xz} \\ I_{xy} & I_{yy} & I_{yz} \\ I_{xz} & I_{yz} & I_{zz} \end{bmatrix}$$

where the x-, y-, and z-axes refer to those of the body's local coordinate system.

Working Model 3D assumes the uniform density to determine the default values of the center of mass (CM) and the central inertia. If you specify a custom CM offset and/or the central inertia, the assumption will cease to hold, and Working Model 3D will be unable to compute the values automatically. To specify the precise mass property of a body with a non-uniform density, you need to specify CM offset and central inertia manually.

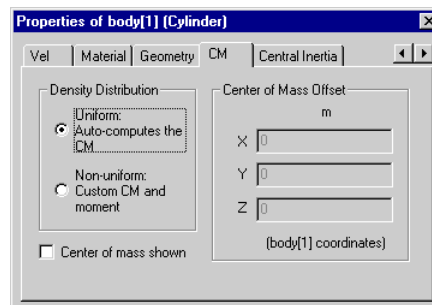
NOTE: It is best to leave the CI and CM settings set to Uniform unless you have advanced skills in this area. If you do choose to change the CM offset manually, the dialog changes to Custom Mode. You must then change the values in the CI matrix (computing these yourself) to ensure that your model still makes physical sense. See the section below for more about changing the Center of Mass offset properties.

► **To change the Center of Mass Offset properties:**

1. Open the Properties window and select the CM tab.

The page appears as shown in Figure 6-23.

Figure 6-23
Properties Window (CM Page)



2. Edit the desired fields.

The CM page provides the following controls:

- Density Distribution—Uniform (Auto-computes the CM)
—Non-Uniform (Custom CM and moment)
- Center of Mass Offset: values for x, y and z
- To show or hide the Center of mass

Changing Material Properties

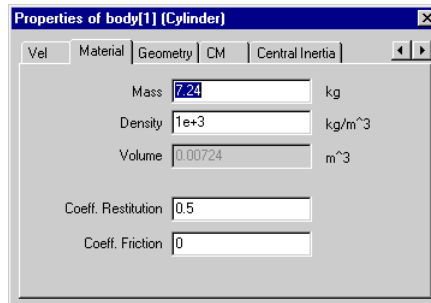
You can control the material properties of a body using the Material tab in the Properties window.

► **To change the material properties of a body:**

1. Open the Properties window and select the Material tab.

The page appears as shown in Figure 6-24.

Figure 6-24
Properties Window (Material Page)



2. Edit the desired properties.

The Material page provides the following controls:

- Mass
- Density
- Volume
- Coefficient of restitution
- Coefficient of friction

Mass and Density

You can control either the mass or the density of the body, and Working Model 3D automatically computes the other property based on the current geometry.

If you modify the geometry of the body by resizing it, Working Model 3D preserves the last mass or density you entered.

Coefficient of Restitution

In mechanics, the coefficient of restitution is really a property of a collision and not of a body. Precisely speaking, the coefficient of restitution is defined as the magnitude ratio of the relative velocities of the colliding bodies immediately before and after the collision.

For example, if the coefficient of restitution is 0.0, the difference in velocities of the two bodies after a collision will be zero—i.e., they will stick together. An elasticity of 1.0 (perfectly elastic collision) means that the difference in velocities after the collision will be the same as before, except that they are in the opposite direction. For more information, see Appendix A, “Simulating Collisions”.

In Working Model 3D, each body is assigned a constant as “coefficient of restitution” in the Properties window. When a collision occurs during the simulation, the coefficient of restitution of a collision is defined as the lower of the two coefficients given to the bodies involved in the collision. Thus, if one body has the coefficient as 0.2, and another body has the coefficient as 0.8, Working Model 3D computes the resulting collision with a coefficient of restitution of 0.2.

You can change the coefficient of restitution of one or more bodies by selecting multiple bodies and entering a value directly into the Properties window. The change will take effect for all the bodies selected.

Coefficient of Friction

Strictly speaking, a friction coefficient represents a property of interaction between two objects. In Working Model 3D, each body is assigned a friction coefficient. The coefficients of friction between two objects are determined by taking the lower of the two coefficients given to the bodies in contact. For example, the coefficient of friction between a body with a value of 0.05 and another body with a value of 0.3 would be 0.05.

You can change the coefficient of friction of one or more bodies by selecting multiple bodies and entering a value directly into the Material page of the Properties window. The change will take effect for all the bodies selected.

For more information, refer to Appendix A, “Simulating Friction”.

6.6 Specifying Collisions Among Bodies

By default, all bodies in Working Model 3D are meant to penetrate one another. You can override this setting and specify exactly which body can collide with another.

NOTE: You can improve the simulation performance by “turning off” as many possible collisions as possible since Working Model 3D spends a significant amount of time checking for collisions.

Working Model 3D allows flexible ways to specify the collision specification:

- You can select a set of bodies and designate them to penetrate all other bodies.
- You can select a set of bodies and designate them to collide (or penetrate) among one another.
- Select a single body and set it to collide with or penetrate all other bodies.

► **To specify collisions among bodies:**

1. Select a set of bodies.

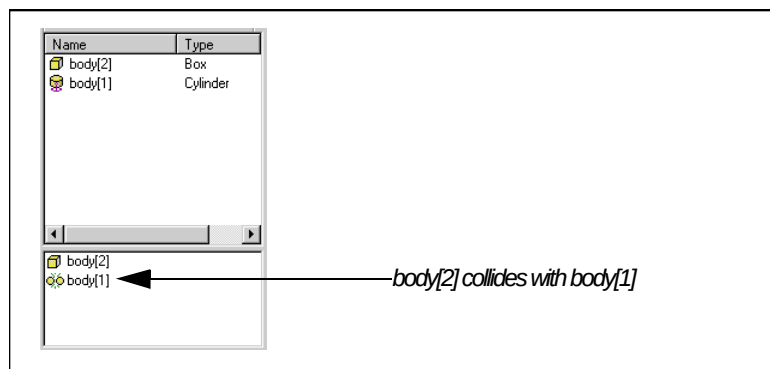
Use box-select, or select individual bodies in succession with the CTRL key down.

2. Choose one of the options—Collide, Penetrate, or Penetrates All—in the Object menu.

You can also click the right-mouse button over the selected bodies to display a pop-up menu, and select an option from this menu.

The bodies that collide with the selected body are displayed in the Object Manager's Connections list as shown in Figure 6-25.

Figure 6-25
Object Manager Connections List



The menu options have the effects shown in the following table:

Collision Options and Behaviors

Option	Behavior
Collide	The selected set of bodies will collide with one another when they come into contact. The menu option is not available if you select a single body.
Penetrate	The selected set of bodies will penetrate one another when they come into contact. The menu option is not available if you select a single body.
Penetrates All	The selected set of bodies will penetrate not only one another but also all the other bodies in the model when they come into contact.

Collision Model

Working Model 3D uses an impulse momentum method to compute collisions among bodies. As discussed in “Changing Mass Properties (Body Properties)” on page 6–28, the coefficient of restitution of a collision is taken as the least of the coefficients assigned to the bodies in contact. See Appendix A, “Simulating Collisions”, for more information.

C H A P T E R 7

Assembling Models

This chapter describes how to build models by putting together bodies with constraints. You will also learn how to manipulate individual bodies and assemblies consisting of multiple bodies; for example, selecting an assembly and rotating it.

For more information, you may wish to consult the following chapters.

- **Chapter 4, “Manipulating Views”** provides you with instructions on controlling how the model is presented on the screen.
- **Chapter 5, “Workspace”** shows how to set up the simulation environment, such as the unit system and gravity control.
- **Chapter 6, “Bodies”** describes how to create bodies and modify their properties.
- **Chapter 8, “Constraints Reference”** provides in-depth information on constraints, classified by their types.

7.1 Putting a Model Together: An Overview

In Working Model 3D, you would typically assemble a model in the following order:

1. **Create bodies.**
2. **Attach Coords (constraint attachment points) to bodies.**
3. **Create constraints between pairs of Coords.**
4. **Repeat the above steps, and build the model progressively.**

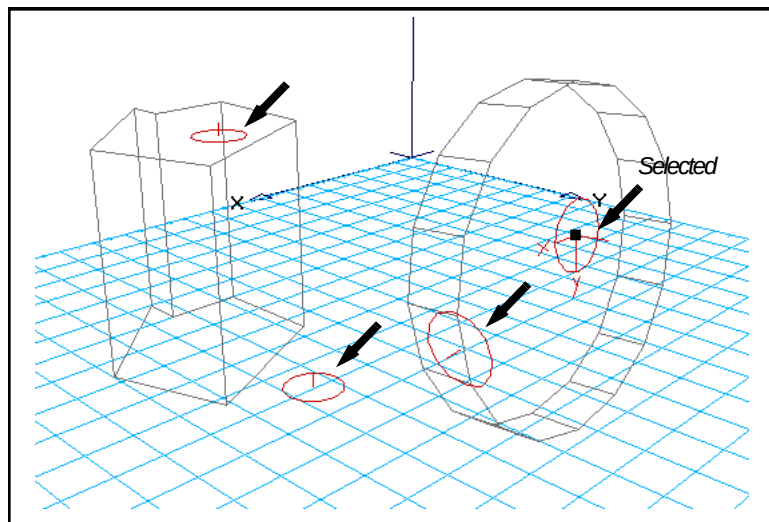
Other chapters of this manual explain how to create bodies and arrange them. Concepts involved in creating constraints that bind bodies warrant an overview.

Coords and Constraints

Coords

Working Model 3D creates constraints between objects called *Coords*. Coords are point elements with a local coordinate system attached. Therefore, a Coord has not only position but also orientation. Figure 7-1 shows a few examples of Coords. Note that each Coord, when selected, reveals a set of coordinate axes that indicates its orientation; see the selected Coord in Figure 7-1. Otherwise, a Coord shows a small red line segment representing its z-axis.

Figure 7-1
Examples of Coords



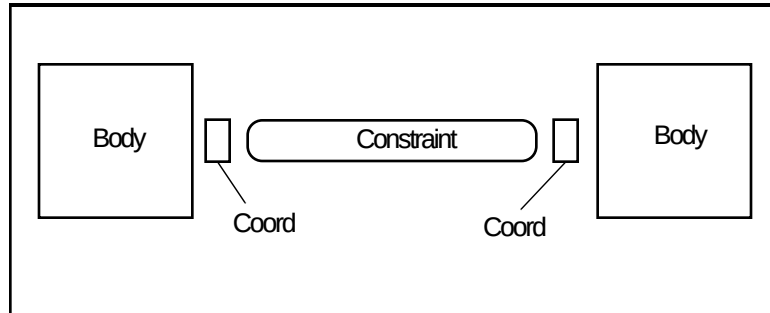
Constraints

In Working Model 3D, a constraint is a representation of certain rules that apply forces and/or torques on bodies. Working Model 3D constructs a constraint from a pair of Coords. In other words, a Coord serves as a medium—like glue—between a body and a constraint. Furthermore, Working Model 3D constructs a constraint based on the position and/or orientation of the two Coords. Therefore, in order to create a constraint, you must first create and attach Coords to bodies.

Figure 7-2 illustrates this concept. Note that you cannot attach a constraint directly to a body.

Figure 7-2

Concept: Construction of a Constraint

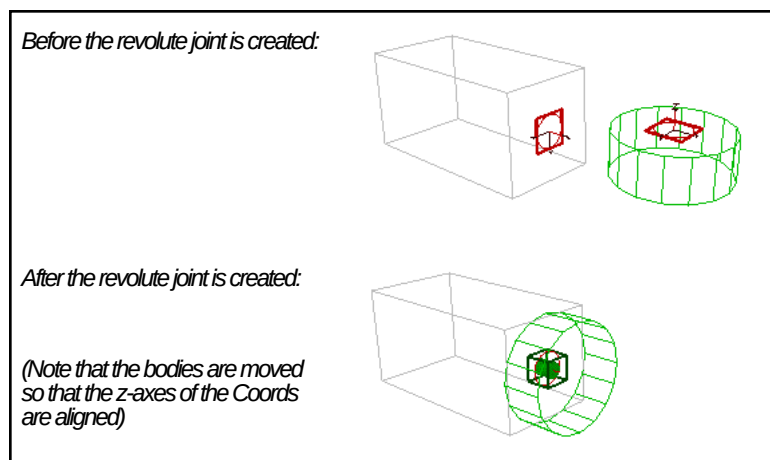


The orientation of the Coords plays a crucial role in defining the property of a constraint for the following reasons:

- In many cases, Working Model 3D creates a constraint by aligning the coordinate axes of the two Coords. For example, when you create a revolute joint (which only allows one degree of freedom) between two Coords, Working Model 3D reorients either Coord—hence the attached body also—to align the axes of two Coords. Figure 7-3 illustrates this principle. The reorientation will not occur for constraints such as springs, which allow free rotations about all axes at their endpoints.
- Working Model 3D reports the degrees of freedom allowed by a constraint by referring to its Coord axes. For example, a revolute joint is specified to allow rotation on the z-axes of the two Coords.

Figure 7-3

How Coord Determines Orientation When Creating Constraints



7.2 Working with Coords

This section shows how to attach, select, and modify Coords.

Creating and Attaching Coords

Coords can be attached to bodies or fixed anywhere in space.

► **To attach a Coord to a body:**



1. Click the Coord tool on the Sketch toolbar.

The mouse pointer turns into a small cross.

2. Click anywhere on the body.

The Coord attaches to the body's surface where you clicked.

► **To fix a Coord in space:**



1. Click the Coord tool on the Sketch toolbar.

The mouse pointer turns into a small cross.

2. Click anywhere on the Edit Grid.

The Coord attaches on the Edit Grid, where the mouse click is projected.

3. Follow the instructions in “Modifying Coord Properties” on page 7-5 to modify the position and orientation of the Coord.

Attaching and Detaching a Coord from a Body

You can attach a Coord to a body. Once the Coord is created, you can freely attach it to or detach it from a selected body.

► **To attach a Coord to a body:**

1. Select the Coord to be attached.

2. **Hold down the control (CTRL) key, and select the body to which the Coord will be attached.**
3. **Choose Attach Coord to Body in the Object menu.**

If the menu item is unavailable, you may have selected multiple bodies and/or Coords, or you may have not selected any Coord. Simply start over from step 1.

The Coord will be attached to the body without changing the global configuration of either. The Coord need not lie on or inside the body; any attached Coords remain fixed with respect to the body thereafter.

► **To detach a Coord from a body:**

1. **Select the Coord to be detached.**
2. **Choose Detach Coord from Body in the Object menu.**

If the menu item is unavailable, you may have selected multiple Coords or Coords that are not attached to any body. Simply start over from step 1.

The Coord is detached from the respective bodies without changing the global configuration of either. Instead, The Coord is fixed to the background space.

Modifying Coord Properties

A Coord has the following properties that you can modify:

- Appearance (name, show/hide, meter and color)
- Position and Orientation

The Properties window in Working Model 3D provides controls to all these properties.

► **To modify the appearance, position or orientation of a Coord:**

1. **Double-click the Coord, either in the document window or in the Object Manager list.**

The Properties window appears.

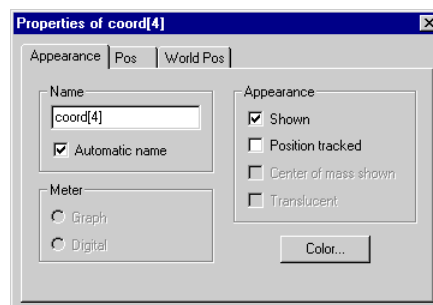
2. Click on the tab for the page containing the properties you want to modify.

Appearance

The Appearance page of the Properties window (as shown in Figure 7-4) allows you to control the name, visibility, and position tracking of a Coord. By default, the automatic name mode assigns each Coord the variable name (such as Coord[2]).

Figure 7-4

*Properties Window
(Appearance Page) for Coord*



Position and World Position

The Position page and the World Position page allow you to specify the configuration (position and orientation) of the Coord. The configuration is expressed in the frame of the attached body on the Position Page (shown in Figure 7-5) and in the World frame on the World Position Page (shown in Figure 7-6). If the Coord is not attached to any body, then both pages are identical.

If the global coordinate system is attached to a body or a Coord, then the Properties window reports the configuration in the current global coordinates and in the World coordinates. See "Attaching Global Coordinate System to a Body" on page 3-4 for more information.

Figure 7-5

*Properties Window
(Position Page) for Coord*

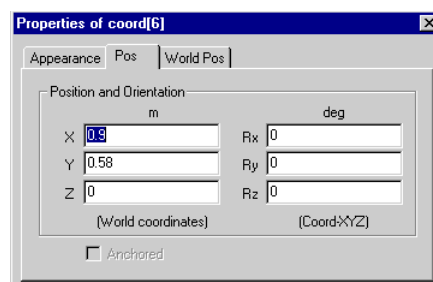
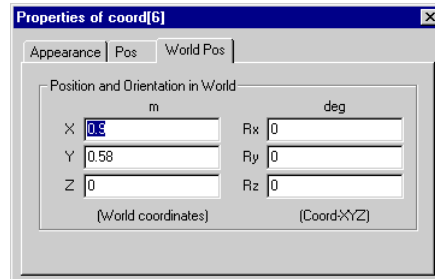


Figure 7-6
Properties Window
(World Position Page) for Coord



7.3 Creating Constraints

A constraint applies a force and/or a torque at its endpoints. Endpoints may be attached to a body or to the world. Typically, you create a constraint by specifying two Coords that will become its endpoints. Working Model 3D allows flexible options to create constraints through its *Create Constraint Window*. Please refer to “The Create Constraint Window” on page 7–13 for more information on the options available in the window.

An Example: Creating a Revolute Joint

To illustrate the concept of creating a constraint in Working Model 3D, we will join two bodies with a revolute joint.

► **To create a revolute joint between two bodies:**

1. **Create a box and a cylinder.**

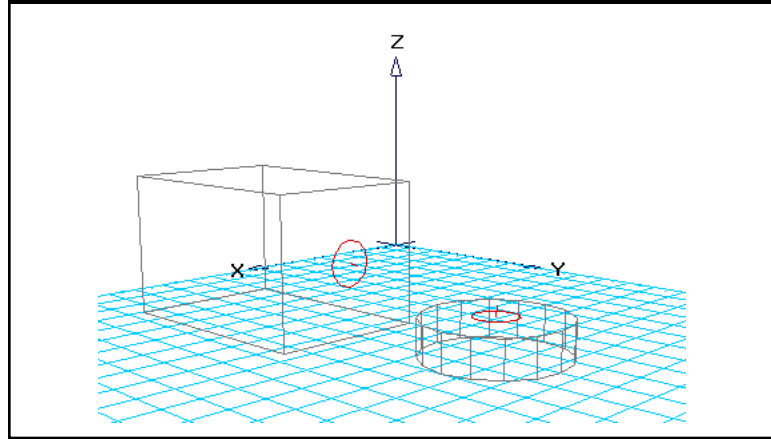
See “Creating Bodies” on page 6–5 for instructions.

2. **Attach a Coord to a face of each body.**

Figure 7-7 shows an example of two bodies with Coords attached.

Figure 7-7

A Cylinder and a Block with a Coord



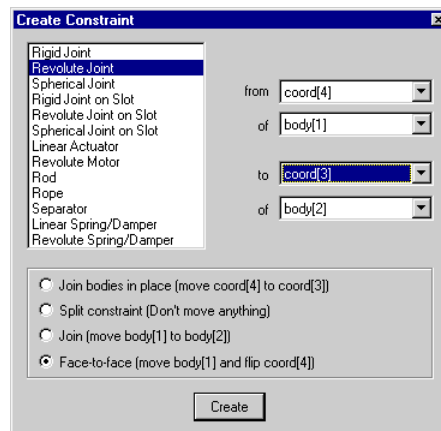
3. Select the Coord attached to the cylinder.
4. Hold down the CTRL key, and select the Coord attached to the block.
5. The order of selection is important, as you will see later.
6. Click the Create Constraint button on the toolbar.



The Create Constraint window appears (see Figure 7-8). Note that most of the parameters are already chosen to reflect the two Coords that you have just selected.

Figure 7-8

Create Constraint Window



7. Set the constraint type to Revolute Joint.

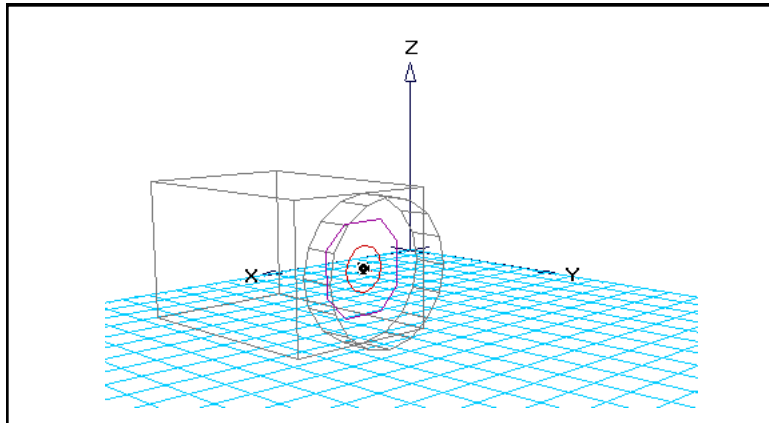
8. Choose the “Face-to-face” option.

The option specifies the behavior of the bodies when the constraint is created. Please refer to “Assembly Behavior” on page 7–15 for more information.

9. Click Create.

The cylinder is moved to the box and attached to it with a revolute joint.

Figure 7-9
Two Bodies Connected with a Revolute Joint



The example illustrated that Working Model 3D created a revolute joint under the following principles:

- (1) **The position of the two Coords are aligned**, because Working Model 3D always creates a revolute joint by matching the positions of two Coords.
- (2) **The z-axes of the two Coords are aligned**, because Working Model 3D creates a revolute joint so that the z-axes of the two Coords were aligned by default.
- (3) **The cylinder was moved to the box, and not the other way around**, because you selected the Coord on the cylinder *first* and selected the Coord on the box *second*. Note that the Create Constraint window as seen on Figure 7-8 reflects the selection order, and Working Model 3D determines the assembly behavior.

Working Model 3D creates other constraints with a variation of these principles. For example, a rigid slot joint is created so that the z-axes of the two Coords are aligned, but the positions of the Coords need not coincide, as the joint allows two Coords to translate (slide) freely. Chapter 8, “Constraints Reference”, provides the construction principles on each constraint type.

An Example: Creating a Linear Spring/Damper

Unlike a revolute joint, a linear spring/damper imposes no restrictions in degrees of freedom. A linear spring/damper applies forces at the endpoint Coords according to its response characteristics (such as the natural length, spring constant, and damper constant).

► **To create a linear spring/damper between two bodies:**

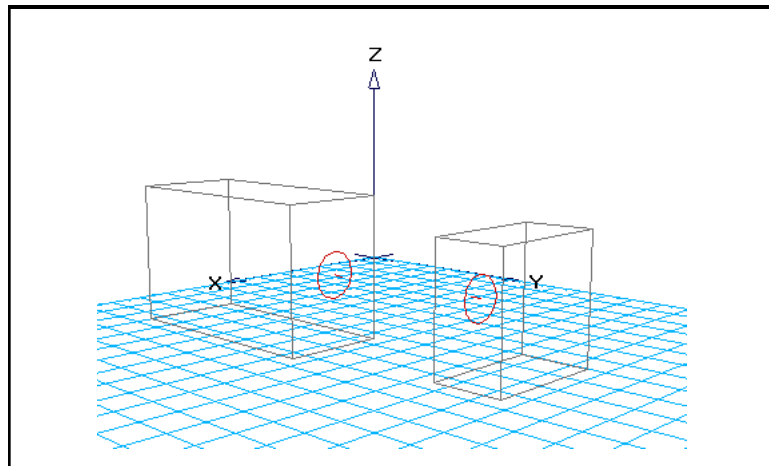
1. **Create two boxes.**

See "Creating Bodies" on page 6-5 for instructions.

2. **Attach a Coord to a face of each box.**

Figure 7-10 shows an example of two bodies with Coords attached.

Figure 7-10
Two Blocks with a Coord



3. **Select the two Coords by box-select, or select them in succession while holding down the CTRL key.**

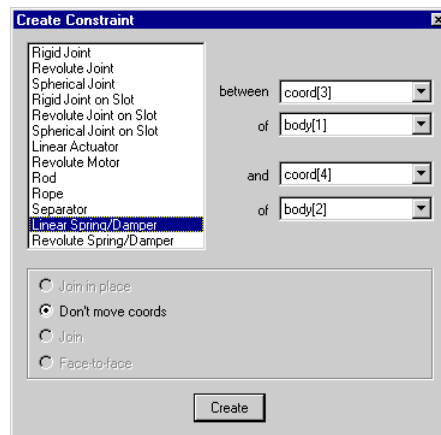
The order of selection is not important for linear spring/dampers.



4. Click the Create Constraint button on the toolbar.

The Create Constraint window appears (see Figure 7-11). Note that most of the parameters are already chosen to reflect the two Coords that you have just selected.

Figure 7-11
Create Constraint Window

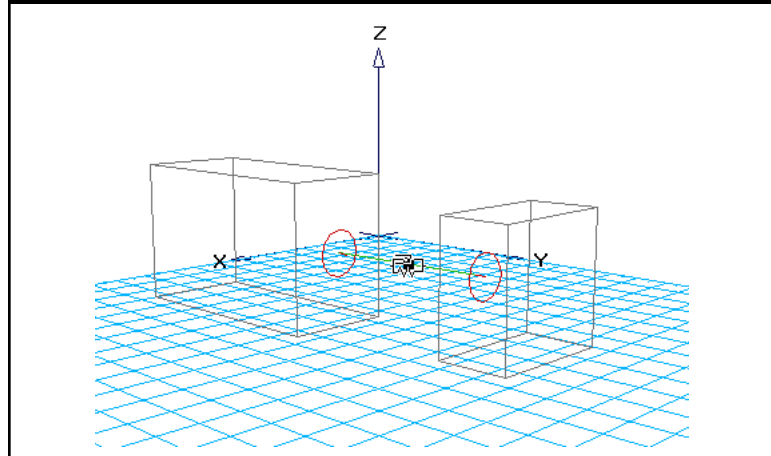


5. Set the constraint type to Linear Spring/Damper.
6. Click Create.

Working Model 3D creates a linear spring/damper between the two Coords without moving the bodies.

Figure 7-12

*Two Bodies Connected with a
Linear Spring/Damper*



The example illustrated that Working Model 3D created a linear spring/damper under the following principles:

- (1) The position and orientation of the two Coords are unaffected,** because Working Model 3D always creates a linear spring/damper between an existing pair of Coords, and sets the natural length of the spring (unloaded length) to the distance between the two Coords.
- (2) No body is moved,** because again, Working Model 3D uses the current configuration (prior to creating the constraint) as a basis to determine the natural length of the linear spring/damper.

Note that the above behaviors differ significantly from the case where you created a revolute joint (see “An Example: Creating a Revolute Joint” on page 7–7). Chapter 8, “Constraints Reference” provides the construction principles on each type of constraints.

Creating Multiple Constraints from a Pair of Coords

Working Model 3D allows you to construct multiple constraints based on a single pair of Coords. For example, you may choose to create a rigid slot joint and a linear actuator from the same pair of Coords, so that two bodies connected by the two can slide in one direction and are subject to the actuator force.

► **To create multiple constraints from a pair of Coords:**

1. **Create the first constraint from an existing pair of Coords.**

For more information, see “An Example: Creating a Linear Spring/Damper” on page 7–10.

2. **Select the same pair of Coords, and click the Create Constraint button.**

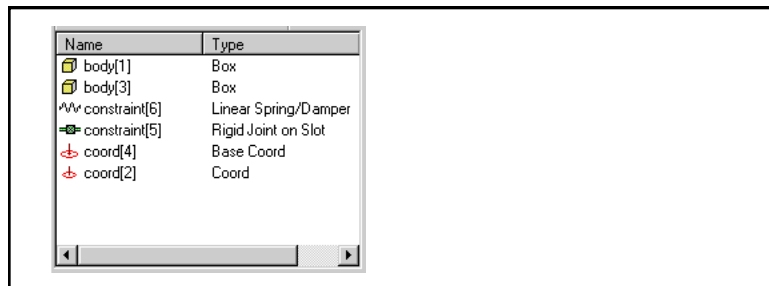
The Create Constraint dialog appears.

3. **Choose the second constraint (such as a rigid joint on slot).**

Working Model 3D creates the constraint using the same pair of Coords.

When you create multiple constraints from a pair of coords, both constraints appear in the Object Manager list (as shown in Figure 7-13). To select an individual constraint, click the corresponding icon in the list.

Figure 7-13
Multiple Constraints in the
Object Manager List

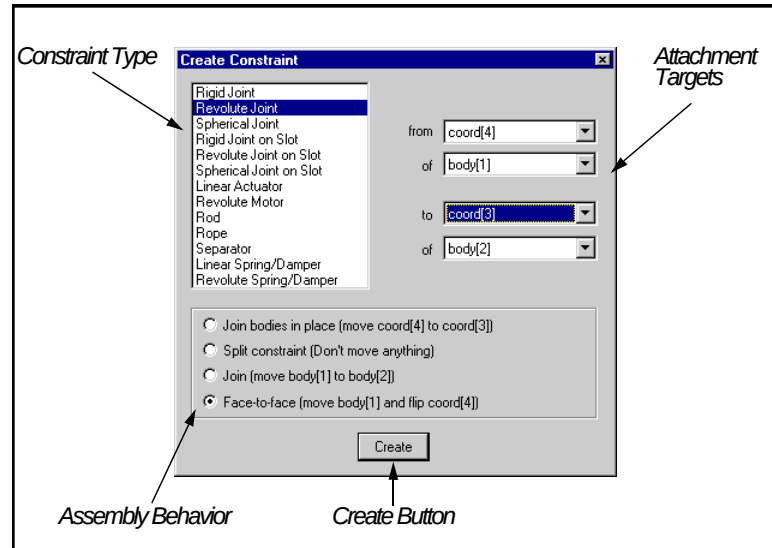


7.4 The Create Constraint Window

The Create Constraint window, as shown in Figure 7-14, is broken into four components: Constraint Type, Attachment Targets, Assembly Behavior, and the Create button. The window appears when you click the Create Constraint button on the toolbar or choose **Create Constraint** in the **Object** menu.

The Create Constraint window is a floating window. You can move it anywhere, and it remains in front; you do not need to close the window when you move on to other tasks in Working Model 3D. The window automatically closes when you click the Create button located at the bottom of the window.

Figure 7-14
Create Constraint Window



Constraint Type

From the Constraint type list box, select the constraint you would like to implement. Please refer to Chapter 8, “Constraints Reference” for more information on the constraint types.

Attachment Targets

The Attachment Target pop-up menus specify which objects will be joined together. The first pop-up menus in the two sets of boxes show the list of Coords, while the second of the two pop-up menus show corresponding bodies (or the background space) to which the Coords are attached. You can always use the mouse to select objects to be joined, and the Create Constraint window will fill these fields automatically.

Depending on the type of constraint, the Attachment Target changes its wording slightly.

- For Coord-on-Coord constraints (such as any joint or a motor), the window lists the Coords and bodies with prepositions *from...to...* to indicate that the first body may be moved to the second upon the construction of the constraint.
- For point-to-point constraints (such as a spring/damper, a rope, or a linear actuator), the window lists the Coords and bodies with prepositions *between...and...* to indicate that no motion will occur upon the construction of the constraint.

Figure 7-15
Changing Wording in the Window

For constraints that may require bodies to move (joints, motors)

from	coord[4]
of	body[3]
to	coord[2]
of	body[1]

For constraints that do not require bodies to move (e.g., linear spring damper, linear actuator)

between	coord[4]
of	body[3]
and	coord[2]
of	body[1]

Assembly Behavior

The Assembly Behavior option allows you to specify how the bodies will be moved when you create the constraint between them. Depending on the objects you select and the constraint type, Working Model 3D presents options as radio buttons in the window.

At most, four options are available:

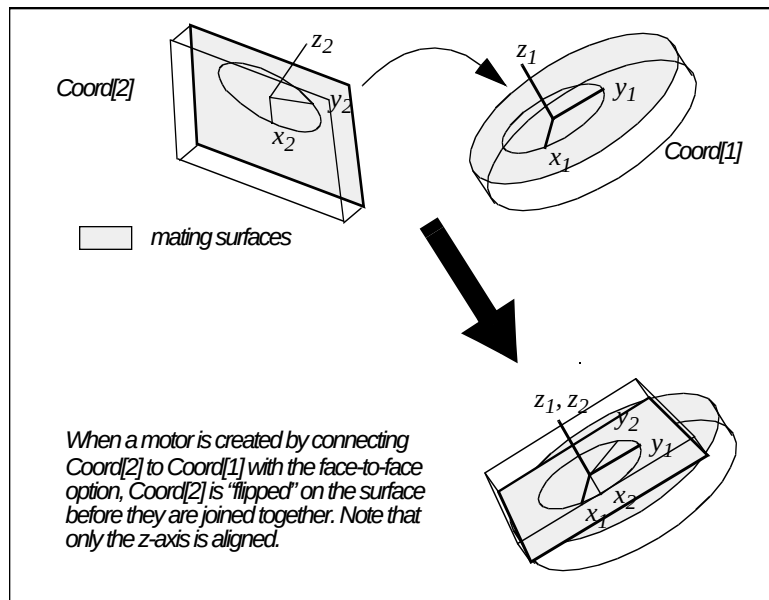
- **Join bodies in place** moves one of the Coords to match the other. If only one Coord is selected, another Coord is created at the same position. The constraint condition is satisfied immediately.
- **Split constraint** moves no Coord or body. Working Model 3D creates the constraint but keeps it split; in other words, the constraint is specified, but the model is not yet assembled to satisfy the constraint condition. See "Splitting and Joining Constraints" on page 7–30 for more information.

- **Join** moves the body so that the configurations of two Coords satisfy the constraint condition.
- **Join Face-to-face** flips one of the Coords, and moves the body so that the configurations of two Coords satisfy the constraint condition. This option is useful when two Coords are attached to a body's surface.

As shown in Figure 7-16, the Face-to-face option provides a convenient feature when you wish to put together two bodies when Coords already exist as the attachment points on the mating surfaces. If you had chosen the “join” option, the box (on which Coord[2] is located) in Figure 7-16 would have blended with the cylinder (on which Coord[1] is located), because Working Model 3D would have attempted to align the z-axes and their directions.

Figure 7-16

Aligning Coords for Motors and Revolute Joints



The following table illustrates which Assembly Behavior options are available, depending on the objects you select.

Assembly Behaviors for Various Constraints

You select	for Coord-on-Coord constraints (e.g., joints)	for point-to-point constraints (e.g., linear spring/damper, rod, rope)
2 Coords attached to distinct bodies	Join bodies in place, Split constraint (join) or face-to-face	Join (Don't move Coords).
a Coord attached to a body	Join in place & creates another Coord at the same global position as the first Coord and attaches it to the background)	Join in place & creates another Coord at the same global position as the first Coord and attaches it to the background)
a body and a Coord attached to a body	Join in place & creates another Coord at the same global position as the first Coord and attaches it to the body)	Join in place & creates another Coord at the same global position as the first Coord and attaches it to the body)

7.5 Applying External Forces or Torques

You can apply external forces or torques to your model in four ways (or combinations thereof) as described below:

- The **Force** constraint applies a force (point load) to a body.
- The **Torque** constraint applies a torque to a body.
- The **Motor** constraint applies a controlled torque between two bodies, or between the background and a body. You can specify the motor to (1) apply a specified torque, (2) maintain a specified angular acceleration, (3) maintain a specified angular velocity, or (4) maintain a specified orientation.
- The **Actuator** constraint applies a controlled force between two bodies, or between the background and a body. You can specify the actuator to (1) apply a specified force, (2) maintain a specified acceleration, (3) maintain a specified velocity, or (4) maintain a specified length.

- The **Linear Spring/Damper** constraint apply external forces or torques when preloaded.

For more information, please refer to the appropriate constraint sections in Chapter 8, “Constraints Reference”.

7.6 Modifying Constraints

Working Model 3D offers a variety of options to modify and customize constraints once you construct them. For example, you can change the constrained axis of a revolute joint, or even change one type of constraint to another.

Understanding Common Constraint Properties

Constraint Classes

The constraints available in Working Model 3D can be classified into three types as follows:

- Coord-on-Coord constraints
- point-to-point constraints
- on-Coord constraints

Coord-on-Coord constraints consist of two Coords and always limit at least one degree of freedom. Coord-on-Coord constraints include all joints (such as revolute joints, spherical joints, slot joints) as well as motors, because a motor is considered as a revolute joint that applies a controlled torque. The term *Coord-on-Coord* is derived from the fact that all such constraints require that positions and/or axes of two Coords coincide.

Point-to-point constraints also consist of two Coords but do not limit any degree of freedom. Point-to-point constraints include linear spring/dampers, ropes, separators, and actuators. The term *point-to-point* is derived from the fact that all such constraints are created from one point to another, without consideration to orientations.

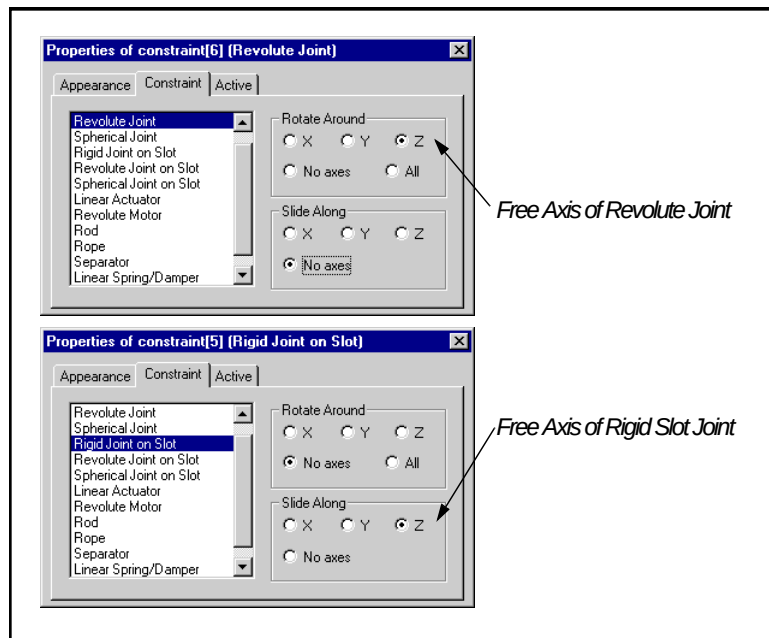
On-Coord constraints consist of a single Coord and do not limit any degree of freedom. The only on-Coord constraints supported by Working Model 3D are the Force and Torque constraints. The term *on-Coord* is derived from the fact that the constraint is attached on a Coord and includes orientation as one of the properties.

Properties on Coord-on-Coord Constraints

A Coord-on-Coord constraint limits at least one degree of freedom, and you can observe the limitation through the Properties window. As shown in Figure 7-17, a revolute joint allows rotation with one axis fixed, whereas a rigid slot joint allows translation along a single axis.

For more information, see Chapter 8, “Constraints Reference”.

Figure 7-17
Properties Window Samples for
Coord-on-Coord Constraints



Selecting Constraints

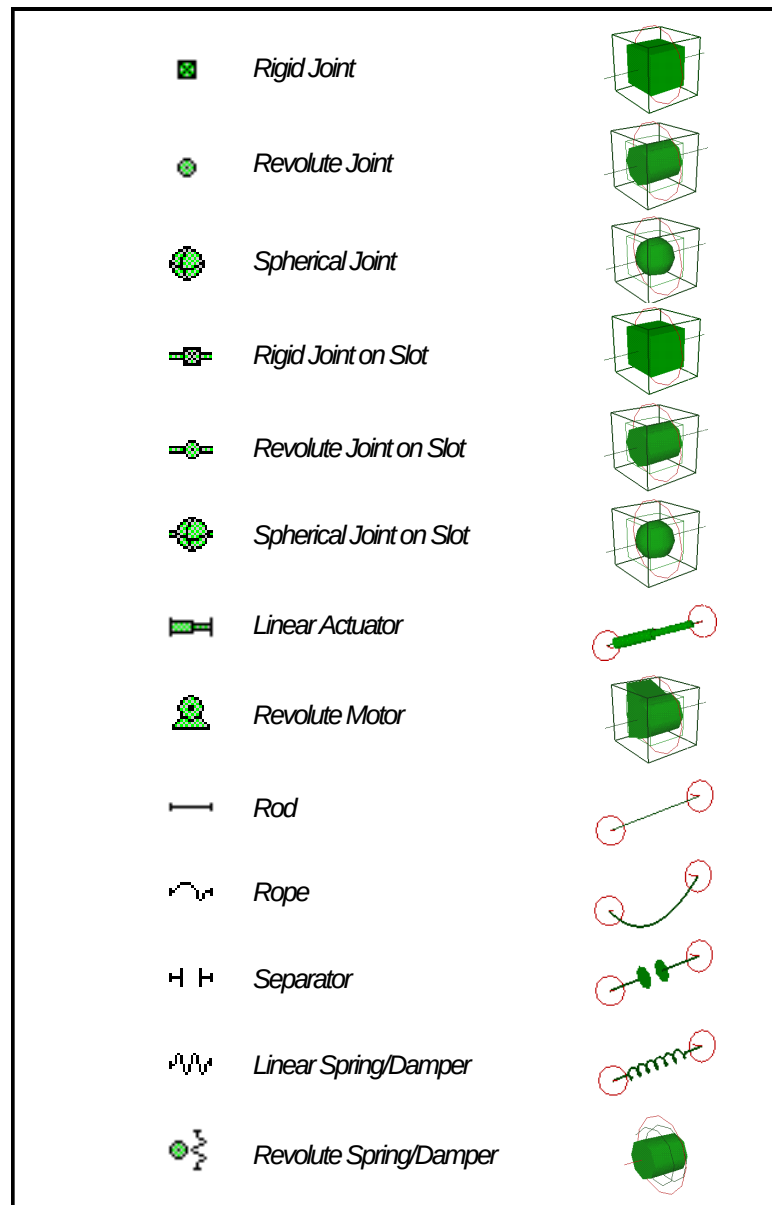
Once a constraint is created, you can select the constraint by simply clicking the constraint icon. You can select a constraint even if it is “hidden” under or located inside bodies. You can also select a constraint from the Object Manager list on the left side of the document window.

They are shown with solid 3D symbols that indicate the degrees of freedom. Point-to-point constraints have visual clues to show displacement (such as slack ropes, corkscrew springs, or extending actuators).

When selected, constraints are shown in thicker outline or with bounding boxes.

In addition, the nature of the constraint is shown by a representative icon both in the Object Manager list (shown on the left side of Figure 7-18) and on the drawing (shown on the right side of Figure 7-18).

Figure 7-18
Constraint Types



Modifying Attachment Positions

You can modify the attachment positions of the constraint by re-positioning the endpoint Coords.

► **To modify the attachment positions:**

1. Select the constraint in the Object Manager list.

The attachments appear in the Object Manager Connection List (shown in Figure 7-19).

2. Double-click the Coord whose position you want to modify in the Object Manager Connection list.

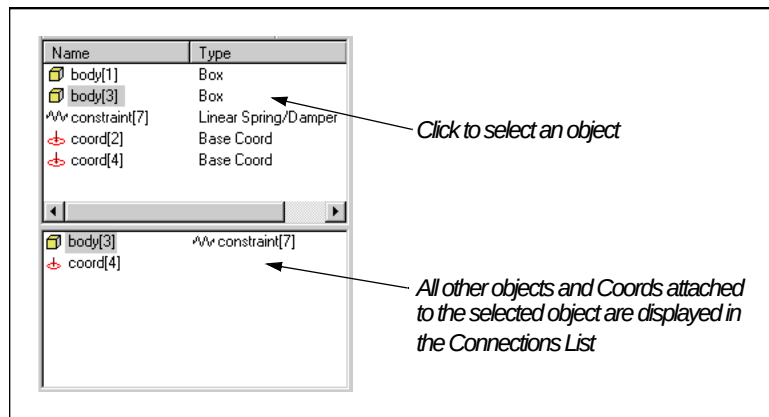
The Properties window appears.

3. Click the Position or World Position tab, then modify the position and orientation of the Coord.

The position and/or orientation of the Coord changes accordingly, while the other Coords remains intact.

4. If you wish to modify the position and orientation of the mating Coord, select it and follow the steps above.

Figure 7-19
Finding the Connections



NOTE: For Coord-on-Coord constraints, once the attachment position is changed, the constraint is in the split state. For point-to-point constraints, such as linear spring-dampers, changing the attachment position does not split the constraint.

Before you run the simulation, be sure to join the constraint. Please see "Splitting and Joining Constraints" on page 7–30 for more information.

Alternatively, you can split the constraint, separate the bodies, and edit the attachment position of individual Coords. Please see "Splitting and Joining Constraints" on page 7–30 for more information.

Modifying Constraint Properties

Constraint properties vary depending on the constraint classes.

Joints and Motors

For joints and motors, you can change the constraint axes. For example, a revolute joint allows z-axis rotation by default, and you can modify the joint so that it allows rotation about the x-axis instead.

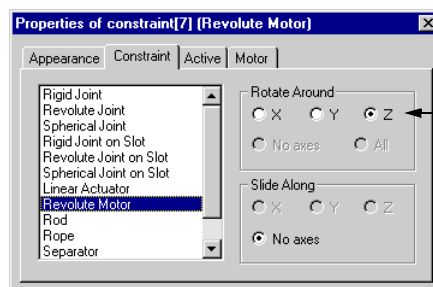
► To change the free axis (axes):

1. Double-click the constraint icon in the document or in the Object Manager list that runs along the left side of the document window.

The Properties window appears with the Constraint page in front (Figure 7-20).

Figure 7-20

Example: Constraint Properties Windows for a Motor



Free Axes Radio Buttons
(Note that motors do not allow any translation or sliding)

2. Choose the axis to rotate around.

The axis designated refers to the coordinate axis of the Coord that appeared on top in the Create Constraint window when the constraint was created. See "The Create Constraint Window" on page 7-13 for more information.

NOTE: Once the free axis is changed, the constraint may results in the split state because the free axes of the Coords may not always have been aligned. Before you run the simulation, make sure to join the constraint. Please see "Splitting and Joining Constraints" on page 7-30 for more information.

3. Choose the axis to slide along, if the constraint has translation and degree of freedom.

Each type of constraint allows an appropriate number of degrees of freedom. If you would like to change the number of degrees of freedom, you need to change the constraint type. See "Changing Constraint Type" on page 7-24 for more information.

Point-to-Point Constraints

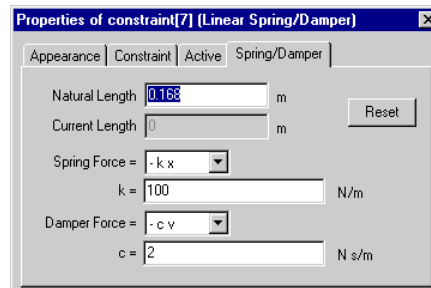
For point-to-point constraints, you have options to change properties that are unique to the constraint. For example, a linear spring/damper has natural length, current length, spring force, and damper force as its properties.

► To change the constraint properties:

- 1. Double-click the constraint.**
- 2. Select the Spring/Damper page of the Properties window.**

The Spring/Damper Properties page is shown here (Figure 7-21).

Figure 7-21
Spring/Damper Properties Page



3. Modify the Spring/Damper properties as desired.

For more information on individual constraint properties, please see Chapter 8, “Constraints Reference”.

Changing Constraint Type

Working Model 3D allows you to change the constraint type even after the constraint is created. You do not need to reconstruct the model from scratch when the constraint type is changed.

The feature may prove useful when you change a component design. For example, you can change a revolute joint to a spherical joint to allow more degrees of freedom, or you can change a revolute joint to a motor to drive the mechanism.

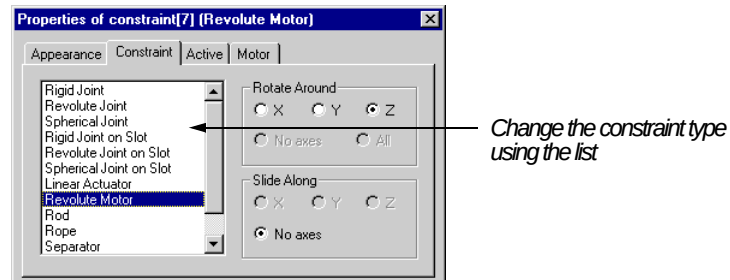
► To change the constraint type:

1. Double-click the constraint icon either in the document or in the Object Manager list that runs along the left side of the window.

The window appears as shown in Figure 7-22.

2. Change the constraint type by selecting another type of constraint from the list.

Figure 7-22
Constraint Properties Page



Split Constraints Caused by Type Change

When the constraint type is changed, the constraint may be in the split state. The following rules dictate this behavior:

- If you change the type of any constraint to a point-to-point constraint (e.g., spring-damper, rod, rope, actuator), the constraint is joined.
- If you change the type of any constraint to a Coord-on-Coord constraint (e.g., revolute joint, slot joint, motor), the constraint is split. However, if the 2 coords are exactly aligned, the constraint will remain joined.

Before running the simulation, be sure to join the split constraints for them to have any effect. Please see "Splitting and Joining Constraints" on page 7–30 for more information.

Modifying Constraint Appearance and Name

You can change the color of the constraint and give it a customized name.

► To change the appearance:

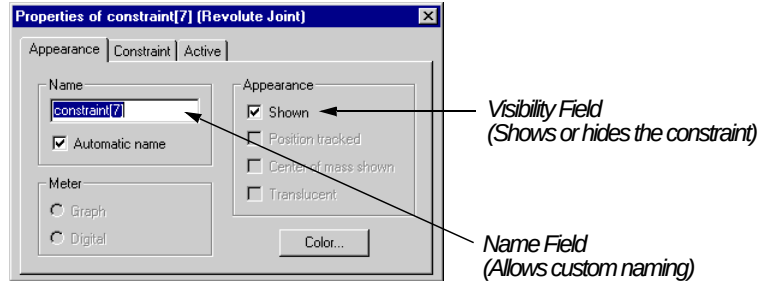
1. **Double-click the constraint icon, either in the document or in the Object Manager list.**
2. **Open the Appearance page of the Properties window.**

The window appears as shown in Figure 7-23.

3. **Enter the desired values for the appropriate control.**

See below for the descriptions on each control.

Figure 7-23
 Constraint Properties Window
 (Appearance Page)



The Appearance page of the Properties window controls the following:

Name

By default, Working Model 3D provides the name of the constraint as `constraint[n]`, where *n* is a unique integer. (This default name is actually used as a variable in Working Model 3D's formula language. For details on the use of formula language, see Chapter 12, "Using Formulas".)

Since the customized constraint name appears in the status bar (see "Status Bar" on page 2–16) as well as the selection list of the Properties window and the Create Constraint window, assigning a custom name helps identification when your model involves a lot of constraints of the same type.

► **To change the constraint name:**

1. Double-click the constraint icon.
2. Select the Appearance page in the Properties window.
3. Highlight the object name field in the window, and type the custom name.
4. Press Enter.

From then on, the custom name appears in the status bar and selection lists.

► **To switch back to the default name:**

1. Follow steps 1 and 2 above.
2. In the Name section of the window, click the box labeled Automatic Name.

From then on, the default name (constraint[n]) appears in the status bar and selection lists.

Show

These options provide controls to show or hide the constraint.

► **To show or hide the constraint:**

1. **Double-click the constraint icon.**
2. **Select the Appearance page in the Properties window.**
3. **Click the Shown checkbox to show or hide the constraint.**

The constraint icon appears on the screen when the checkbox is checked and hides from the screen when the checkbox is blank.

NOTE: The Shown checkbox only controls the visual presence of the constraint. Whether the constraint is visible or not, Working Model 3D takes the constraint into account while computing the simulation.

Track Position

You can also track the position of the constraint as the model is run.

► **To track the history:**

1. **Double-click the constraint icon.**
2. **Select the Appearance page in the Properties window.**
3. **Click the Track Position checkbox to show a trail of the position of the constraint during the last 150 frames of the simulation.**

Deleting Constraints

When you delete a constraint, the Coords to which the constraint was attached remain intact. Therefore, you can create another constraint using the same set of Coords.

► **To delete a constraint:**

1. **Select the constraint by clicking on the constraint icon either in the document window or in the Object Manager list.**

The constraint is outlined in a darker color or a bounding box appears around the constraint (see “Selecting Constraints” on page 7–19 for more information).

2. Press the Delete key.

The constraint disappears, while the Coords to which the constraint was attached remain.

7.7 Editing Constrained Bodies

Once you connect bodies with constraints, you can change the body configurations without breaking the constraints. The behaviors differ depending on whether bodies are connected with Coord-on-Coord constraints or point-to-point constraints.

Modifying Configuration of a Constrained Body

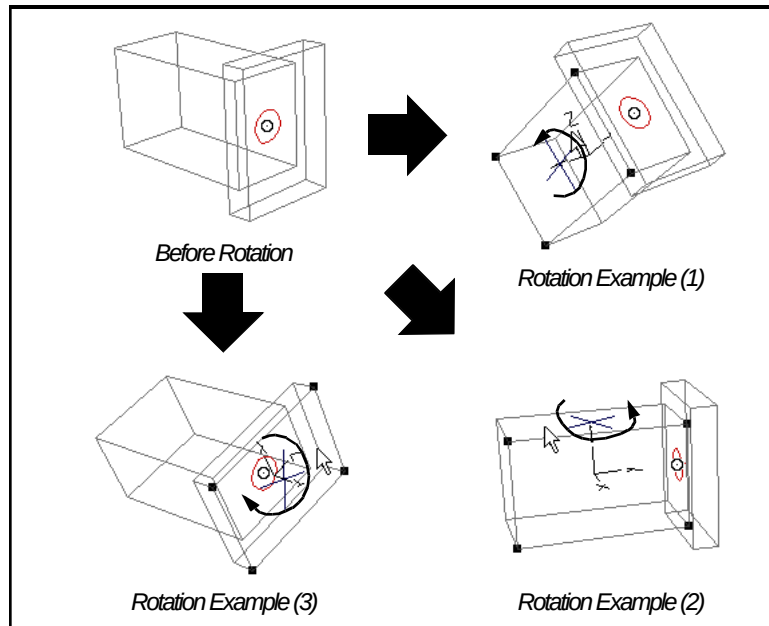
After you assemble a model, you may wish to modify a configuration of a body without breaking the constraint in order to create an initial condition for a simulation. See section “Splitting and Joining Constraints” on page 7–30 for instructions.

Dragging and Rotating Bodies

Connected with Coord-on-Coord Constraints

If you have bodies connected with Coord-on-Coord constraints, you can move one of the bodies, and the others will follow without breaking the constraints. For example, consider two bodies connected with a revolute joint. When you use the Rotate tool to rotate one body, the other body follows (see Figure 7-24).

Figure 7-24
 Rotating Bodies Connected with a Revolute Joint

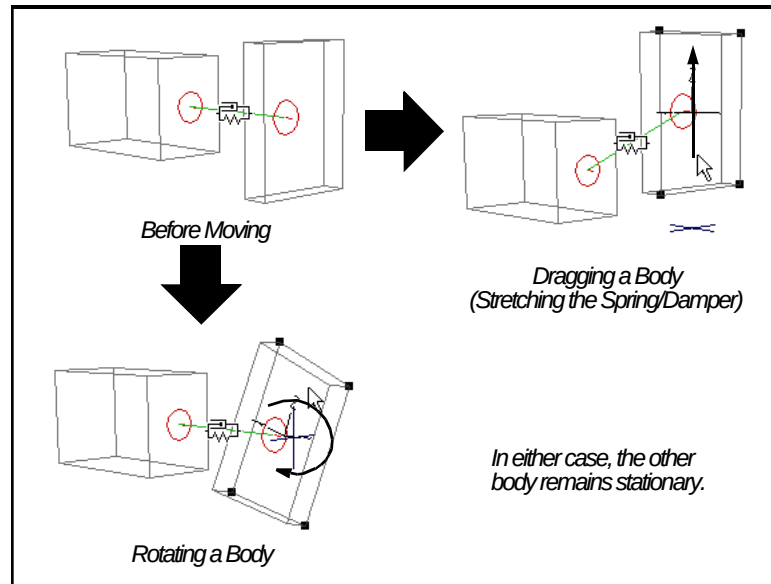


NOTE: In *Rotation Example (3)* above, the box rotates with the thin slab despite the revolute joint connecting the two. Although a revolute joint allows rotation during the simulation, Working Model 3D does not allow you to “drag” bodies independently if they are bound by a joint. You can temporarily disable this synchronized behavior by splitting—temporarily disabling—the desired constraint(s). Please see “Splitting and Joining Constraints” on page 7–30 for more information.

Connected with Point-to-Point Constraints

If you have bodies connected with point-to-point constraints, Working Model 3D does *not* synchronize the motion of the bodies while you move them. For example, consider two bodies connected with a linear spring/damper. When you use the drag tool to move one body, the other body does *not* follow. Instead, the spring/damper stretches, causing the spring to be in tension or compression.

Figure 7-25
Moving Bodies Connected with a Spring/Damper



7.8 Splitting and Joining Constraints

When a Coord-on-Coord constraint (such as joints and motors) connects two bodies together, you can drag one body and the other will follow. However, to edit part of a model, you may wish to disable this binding temporarily.

Splitting Joints

When you “split” a joint, you can modify the configuration of the bound bodies independently. After modifying the bodies, you can “join” the joint to re-assemble your model.

► ***To split a constraint:***

1. **Select the constraint icon, either in the document window or in the Object Manager list.**

The Split button on the toolbar becomes available to indicate that the button is active.

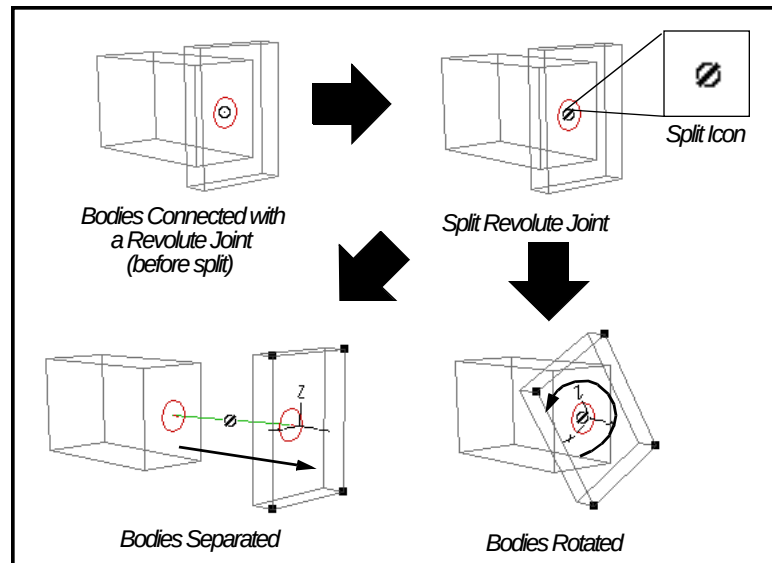
If the Split button is not active, you may have chosen a constraint that is already split. Split constraints bear a special icon as shown in Figure 7-26. Otherwise, you may have not selected the constraint properly. See “Selecting Constraints” on page 7-19 for more information.



2. Click the Split button on the Edit toolbar.

In the Object Manager, the constraint icon shows the crossed-out line to indicate that the constraint is split. In the document window, the constraint is wire-frame instead of solid.

Figure 7-26
Split Constraint



Now that the constraint still exists but is in the split state, you can drag one body and leave the other intact. You can later select the split constraint and reassemble it with the Join command (see below).

NOTE: Split constraints do not have any effect (do not apply any force or torque to any body) during the simulation run. You need to join them before you run the simulation.

Joining Constraints

As shown in Figure 7-26, the split constraint retains connection to the original Coords. Working Model 3D allows you to put it back together very easily.

Please note that Working Model 3D automatically splits constraints under the following circumstances:

- If you change the type of any constraint to a point-to-point constraint (e.g., spring-damper, rod, rope, actuator), the constraint is joined.
- If you change the type of any constraint to a Coord-on-Coord constraint (e.g., revolute joint, slot joint, motor), the constraint is split. However, if the 2 coords are exactly aligned, the constraint will remain joined. See “Changing Constraint Type” on page 7–24 for more information.
- When you modify the constraint condition, the constraint is split in certain cases. See “Modifying Constraint Properties” on page 7–22 for more information.

► **To join a split constraint:**

1. Select the split constraint icon.

The Join button becomes available to indicate that the button is active.

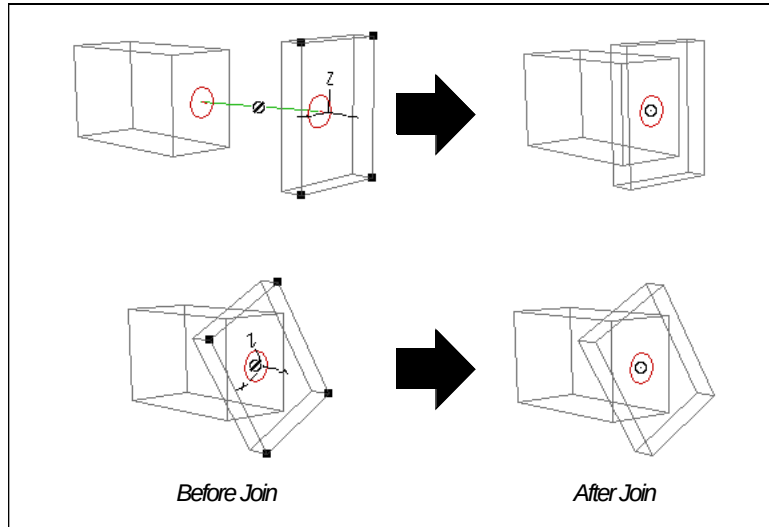
If the Join button is unavailable, you may have chosen a constraint that is yet to be split. In the Object Manager, the constraint icon shows the crossed-out line to indicate that the constraint is split. In the document window, the constraint is wire-frame instead of solid. If you have trouble selecting a constraint with your mouse, see “Selecting Constraints” on page 7–19 for more information.



2. Click the Join button on the Edit toolbar.

The bodies move together to satisfy the constraint condition. Depending on the configuration, Working Model 3D may show a progress dialog to show the iterative process of finding a converging solution. See “Constructing a Closed-Loop Assembly” on page 7–33 for more information.

Figure 7-27
Before and After Join



Constructing a Closed-Loop Assembly

The examples shown in Figure 7-27 are classified as *open-loop assemblies*. When you create or join constraints in an open-loop assembly, bodies or Coords move immediately to satisfy the constraints.

Figure 7-28
A Closed-Loop Assembly

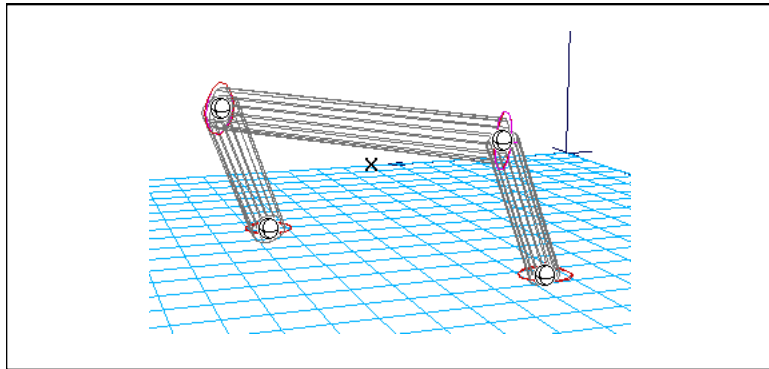
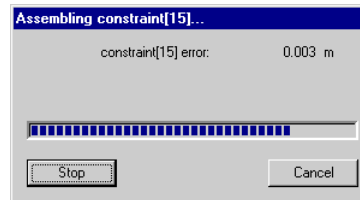


Figure 7-28, on the other hand, shows an example of a closed-loop assembly. Observe that a series of bodies are linked in a closed loop (the background is considered as a single link).

When you click the Join button to put together a model to satisfy all the constraints, Working Model 3D starts an iterative process to find the converging solution to put together the model, while showing the progress in the Assembly Progress dialog (Figure 7-29). At the same time, the bodies move to an assembled state.

Figure 7-29
Assembly Progress Dialog



You can either:

- click the Stop button, to leave the bodies halfway through the assembly process without being completely assembled, or
- click the Cancel button, to cancel the assembly command completely and leave the bodies intact.

The assembly dialog shows the indicators in the following table. During a normal assembly process, the magnitude of these numbers decreases steadily.

Assembly Dialog Indicators and Descriptions

Indicator	Description
Total Error	The sum of all positional and orientational errors.
Maximum Positional Error	Reports the constraint ID where the maximum positional error is observed.
Maximum Orientational Error	Reports the constraint ID where the maximum orientational error is observed.

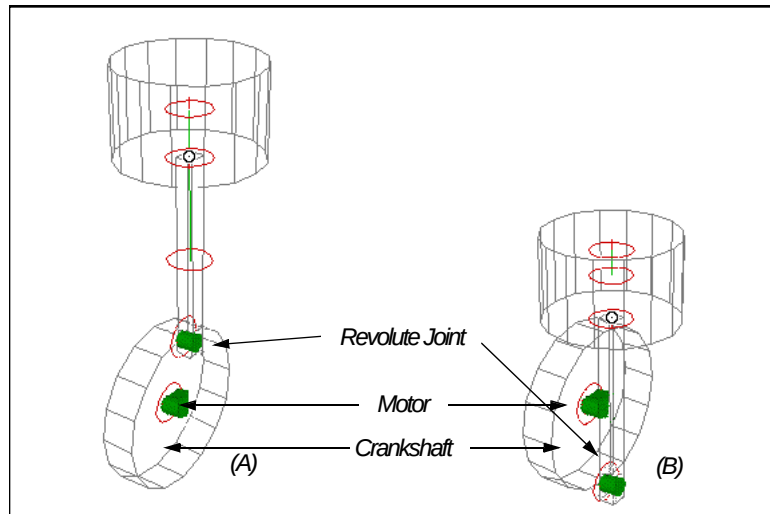
When you attempt to assemble an impossible configuration, the progress may halt. See “Impossible Configurations” on page 7–36 for more information.

Changing Configuration of a Closed-Loop Assembly

When you click the Join button, Working Model 3D finds any solution that satisfies the given constraint conditions. In some cases, however, the assembled configuration may be different from the desired initial condition.

For example, Figure 7-30 below shows a piston engine mechanism with two possible configurations. After specifying all the constraints, Working Model 3D may assemble your mechanism to the state shown as (A), whereas you want the initial condition to look like (B). Note that when the simulation starts, the phase of the combustion cycle would be 180° off between the two.

Figure 7-30
Multiple Configurations for a
Closed-Loop Assembly



Follow the simple steps shown below to change the configuration.

► **To specify the initial configuration for a closed-loop assembly:**

1. Identify the “reference body”, or the body which you wish to use as a reference to specify the initial condition.

For example, in the example shown in Figure 7-30 above, suppose that you wish to change the crankshaft angle. In this case, the reference body is the crankshaft.

2. Split all constraints that are attached to the reference body.

In the example above, the motor and a revolute joint shown in Figure 7-30 are the two constraints.

- 3. Specify the configuration of the reference body to match the initial condition.**

If the reference body is to be attached to the background with a constraint, the body must satisfy the constraint condition. In this example, specify the configuration of the crankshaft without breaking the motor constraint.

- 4. In the Properties window's Position page of the reference body, check the Anchored box to fix the body in place.**

See “Fixing a Body in the World” on page 6–24 for more information on anchoring.

- 5. Join the constraints connecting the reference body and other bodies.**

Working Model 3D will reassemble the closed-loop assembly, while leaving the reference body intact.

- 6. Release the anchor on the reference body.**

You must anchor the body before joining the constraint that connects the body to the background.

- 7. If the reference body is attached to the background with a constraint, join the constraint.**

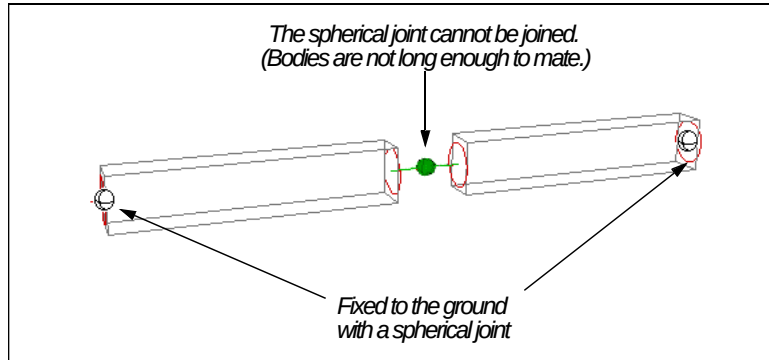
In the example, join the motor constraint.

Impossible Configurations

When you click the Join button, Working Model 3D attempts to find the convergent solution to satisfy all constraints, including the one being “joined”. If the solution is not possible, the assembly progress dialog halts, and the magnitudes of the error parameters cease to decrease.

For example, Figure 7-31 shows a configuration for which no solution exists to join the spherical joint positioned between the two bodies.

Figure 7-31
Impossible Configuration for Join



When you attempt to join this type of assembly, the bodies may appear to start breaking all constraints, and the assembly progress halts. Click the Cancel button in the Assembly Progress dialog to abort the assembly, and modify the model to avoid impossible configurations.

C H A P T E R 8

Constraints Reference

This chapter serves as a reference for the full description of constraint characteristics. The chapter is organized by the type of the constraint, rather than tasks involved in creating them.

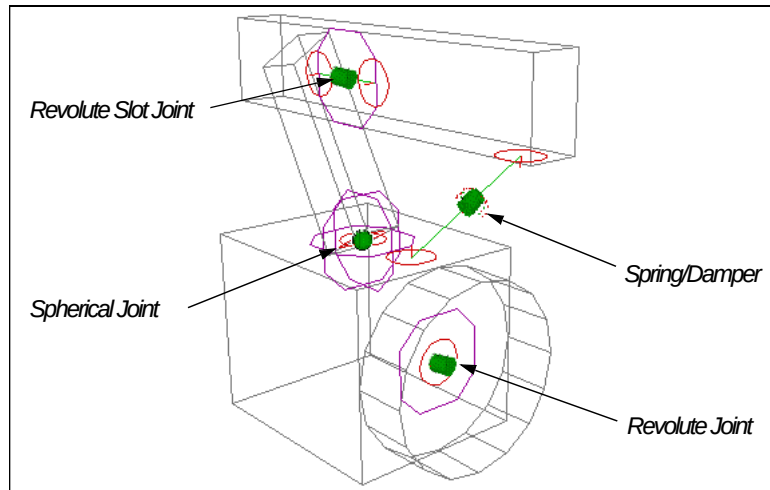
If you are interested in learning how to create and modify constraints, please see Chapter 7, “Assembling Models”.

8.1 Introduction

What is a constraint?

In Working Model 3D, a constraint is a representation of certain rules that govern the behavior of bodies among themselves. For example, a *revolute joint* (like a pin or a hinge) is used to limit the motion of two bodies, so that one body can only rotate about a certain axis with respect to the other body (see Figure 8-1). The bodies connected by a revolute joint cannot fly apart because the constraint prohibits such a motion. Other examples of constraints are *joints*, *springs*, and *forces*.

Figure 8-1
Constraints in Working Model 3D



A constraint applies force at the respective Coords that are attached to the body or to the background. If the constraint applies torque, the torque is applied to the body to which the Coord is attached.

Working Model 3D considers constraints to be massless, volumeless objects in simulations. For example, springs would not collide with another spring or a body. A damper does not have any mass. Yet a constraint applies force and/or torque to bodies at the Coords where it is attached.

Creating Constraints

Typically, you will be creating constraints by specifying two points that will become its endpoints. Working Model 3D allows many more flexible options to create constraints. For example, you can also select a single Coord attached to a body, and Working Model 3D will automatically generate a matching Coord necessary to construct the constraint.

This chapter is dedicated toward constraint properties, and the section "Creating Constraints" on page 7-7 will guide you through the process of creating constraints.

8.2 List of Constraints

Working Model 3D offers the following constraint types:

- Rigid Joint
- Revolute Joint
- Spherical Joint
- Rigid Joint on Slot
- Revolute Joint on Slot
- Spherical Joint on Slot
- Linear Actuator
- Revolute Motor
- Rod
- Rope
- Separator
- Linear Spring/Damper
- Revolute Spring/Damper

The remainder of the chapter describes each constraint in greater detail, as well as how to apply force and torque.

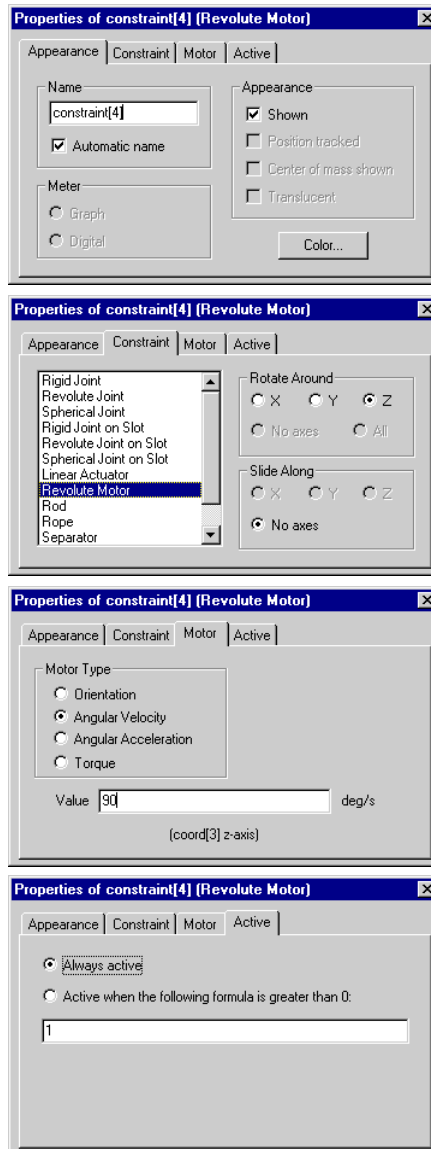
8.3 The Constraint Properties Window

The floating Properties window (Figure 8-2) allows easy access to constraint properties. The window provides access to four pages as follows: The pages are available to all constraint types.

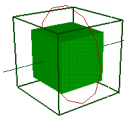
- The **Appearance** page shows the visual characteristics of the constraint (name, visibility, and color) used in rendering.
- The **Constraint** page shows the type of constraint and axis settings.
- The **Active** page allows you to set when the constraint will be active.
- The (Constraint Type) page shows the type and value of the constraint.

Figure 8-2 shows the Properties window (with the four pages) for a motor as an example.

Figure 8-2
Properties Window Pages
(for Motor)



To change from one page to another, simply click the desired tab (see Figure 8-2). For detailed discussions on modifying constraint properties, please refer to “Modifying Constraints” on page 7–18. Subsequent sections of this chapter serves as a reference for individual constraint types.



8.4 Rigid Joint

A rigid joint locks together two bodies completely. Working Model 3D considers two bodies locked together by a rigid joint as a single body, thereby reducing computing effort. A single rigid joint is sufficient to lock two bodies together, even when the connected bodies have multiple contact points.

NOTE: A single rigid joint is the most efficient and the only proper way to lock two bodies together in Working Model 3D. Do *not* use multiple joints (such as two revolute joints or two rigid joints), or your simulation will suffer from instability, inaccuracy, or poor performance.

Assembly Behavior of Rigid Joint

When the constraint is created, Working Model 3D arranges bodies and/or Coords so that the two Coords are completely aligned (i.e., their origins coincide, and all axes are aligned).

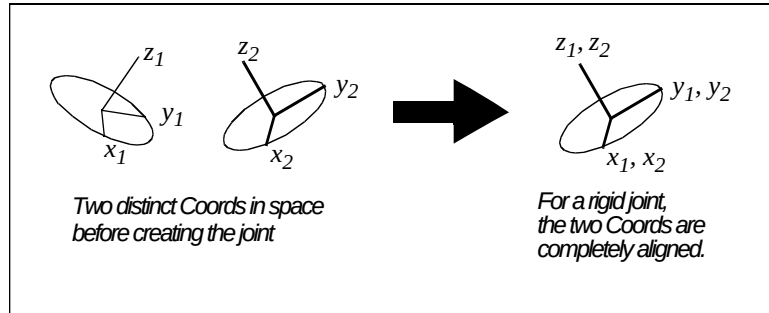
The Create Constraint window presents the following options when you attempt to create a rigid joint.

Assembly Behavior of a Rigid Joint

Selected Objects	Join Option	Behavior
Two Coords attached to bodies	Join body in place	Moves the first Coord so that it is aligned with the second. Creates a rigid joint connecting the two bodies without moving them.
	Don't move anything	Creates a rigid joint but does not move a Coord or body. You need to join the constraint later for it to be active.
	Join	Moves the first body so that the axes and origins of the two Coords are aligned.
	Face-to-face	Rotates the first Coord 180° with its x-axis fixed, and moves the first body so that the axes and origins of the two Coords are aligned.
One Coord attached to a body	Join in place	Creates another Coord attached to the background, with the same global configuration as the selected Coord. Creates a rigid joint to lock the body in place.
One body	Join in place	Creates two Coords at the coordinate origin of the body with the same configuration. Creates a rigid joint to lock the body in place.
One Coord attached to a body and one lone body	Join in place	Creates another Coord attached to the lone body, with the same global configuration as the selected Coord. Creates a rigid joint to lock the bodies in place.

When a rigid joint is created from two existing Coords, Working Model 3D aligns the origin and the coordinate axes as shown in Figure 8-3. The attached bodies have no degree of freedom relative to each other.

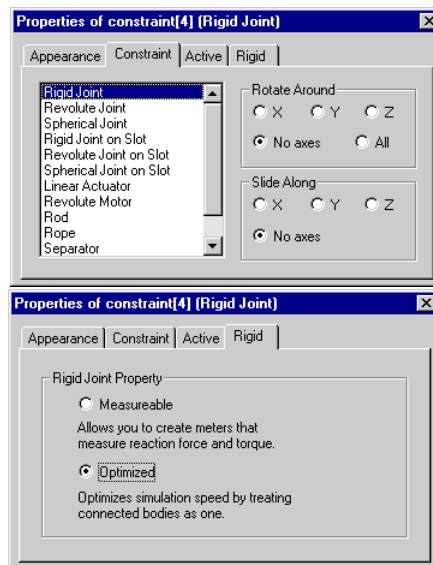
Figure 8-3
Aligning Coords for Rigid Joint



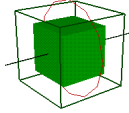
Rigid Joint Properties

As shown in the Constraint Properties page (Figure 8-4), a rigid joint has no degrees of freedom. You can set whether the force and torque on the rigid joint will be measurable with meters, or if the connected bodies should be treated as one to speed simulation.

Figure 8-4
Properties Window Pages for a Rigid Joint



See "The Constraint Properties Window" on page 8-3 for additional properties that are common to all constraints.



8.5 Rigid Joint on Slot

A slot with rigid joint allows a single degree of freedom in translation. Bodies connected with a slot with rigid joint can “slide” with respect to each other, but cannot rotate at all. A slot with rigid joint is also known as a *sleeve joint* in the mechanical design.

Assembly Behavior of Rigid Joint on Slot

The Create Constraint window presents the following options when you attempt to create a slot with rigid joint. In all of the following cases, the newly created slot with rigid joint allows translation in the direction of the z-axes of the two Coords. Note that in any of the cases below, Working Model 3D arranges bodies and/or Coords so that the z-axes of the Coords are collinear upon creating the joint.

You can change the translation axis to be the y-axis or z-axis as well after you create the constraint. Please see “Properties of a Rigid Joint on Slot” on page 8–10 for more information.

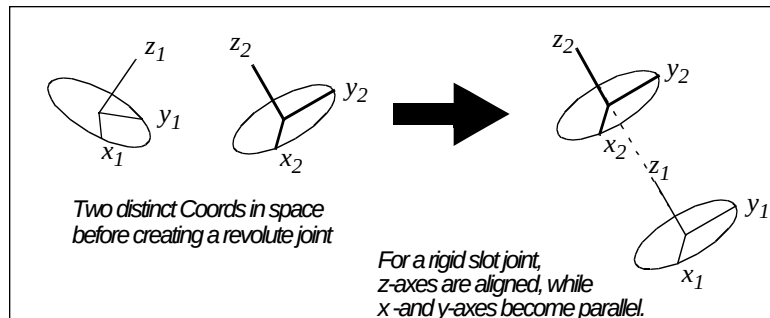
Assembly Behavior of a Rigid Joint on Slot

Selected Objects	Join Option	Behavior
Two Coords attached to bodies	Join body in place	Moves the first Coord so that its z-axis is collinear with that of the second and that x and y-axes are parallel. Connects the two bodies with the slot joint without moving them.
	Don't move anything	Creates a rigid slot joint but does not move a Coord or body. You need to join the constraint later for it to be active.
	Join	Moves the first body so that the z-axes of the two Coords are collinear and x and y-axes are parallel.
	Face-to-face	Rotates the first Coord 180° with its x-axis fixed and then moves the first body so that the z-axes of the two Coords are collinear and y-axes are parallel.
One Coord attached to a body	Join in place	Creates another Coord attached to the background with the same global configuration as the selected Coord. Connects the two Coords with the joint.
One body	Join in place	Creates two Coords at the coordinate origin of the body with the same configuration. Connects the two Coords with the joint.
One Coord attached to a body and one lone body	Join in place	Creates another Coord attached to the lone body with the same global configuration as the selected Coord. Connects the two Coords with the joint.

When a rigid slot joint is created from two existing Coords, Working Model 3D aligns the z-axes, and rotates the Coords so that the z- and y-axes are parallel, as shown in Figure 8-5. The attached bodies are free to translate (slide) along the x-axes (x_1, x_2) but have no rotational degrees of freedom.

Figure 8-5

Aligning Coords for Rigid Joint on Slot

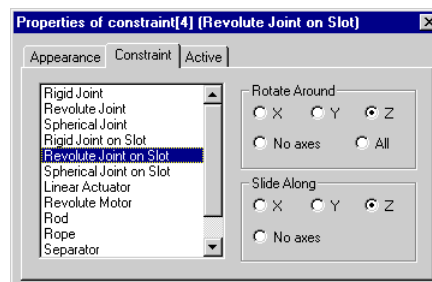


Properties of a Rigid Joint on Slot

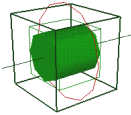
A rigid slot joint allows translation in one of the x-, y-, or z-axes. By default, the constraint allows translation along the z-axis, and the Properties window provides radio buttons to change the translation axis (Figure 8-6). Also, see "The Constraint Properties Window" on page 8-3 for additional properties that are common to all constraints.

Figure 8-6

Properties Window Page for a Rigid Joint on Slot



If you change the translation axis, Working Model 3D splits the constraint immediately. The subsequent join rearranges the bodies so that the bodies and attached Coords satisfy the constraint condition.



8.6 Revolute Joint

A revolute joint allows a single degree of freedom in rotation. Bodies connected with a revolute joint can rotate about a fixed axis with respect to each other.

Assembly Behavior of a Revolute Joint

By default, a revolute joint allows rotation about the z-axis of the Coords. When the constraint is created, Working Model 3D arranges bodies and/or Coords so that the origins of the Coords coincide and the z-axes of the Coords are collinear and in the same direction.

You can change the translation axis after you create the constraint. Please see “Properties of a Revolute Joint” on page 8–13 for more information.

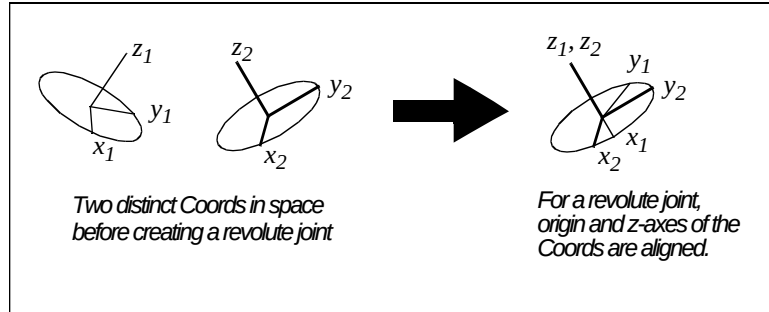
The Create Constraint window presents the following options when you select objects to create a revolute joint.

Assembly Behavior of a Revolute Joint

Selected Objects	Join Option	Behavior
Two Coords attached to bodies	Join bodies in place	Moves the first Coord so that the origins of the two Coords coincide and their z-axes are collinear.
	Split constraint	Does not move a Coord or body. You need to join the constraint later for it to be active. When the constraint is joined, the assembly behavior follows the "Join" option below.
	Join	Moves the first body so that the origins of the two Coords coincide and the z-axes are collinear.
	Face-to-face	Rotates the first Coord 180° with its x-axis fixed, and moves the first body so that the origins of the two Coords coincide and the z-axes are collinear.
One Coord attached to a body	Join in place	Creates another Coord attached to the background with the same global configuration as the selected Coord. Connects the two with a revolute joint.
One body	Join in place	Creates two Coords at the coordinate origin of the body with the same configuration. Connects the two with a revolute joint.
One Coord attached to a body and one lone body	Join in place	Creates another Coord attached to the lone body with the same global configuration as the selected Coord. Connects the two with a revolute joint.

When a revolute joint is created from two existing Coords, Working Model 3D aligns the z-axes and the origin, and matches the directions as shown in Figure 8-7. The attached bodies are free to rotate about the z-axes but have no translational degrees of freedom relative to each other.

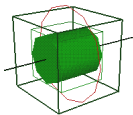
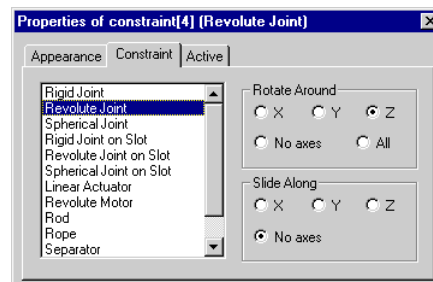
Figure 8-7
Aligning Coords for Revolute Joint



Properties of a Revolute Joint

A revolute joint allows rotation in one of the x-, y-, or z-axes of the Coords. By default, the constraint allows rotation about the z-axis. The Properties window provides radio buttons to change the rotation axis (Figure 8-8). Also, see "The Constraint Properties Window" on page 8-3 for additional properties that are common to all constraints.

Figure 8-8
Properties Window Page for a Revolute Joint



8.7 Revolute Joint on Slot

A slot with revolute joint allows two degrees of freedom, one in translation and the other in rotation. Bodies connected with a slot with revolute joint can not only slide but also rotate with respect to each other.

Assembly Behavior Revolute Joint on Slot

When you create a slot with revolute joint, the slot is attached along the z-axis of the second Coord (the Coord you selected last before creating the constraint). Initially, a slot with revolute joint allows translation in the direction of the z-axes and rotation about the z-axes of the two Coords.

Therefore, when the constraint is created, Working Model 3D arranges bodies and/or Coords so that the x-axes of the Coords are collinear and the z-axes of the Coords are parallel. You can change the translation and rotation axes after you create the constraint. Please see “Properties of Revolute Joint on Slot” on page 8–15 for more information.

The Create Constraint window presents the following options when you select objects to create a slot with revolute joint.

Assembly Behavior of a Revolute Joint on Slot

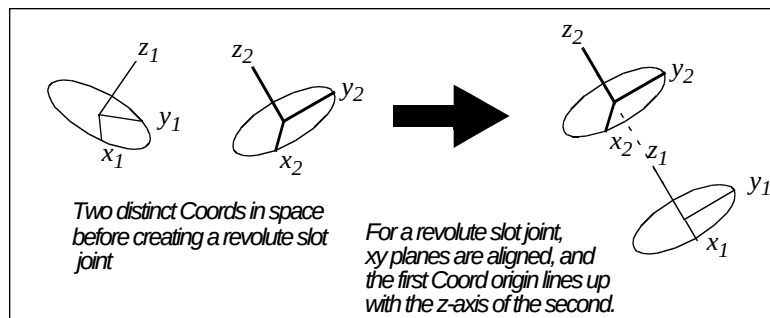
Selected Objects	Join Option	Behavior
Two Coords attached to bodies	Join bodies in place	Moves the first Coord so that its xy-plane is coplanar with that of the second, and the origin of the first Coord lies along the z-axis (the slot) of the second Coord.
	Split constraint	Does not move a Coord or body. You need to join the constraint later for it to be active. When the constraint is joined, the assembly behavior follows “Join” option below.
	Join	Moves the first body so that its xy-plane is coplanar with that of the second, and that the origin of the first Coord lies along the z-axis of the second Coord. Does not move Coords relative to the attached bodies.
	Face-to-face	Rotates the first Coord 180° with its x-axis fixed, and then moves the first body so that its xy-plane is coplanar with that of the second, and that the origin of the first Coord lies along the z-axis (the slot) of the second Coord.

Assembly Behavior of a Revolute Joint on Slot

Selected Objects	Join Option	Behavior
One Coord attached to a body	Join in place	Creates another Coord attached to the background with the same global configuration as the selected Coord. Connect the two with the joint.
One body	Join in place	Creates two Coords at the coordinate origin of the body with the same configuration. Connect the two with the joint.
One Coord attached to a body and one lone body	Join in place	Creates another Coord and attaches it to the lone body, with the same global configuration as the selected Coord. Connect the two with the joint.

When a revolute slot joint is created from two existing Coords, Working Model 3D aligns the xy-planes of the two Coords, and moves the first Coord so that its coordinate origin lies along the z-axis of the second, as shown in Figure 8-9. The attached bodies are free to translate (slide) along the x-axes of the second Coord (x_2) and free to rotate about the z-axes of the two Coords.

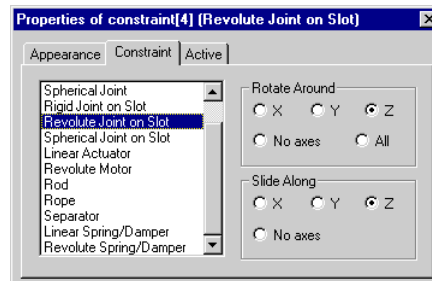
Figure 8-9
Aligning Coords for Revolute Joint on Slot

**Properties of Revolute Joint on Slot**

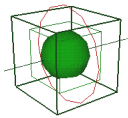
A slot with revolute joint allows translation in one of the x-, y-, or z-axis, and rotation in one of the x-, y- or z-axes. The axis refers to that of the second Coord you selected when you created the constraint.

By default, the constraint allows translation along the z-axis and rotation about the z-axis. The Properties window provides radio buttons to change the translation and rotation axes (Figure 8-10). Also, see "The Constraint Properties Window" on page 8-3 for additional properties that are common to all constraints.

Figure 8-10
*Properties Window Page
for a Revolute Joint on Slot*



If you change the translation or rotation axis, Working Model 3D splits the constraint immediately. The subsequent join rearranges the bodies so that the bodies and attached Coords satisfy the constraint condition.



8.8 Spherical Joint

A spherical joint allows all three degrees of freedom in rotation. Bodies connected with a spherical joint can rotate freely with respect to each other.

Assembly Behavior of a Spherical Joint

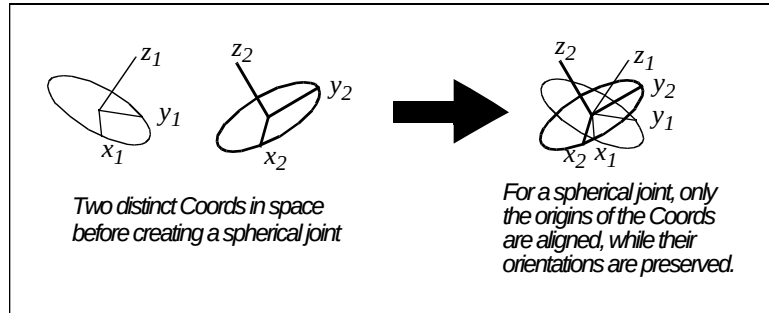
When the constraint is created, Working Model 3D arranges bodies and/or Coords so that the origins of the Coords coincide.

The Create Constraint window presents the following options when you select objects to create a slot with spherical joint.

Assembly Behavior of a Spherical Joint

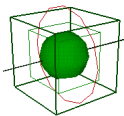
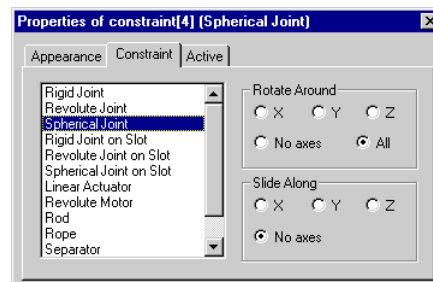
Selected Objects	Join Option	Behavior
Two Coords attached to bodies	Join bodies in place	Moves the first Coord so that the origins of the two Coords coincide.
	Split constraint	Does not move a Coord or body. You need to join the constraint later for it to be active. When the constraint is joined, the assembly behavior follows the "Join" option below.
	Join	Moves the first body so that the origins of the two Coords coincide.
	Face-to-face	Rotates the first Coord 180° with its x-axis fixed, and moves the first body so that the origins of the two Coords coincide.
One Coord attached to a body	Join in place	Creates another Coord attached to the background with the same global configuration as the selected Coord. Connects the two with a spherical joint.
One body	Join in place	Creates two Coords at the coordinate origin of the body with the same configuration. Connects the two with a spherical joint.
One Coord attached to a body and one lone body	Join in place	Creates another Coord attached to the lone body with the same global configuration as the selected Coord. Connects the two with a spherical joint.

When a revolute joint is created from two existing Coords, Working Model 3D simply aligns the origins of the two as shown in Figure 8-11. The attached bodies are free to rotate about all axes but have no translational degrees of freedom relative to each other.

Figure 8-11*Aligning Coords for Spherical Joint*

Properties of a Spherical Joint

A spherical joint allows rotation in all of the x-, y-, or z-axes but does not allow translation along any of the axes. Therefore, the Properties window (Figure 8-12) does not provide any control property. Also, see "The Constraint Properties Window" on page 8-3 for additional properties that are common to all constraints.

Figure 8-12*Properties Window Page for a Spherical Joint*

8.9 Spherical Joint on Slot

A spherical joint on slot allows four degrees of freedom: one in translation and three in rotation. Bodies connected with a spherical joint on slot can not only slide but also rotate freely with respect to each other.

Assembly Behavior Spherical Joint on Slot

When you create a spherical joint on slot, the slot is attached along the z-axis of the second Coord (the Coord you selected last before creating the constraint). Initially, a slot with spherical joint allows translation in the direction of the z-axis and rotation about all axes.

Therefore, when the constraint is created, Working Model 3D arranges bodies and/or Coords so that the origin of the first Coord lies along the z-axis of the second. You can change the translation axis after you create the constraint. Please see “Properties of a Spherical Joint on Slot” on page 8–20 for more information.

The Create Constraint window presents the following options when you select objects to create a spherical joint on slot.

Assembly Behavior of a Spherical Joint on Slot

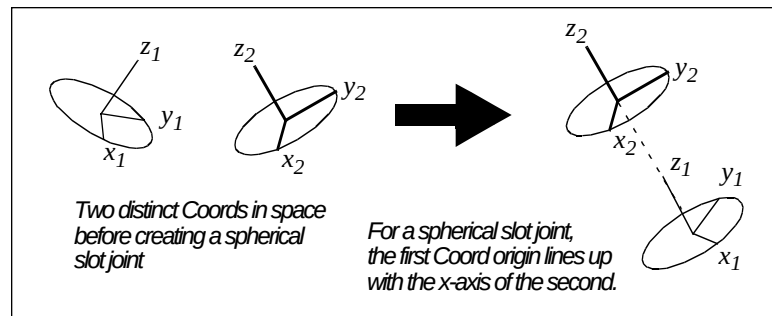
Selected Objects	Join Option	Behavior
Two Coords attached to bodies	Join bodies in place	Moves the first Coord so that the origin of the first Coord lies along the z-axis (the slot) of the second Coord.
	Split constraint	Does not move a Coord or body. You need to join the constraint later for it to be active. When the constraint is joined, the assembly behavior follows “Join” option below.
	Join	Moves the first body so that the origin of the second Coord lies along the z-axis of the first Coord.
	Face-to-face	Rotates the first Coord 180° with its x-axis fixed, and then moves the first body so that the origin of the first Coord lies along the z-axis (the slot) of the second Coord.
One Coord attached to a body	Join in place	Creates another Coord attached to the background with the same global configuration as the selected Coord. Connect the two with the joint.

Assembly Behavior of a Spherical Joint on Slot

Selected Objects	Join Option	Behavior
One body	Join in place	Creates two Coords at the coordinate origin of the body with the same configuration. Connect the two with the joint.
One Coord attached to a body and one lone body	Join in place	Creates another Coord and attaches it to the lone body, with the same global configuration as the selected Coord. Connect the two with the joint.

When a spherical slot joint is created from two existing Coords, Working Model 3D moves the first Coord so that its coordinate origin lies along the x-axis of the second, as shown in Figure 8-13. The attached bodies are free to translate (slide) along the z-axes of the second Coord (x_2) and free to rotate about the all axes.

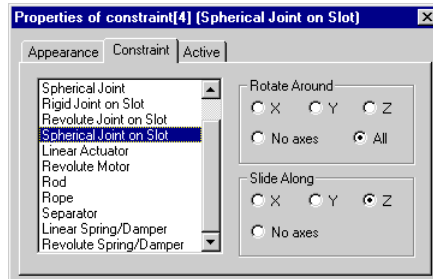
Figure 8-13
Aligning Coords for Spherical Slot Joint

***Properties of a Spherical Joint on Slot***

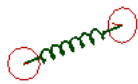
A spherical joint on slot allows translation in one of the x-, y-, or z-axis, and rotation in all axes. The axis refers to that of the second Coord you selected when you created the constraint.

By default, the constraint allows translation along the z-axis and rotation about all axis. The Properties window provides radio buttons to change the translation axis (Figure 8-14). Also, see "The Constraint Properties Window" on page 8-3 for additional properties that are common to all constraints.

Figure 8-14
*Properties Window
for a Spherical Joint on Slot*



If you change the translation axis, Working Model 3D splits the constraint immediately. The subsequent join rearranges the bodies so that the bodies and attached Coords satisfy the constraint condition.



8.10 Linear Spring/Damper

A linear spring/damper is a combination of a linear spring and linear damper. The constraint applies forces at its endpoints that are equal in magnitude and opposite in direction but applies no torque. A linear spring/damper does not eliminate any degree of freedom. You can set the damping coefficient to zero to create a pure spring, or set the spring coefficient to zero to create a pure damper.

Assembly Behavior of a Linear Spring/Damper

The Create Constraint window presents the following options when you attempt to create a linear spring/damper.

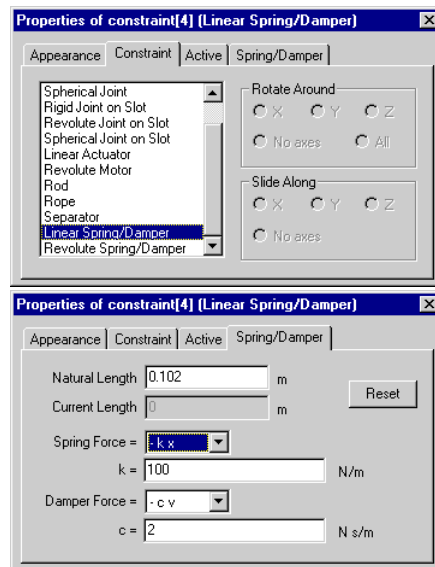
Assembly Behavior of a Linear Spring/Damper

Selected Objects	Join Option	Behavior
Two Coords attached to bodies	Don't move anything	Creates a spring/damper between the two Coords.
One Coord attached to a body	Join in place	Creates another Coord attached to the background with the same global configuration as the selected Coord. Creates a zero-length spring/damper with zero natural length.
One body	Join in place	Creates two Coords at the coordinate origin of the body with the same configuration. Creates a zero-length spring/damper.
One Coord attached to a body and one lone body	Join in place	Creates another Coord attached to the lone body with the same global configuration as the selected Coord. Creates a zero-length spring/damper.

Properties of a Linear Spring/Damper

As shown in the Constraint Properties page (Figure 8-15), a spring/damper has the following properties:

Figure 8-15
 Properties Window Pages
 for a Spring/Damper



Natural Length and Current Length

The *natural length* refers to the unloaded, rest length of the spring. The number shown immediately below the natural length is the *current length* of the spring/damper. Just like in a real physical spring:

- If the current length is greater than the natural length, the spring is in tension.
- If the current length is less than the natural length, the spring is in compression.

When you create a spring/damper, Working Model 3D automatically calculates the distance between the two Coords that serve as the spring/damper endpoints, and automatically sets the natural length to be equal to the current length. Therefore, the spring is not loaded by default.

To pre-load a spring/damper with compression or tension prior to the simulation, simply specify its natural length so that the difference between the current length and the natural length generates an appropriate force. Also see “Spring Characteristics” below.

When you drag one of the bodies connected with a spring/damper, the current length may be different from the natural length. You can quickly set the natural length to be equal to the current length by clicking the reset button (see Figure 8-15).

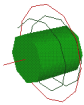
Spring Characteristics

The direction and magnitude of the spring force is determined by the distance between its two endpoints. You can set the spring force characteristics via force equation and spring constant. The default force equation is $F = -kx$ (a linear spring), where x is the difference between the natural length and the current length. You can also change the response to $F = -kx^2$, $F = -kx^3$, $F = -\frac{k}{x}$, and $F = -\frac{k}{x^2}$.

Damper Characteristics

The direction and magnitude of the damper force is determined by the relative velocity between its two endpoints. You can set the damper force characteristics via the force formula and the damping constant. The default force equation is $F = -cv$ (a linear damper), where v is the relative speed between the endpoints. You can also change the response to $F = -cv^2$, $F = -cv^3$, $F = -\frac{c}{v}$, and $F = -\frac{c}{v^2}$.

Also, see "The Constraint Properties Window" on page 8-3 for additional properties that are common to all constraints.



8.11 Revolute Spring/Damper

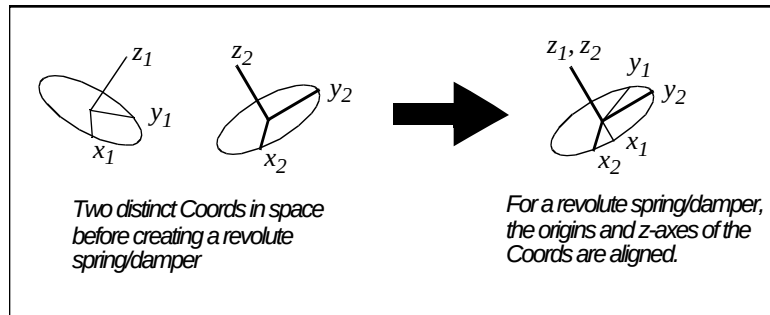
A revolute spring/damper is essentially a combination of a revolute joint, torsion spring, and a torsion damper. Like a revolute joint, a revolute spring/damper allows a single degree of freedom in rotation. Bodies connected with a revolute joint can rotate with respect to each other, while a revolute spring/damper applies torque determined by the relative orientation and/or angular velocity determined from the bodies.

Assembly Behavior of a Revolute Spring/Damper

Since a revolute spring/damper is no more than a revolute joint with a torsion damper and a torsion spring combined, the assembly behavior of a revolute spring/damper is identical to that of a revolute joint. Please refer to "Linear Spring/Damper" on page 8-21 for information.

When a revolute spring/damper is created from two existing Coords, Working Model 3D aligns the z-axes and the origin, and matches the directions as shown in Figure 8-16. The attached bodies can rotate about the z-axes but have no translational degrees of freedom relative to each other.

Figure 8-16
Aligning Coords for Revolute Spring/Damper



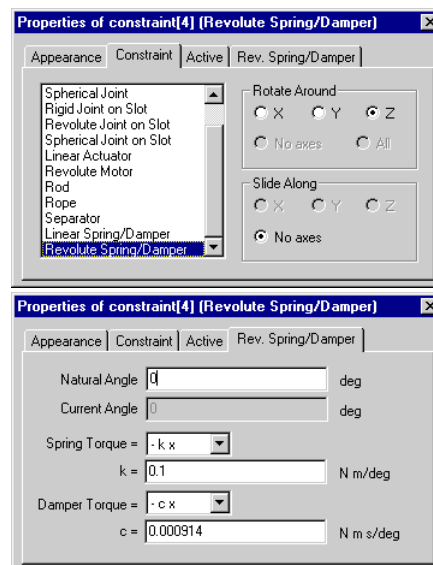
You can change the rotation axis after you create the constraint. Please see below for more information.

Properties of a Revolute Spring/Damper

A revolute spring/damper allows rotation in one of the x-, y-, or z-axes. By default, the constraint allows rotation about the z-axis. The Constraint Properties page of the Properties window provides radio buttons to change the rotation axis (Figure 8-17).

In addition, a revolute spring/damper has the following properties:

Figure 8-17
Properties Window Pages for a Revolute Spring/Damper



Natural Angle and Current Angle

The *natural angle* refers to the natural, unloaded rotation angle of the torsion spring. The number shown immediately below the natural angle is the *current angle* of the revolute spring/damper. The sign convention is as follows:

- If the current angle is greater than the natural angle, the torsion spring exerts a torque counterclockwise, when observed head-on from the rotation axis of the second Coord.
- If the current angle is less than the natural angle, the torsion spring exerts a torque in clockwise direction, when observed head-on from the rotation axis of the second Coord.

When you create a spring/damper, Working Model 3D automatically calculates the orientation between the two Coords that serve as the spring/damper endpoints, and automatically sets the natural angle to be equal to the current angle. Therefore, the spring is unloaded by default.

To pre-load a spring/damper with compression or tension prior to the simulation, simply specify its natural angle so that the difference between the current angle and the natural angle results in the desired torque. Also see the descriptions of the spring characteristics below.

When you change the orientation of bodies connected with a spring/damper, the current angle may be different from the natural angle. You can quickly set the constraint's natural angle to be equal to the current angle by clicking the reset button (see Figure 8-17).

Spring Characteristics

The direction and magnitude of the spring torque is determined by the relative angle between its two endpoints. You can set the spring torque characteristics via the torque equation and spring constant. The default torque equation is $\tau = -k\phi$ (proportional correlation), where ϕ is the difference between the natural angle and the current angle. You can also change the response to $\tau = -k\phi^2$, $\tau = -k\phi^3$, $\tau = -\frac{k}{\phi}$, and $\tau = -\frac{k}{\phi^2}$.

Damper Characteristics

The direction and magnitude of the damper torque is determined by the relative angular velocity between its two Coords. You can set the damper torque characteristics via the torque formula and the damping constant. The default torque equation is $\tau = -c\omega$ (proportional correlation), where ω is the relative angular velocity between the Coords. You can also change the response to $\tau = -c\omega^2$, $\tau = -c\omega^3$, $\tau = -\frac{c}{\omega}$, and $\tau = -\frac{c}{\omega^2}$.

Also, see "The Constraint Properties Window" on page 8-3 for additional properties that are common to all constraints.



8.12 Linear Actuator

A linear actuator is a multi-purpose constraint which exerts whatever force necessary to maintain the specified length, velocity, or acceleration between its endpoint Coords. A linear actuator applies forces at the endpoints that are equal in magnitude and opposite in direction, but applies no torque; an actuator does not limit any degrees of freedom.

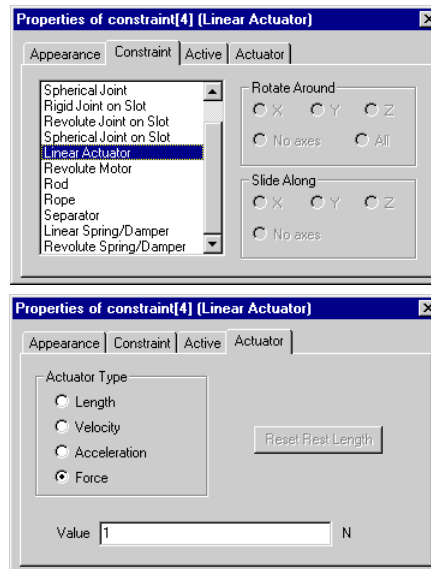
Assembly Behavior of a Linear Actuator

The assembly behavior of a linear actuator is identical to that of a spring/damper. Please refer to "Assembly Behavior of a Linear Spring/Damper" on page 8-22 for information.

Properties of a Linear Actuator

As shown in the Constraint Properties page (Figure 8-18), a linear actuator has the following properties:

Figure 8-18
*Properties Window Pages
 for a Linear Actuator*



Actuator Type

The actuator exerts as much force as necessary to maintain the given condition. You can specify the constraint condition in one of the four terms as follows:

- Length
- Velocity
- Acceleration
- Force

A **force** actuator exerts a force specified in the Value edit box. A **length** actuator exerts the necessary force to maintain the length specified in the Value edit box. A **velocity** actuator exerts the necessary force to maintain the velocity specified in the Value edit box. An **acceleration** actuator exerts the necessary force to maintain the acceleration specified in the Value edit box.

Value

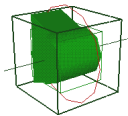
In this edit box, you can specify the value associated with the actuator type selected above. Working Model 3D interprets the value in the current unit system. For force, velocity, and acceleration actuators, a negative value instructs the actuators to compress, whereas a positive

value instructs them to extend. Length actuators behave so that the difference between the current length and the specified length diminishes as quickly as possible, but not instantly.

You can also type a formula expression instead of a numerical constant. For example, you could set the actuator's velocity to be time dependent or combined with conditional statements. See Chapter 12, "Using Formulas" for more examples.

NOTE: When you specify the linear actuator force, length, velocity, or acceleration, make sure that the actual length of the actuator does not become zero during the simulation. A zero length actuator will produce unpredictable simulation results.

Also, see "The Constraint Properties Window" on page 8-3 for additional properties that are common to all constraints.



8.13 Revolute Motor

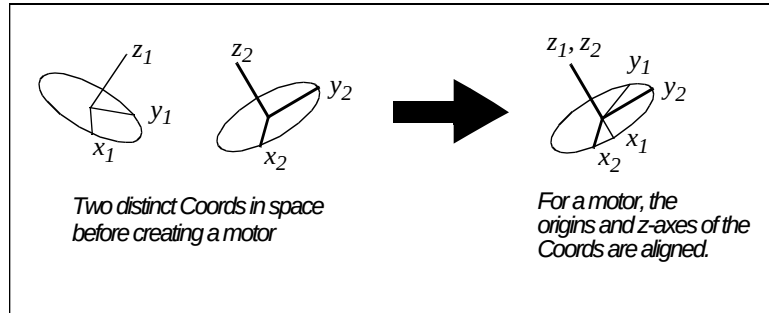
A revolute motor has a built-in revolute joint. Accordingly, a revolute motor allows a single degree of freedom in rotation. A revolute motor functions as a multi-purpose constraint that exerts the torque necessary to maintain the specified angle, angular velocity, or angular acceleration between the two bodies connected.

Assembly Behavior of a Revolute Motor

Since a motor is no more than a revolute joint with a motor combined, its assembly behavior is identical to that of a revolute joint. Please refer to "Linear Spring/Damper" on page 8-21 for information.

When a motor is created from two existing Coords, Working Model 3D aligns the z-axes and the origin, and matches the directions as shown in Figure 8-19. The attached bodies can rotate about the z-axes but have no translational degrees of freedom relative to each other.

Figure 8-19
Aligning Coords for Motor



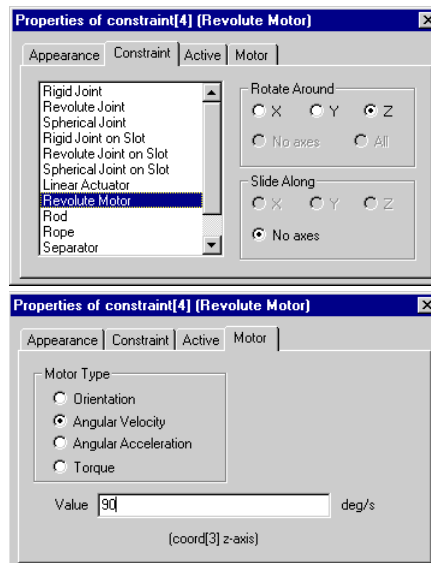
You can change the rotation axis after you create the constraint. Please see “Properties of a Revolute Motor” on page 8–30 for more information.

Properties of a Revolute Motor

A motor allows rotation in one of the x-, y-, or z-axes. By default, the constraint allows rotation about the z-axis. The Constraint Properties page of the Properties window provides radio buttons to change the rotation axis (Figure 8-20).

In addition, a motor has the following properties:

Figure 8-20
Properties Window for a Motor



Motor Type

The motor exerts as much torque as necessary to maintain the given condition. You can specify the constraint condition in one of the four terms as follows:

- Torque
- Rotation
- Velocity
- Acceleration

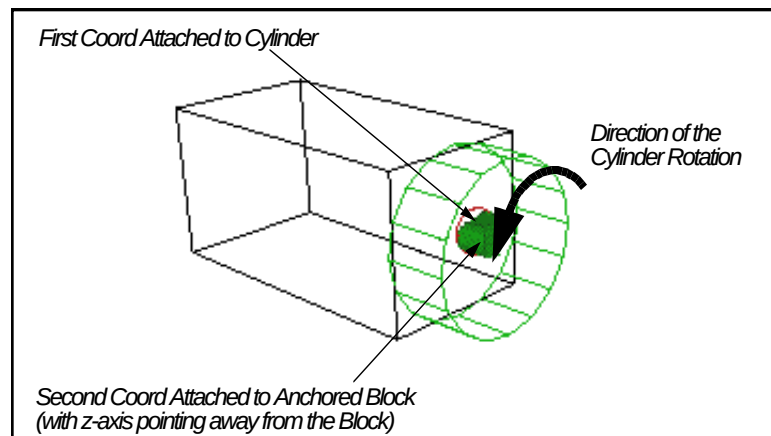
A **torque** motor exerts a torque specified in the Value edit box. A **rotation** motor exerts the necessary torque to maintain the angle specified in the Value edit box. A **velocity** motor exerts the necessary torque to maintain the angular velocity specified in the Value edit box. An **acceleration** motor exerts the necessary torque to maintain the angular acceleration specified in the Value edit box.

Value

In this edit box, you can specify the value associated with the motor type selected above. Working Model 3D interprets the value in the current unit system. You can also type a formula expression instead of a numerical constant. For example, you could set the motor's angular velocity to be time dependent or combined with conditional statements. See Chapter 12, "Using Formulas" for more examples.

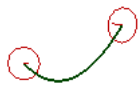
In principle, Working Model 3D considers the second Coord selected (when creating the motor) as the base of the motor, and interprets the motor specification accordingly. For example, if you specify the velocity motor with 20 deg/sec of angular velocity with the model shown in Figure 8-21, the cylinder rotates in the direction shown with an arrow.

Figure 8-21
Revolute Motor with Positive
Angular Velocity Specification



Also, see "The Constraint Properties Window" on page 8–3 for additional properties that are common to all constraints.

Note: Rotations must stay between -2π and 2π . Rotation motors cannot “wrap”. Use a velocity motor instead.



8.14 Rope

A rope applies forces at its endpoints so that they do not become apart beyond the specified distance, or the *rope length*. A rope applies no force at all when the distance between the endpoints are less than the rope length. A rope applies forces at the endpoints that are equal in magnitude and opposite in direction, while no torque is applied; a rope does not limit any degrees of freedom.

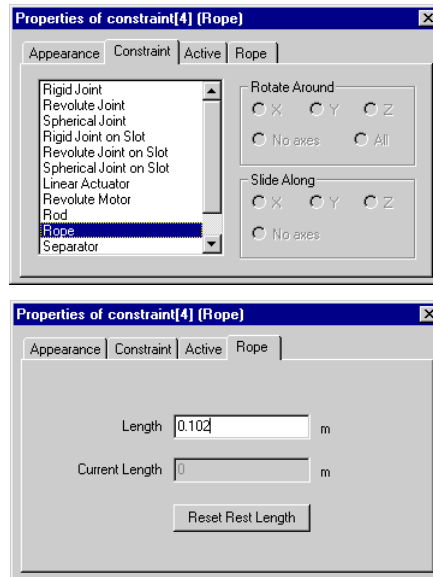
Assembly Behavior of a Rope

The assembly behavior of a rope is identical to that of a linear spring/damper. Please refer to "Assembly Behavior of a Linear Spring/Damper" on page 8–22 for information.

Properties of a Rope

As shown in the Constraint Properties page (Figure 8-22), a rope has the following properties:

Figure 8-22
*Properties Window Pages
 for a Rope*



Length and Current Length

The *length* refers to the stretched length of the rope. The number shown immediately below the rest length is the *current length* of the rope. Just like in a real, physical rope:

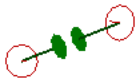
- If the length is greater than the current length, the rope is slack and applies no force.
- The rope's current length never exceeds the rope length (unless you so specify at the start of the simulation; see below).

When you create a rope, Working Model 3D automatically calculates the distance between the two Coords that serve as the rope endpoints, and automatically sets the rope length to be equal to the current length. Therefore, the rope is taut but not overstretched by default.

If you specified the rope length to be less than the current length before starting the simulation (i.e., overstretched rope), the rope will apply the necessary force to diminish the difference between the two lengths.

When you drag one of the bodies connected with a rope, the current length may be different from the rest length. You can quickly set the rest length to be equal to the current length by clicking the Reset Rest Length button (see Figure 8-22).

Also, see "The Constraint Properties Window" on page 8-3 for additional properties that are common to all constraints.



8.15 Separator

A separator applies forces at its endpoints so that they do not become closer than the specified distance, or the *separator length*. A separator applies no force at all when the distance between the endpoints are greater than the separator length. Separator forces at the endpoints are equal in magnitude and opposite in direction, while no torque is applied; a separator does not limit any degrees of freedom.

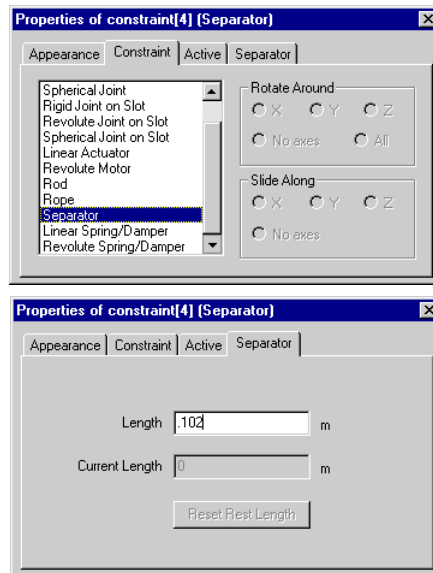
Assembly Behavior of a Separator

The assembly behavior of a separator is identical to that of a linear spring/damper. Please refer to "Assembly Behavior of a Linear Spring/Damper" on page 8-22 for information.

Properties of a Separator

As shown in the Constraint Properties page (Figure 8-23), a separator has the following properties:

Figure 8-23
Properties Window for a Separator



Length and Current Length

The *length* refers to the minimum clearance of the separator; the two endpoints of the separator will not be closer than this length. The number shown immediately below the length is the *current length* of the separator. The separator behaves so that:

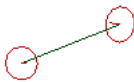
- If the current length is greater than the length, the separator applies no force.
- The separator's current length will never become less than the separator length (unless you so specify at the start of the simulation; see below).

When you create a separator, Working Model 3D automatically calculates the distance between the two Coords that serve as the separator endpoints, and automatically sets the separator length to be equal to the current length. Therefore, the separator is snug but not overcompressed by default.

If you specify the separator length to be greater than the current length before starting the simulation (i.e., overcompressed separator), the separator will apply the necessary force to diminish the difference between the two lengths; i.e., the connected objects will be repelled.

When you drag one of the bodies connected with a separator, the current length may be different from the rest length. You can quickly set the rest length to be equal to the current length by clicking the Reset Rest Length button (see Figure 8-23).

Also, see "The Constraint Properties Window" on page 8-3 for additional properties that are common to all constraints.



8.16 Rod

A rod constantly applies forces at its endpoints to maintain the specified distance, or the *rod length*. Rod forces at the endpoints are equal in magnitude and opposite in direction, while no torque is applied; a rod does not limit any degrees of freedom.

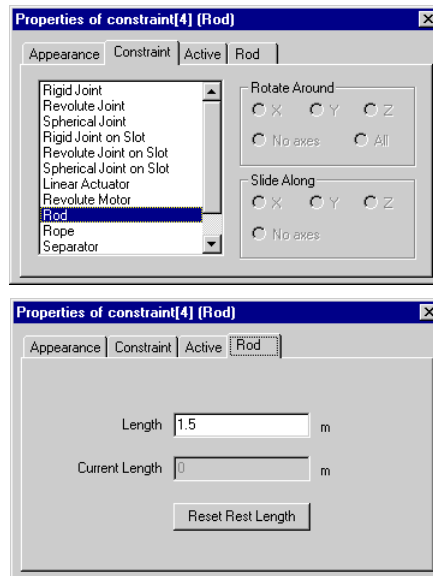
Assembly Behavior of a Rod

The assembly behavior of a rod is identical to that of a linear spring/damper. Please refer to "Assembly Behavior of a Linear Spring/Damper" on page 8-22 for information.

Properties of a Rod

As shown in the Constraint Properties page (Figure 8-24), a rod has the following properties:

Figure 8-24
Properties Window for a Rod



Length and Current Length

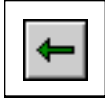
The *length* refers to the rod length. The number shown immediately below the length is the *current length* of the rod. As long as the rod length and the current length are equal at the beginning of the simulation, the distance between the rod endpoints is kept constant.

When you create a rod, Working Model 3D automatically calculates the distance between the two Coords that serve as the rod endpoints, and automatically sets the rod length to be equal to the current length.

If you specified the rod length to be different from the current length before starting the simulation (i.e., “stretched” or “compressed” rod), the rod will apply the necessary force to diminish the difference between the two lengths.

When you drag one of the bodies connected with a rod, the current length may be different from the rod length. You can quickly set the rod length to be equal to the current length by clicking the reset button (see Figure 8-24).

Also, see “The Constraint Properties Window” on page 8–3 for additional properties that are common to all constraints.



8.17 Force

In Working Model 3D, a force is a special constraint that allows you to specify external point load to any body in your model. You can apply as many forces to as many bodies as you desire.

Unlike all the other constraints (except torques), a force has only one Coord, which acts as the *point of application*.

Attaching Force

You can attach a force to any body by simply clicking it.

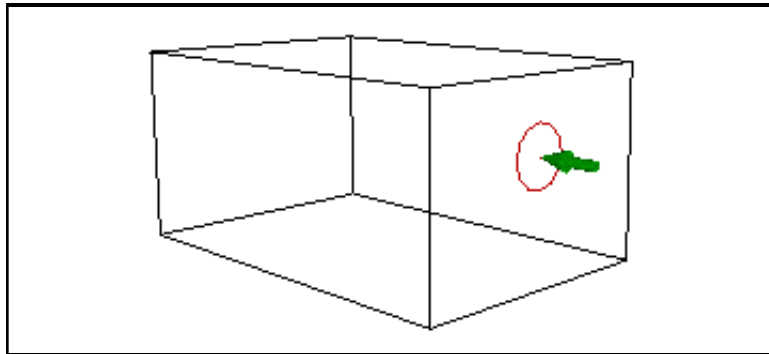
► **To attach a force to a body:**



1. Click the Force tool on the Sketch toolbar.
2. Click the body to which you would like to apply the force.

Working Model 3D attaches a Coord that shows the Force icon (Figure 8-25).

Figure 8-25
Force Constraint



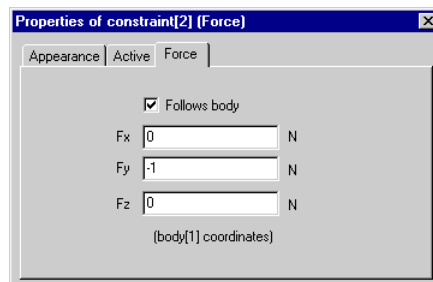
3. To modify the attachment point (point of application), simply double-click the base Coord of the force and modify its position through the Properties window.

See “Modifying Constraint Properties” on page 7-22 for more information.

Properties of Force

A force constraint allows you to control the xyz components independently. Simply double-click the force constraint icon to open the Properties window (Figure 8-26).

Figure 8-26
Force Properties Page
for Force



The Properties window provides the following controls:

- Fx, Fy, and Fz components
- the Follow Body option

Fx, Fy, and Fz components specify the force components in the global coordinates, unless the **Follow Body** option is checked. If the option is checked, the components are specified in the attached body's frame.

In the components edit boxes, you can also type a formula expression instead of a numerical constant. For example, you could set the force to be time dependent or combined with conditional statements. See Chapter 12, "Using Formulas" for more examples.



8.18 Torque

In Working Model 3D, a torque is a special constraint that allows you to specify external torque to any body in your model. You can apply as many torques to as many bodies as you desire.

Unlike all the other constraints (except forces), a torque has only one Coord, called the *base Coord*.

Attaching Torque

You can attach a torque to any body by simply clicking it.

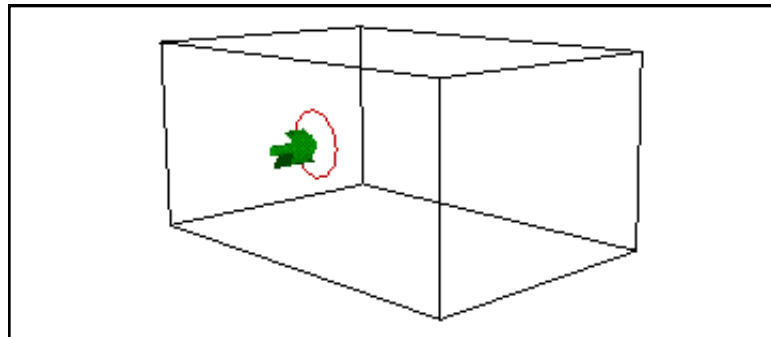
► **To attach a torque to a body:**



1. Click the Torque tool on the Sketch toolbar.
2. Click the body to which you would like to apply the torque.

Working Model 3D attaches a Coord that shows the torque icon (Figure 8-27).

Figure 8-27
Torque Constraint



3. To modify the attachment point (point of application), simply double-click the base Coord of the force and modify its position or orientation through the Properties window.

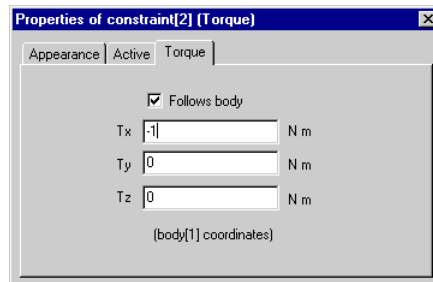
See “Modifying Constraint Properties” on page 7–22 for more information.

NOTE: As with dynamics, the position and orientation of the torque’s base Coord has no significance on the simulation result.

Properties of Torque

A torque constraint allows you to control the xyz components independently. Simply double-click the torque constraint icon to open the Properties window (Figure 8-28).

Figure 8-28
Torque Properties Window



The Properties window provides the following controls:

- Tx, Ty, and Tz components
- the Follow Body option

Tx, Ty, and Tz components specify the torque components in the World coordinates, unless the **Follow Body** option is checked. If the option is checked, the components are specified in the attached body's frame.

In the components edit boxes, you can also type a formula expression instead of a numerical constant. For example, you could set the torque to be time dependent or combined with conditional statements. See Chapter 12, "Using Formulas" for more examples.

CHAPTER 9

Running Simulations

This chapter will show you how to:

- run and stop simulations
- control simulation parameters
- optimize your simulation for speed or accuracy

9.1 Opening, Saving, and Closing Files

File operations in Working Model 3D are the same as most other Windows applications.

► **To create a new Working Model 3D file:**

1. **Choose New in the File menu.**

A blank Working Model 3D document appears.

► **To save a new Working Model 3D file:**

1. **Choose Save As in the File menu.**

The Save As dialog appears and prompts for the file name and directory location.

2. **Enter the desired filename and directory in which the file will be stored.**

Working Model 3D will add the “.WM3” extension by default.

3. **Click OK.**

The Save As dialog disappears and the file is saved.

NOTE: Working Model 3D saves the simulation history. Please see “Erasing Simulation History” on page 9-4 for more information.

► **To save a Working Model 3D file:**

1. Choose Save in the File menu.

► **To close a Working Model 3D file:**

1. Choose Close in the File menu.

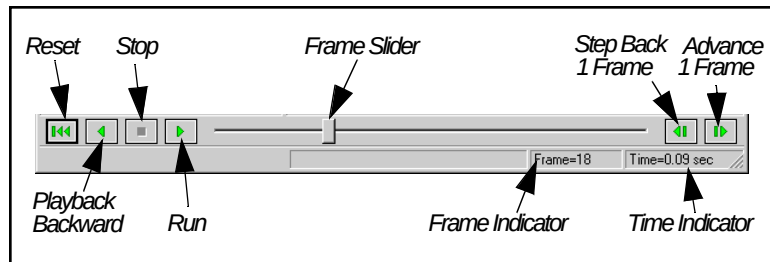
If the document has been modified in any way since the last time it was saved, Working Model 3D prompts for saving the file before closing the file.

9.2 Running and Stopping a Simulation

Working Model 3D provides the following controls for the simulation (Figure 9-1):

- The Tape Player Control allows you to run, stop, observe each frame at a time, or run a simulation backwards.
- Commands for controlling simulations are also available in the World menu.

Figure 9-1
Tape Player Control





► **To run a simulation:**

1. Click the Run button on the Tape Player Control.

You may see a frame progress indicator in the status bar (below the Tape Player Control) for frames that are difficult to calculate.



► **To stop a simulation:**

1. Click the Stop or Reset button on the Tape Player Control.

The Stop button simply stops the simulation, leaving the last frame calculated, whereas the Reset button stops the simulation and resets to the initial condition.



► **To reset a simulation to the initial condition:**

1. Click the Reset button on the Tape Player Control.



9.3 Recording a Simulation

Simulation History

When you run a simulation of your model for the first time, Working Model 3D stores the data—called *simulation history*—in your computer's memory as it computes each frame. Therefore, when you run the same simulation next time, Working Model 3D only needs to “playback” the simulation history without computing each frame all over again. If an indicator with a number appears in the Tape Player Control near the bottom of the window, the current document has simulation history stored in memory.

Figure 9-2 illustrates several examples.

Figure 9-2
States of Tape Player Control



The simulation history stays valid as long as your model remains unchanged. For example, if you change an initial condition of one of the bodies in your model, Working Model 3D erases the simulation history from memory as the simulation needs to be re-computed entirely.

Erasing Simulation History

When you change the initial conditions of a simulation (e.g., moving a body, changing a property of a constraint), Working Model 3D automatically erases the currently stored simulation history. You can also force Working Model 3D to “forget” the simulation history. For example, you can use the intermediate state of an existing simulation to be the initial condition of a new simulation.

► **To erase the simulation history:**

1. **Choose Erase History in the World menu.**

The Frame Indicator disappears, meaning that the simulation history is discarded.

Alternatively, you can press Ctrl+H to erase history.

Setting Initial Condition from an Existing Simulation

You can use the current condition of a simulation as the initial condition for a new simulation. For example, when you design a vehicle suspension, you can start the simulation by “dropping” the vehicle to the ground, let it reach its stable condition (or steady-state), and use the condition as the initial condition of a next simulation (e.g., starting the vehicle motor).

► ***To use set initial condition from an existing simulation:***

1. **Advance your simulation frame to the desired starting point.**
2. **Choose Erase History from the World menu.**

The current frame becomes frame zero, or the new initial condition. Note that Working Model 3D preserves not only the configuration (position and orientation) but also velocity and acceleration information at the frame. For example, if a body was moving at a certain velocity at the frame where you chose Erase History, the velocity becomes the initial velocity of the body for the next simulation run.

9.4 Playing Back a Simulation

Since Working Model 3D stores the simulation history as it calculates, playing back a simulation playback is faster than running the simulation for the first time, as long as your model remains unchanged. You can tell whether the document has simulation history or not by observing the tape player control (see Figure 9-2).

► ***To play back a simulation history:***

1. **Click the Run button on the Tape Player Control.**



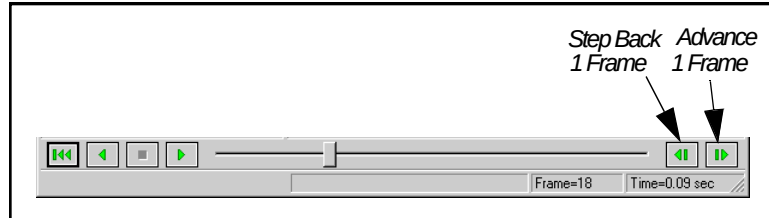
The play back starts, as the tape player indicator progresses.

If the document does not have any simulation history, Working Model 3D simply proceeds to calculate the model. As soon as the simulation history has run out, Working Model 3D starts to calculate the simulation from the last frame on, and accumulates the simulation history.

Frame-by-Frame Playback

Working Model 3D allows you to view the recording of a simulation frame by frame.

Figure 9-3
Frame Advance Buttons



Click on either frame advance button (forward or backward) in the Tape Player Control to move one frame at a time.

Skipping Frames for Faster Animation

You can change the playback speed by skipping one or more frames. Working Model 3D always updates frames as fast as possible, but the simulation duration will be shorter as frames are skipped. This feature is useful for situations such as a camera shutter mechanism, where you recorded a simulation with a very small time step to sustain high accuracy, but wish to play back the demonstration at faster speed.

► To skip frames for faster animation:

1. **Choose Skip Frames in the World menu.**

A submenu appears to show the skip rate options.

2. **Choose the number of frames you wish to skip from the Skip Frames sub menu.**

The default setting is at “1”, indicating that every frame is displayed. The setting at “2” means that every other frame is displayed, and “4” means that one out of every four frames is displayed.

Another way to control the speed of animation is to adjust the time step. Please see “Animation Step” on page 9–10 for more information.

Playing a Simulation Backward

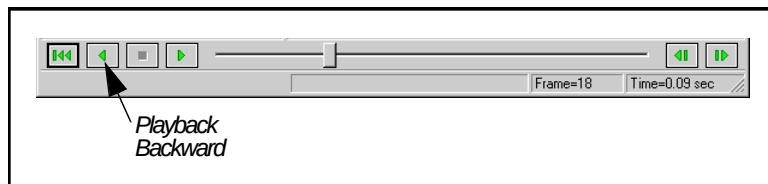
After you have run a simulation, you can play it backward.

► **To play back the simulation backward:**

1. Click the play backward button on the Tape Player Control (Figure 9-4).

The simulation begins running in reverse. You can stop the simulation at any time. The simulation will stop by itself when it reaches the first frame.

Figure 9-4
Play Backward Button



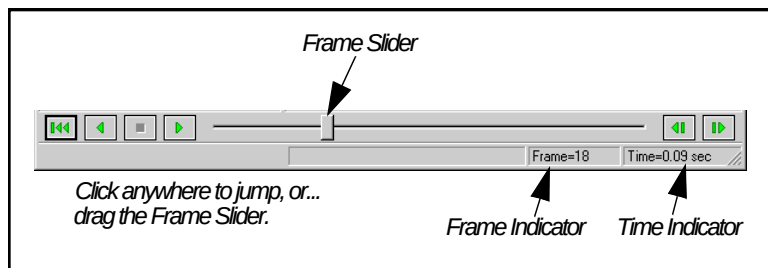
2. Click either the play forward or the play backward control to resume animation.

As the animation runs backward, the frame indicator moves to the left while displaying the number of the current frame of animation.

Moving to a Specific Frame

To move quickly to any frame in a recorded simulation, drag the Frame Slider left or right.

Figure 9-5
Dragging to a Specific Frame



You can also click any portion of the gray region on the Tape Player Controls to immediately move the Frame Slider to that location.

To continue the simulation beyond the current recording, drag the Frame Slider as far to the right as it will go and then click the Run button. Working Model 3D now continues the simulation.

9.5 Printing Simulations

You can print the current simulation frame.

► **To print the simulation:**

1. **Select the window which you would like to print.**
2. **Choose Print in the File menu.**

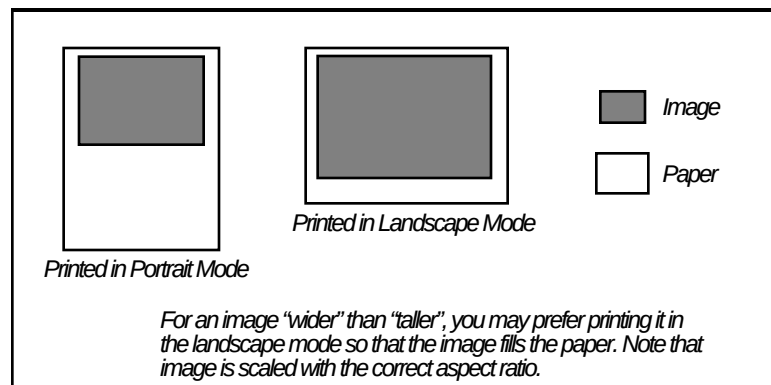
The standard Windows print dialog appears.

3. **Specify the appropriate options, and click the OK button.**

Working Model 3D prints the currently active window. The printed image is maximized on the printed page, while the image's aspect ratio is maintained. The orientation of the printed page (i.e., whether in the portrait mode or landscape mode) follows the Windows page setup settings.

For example, if your window is rectangular, you may prefer printing the image in the landscape mode so that the image spreads over the paper nicely (see Figure 9-6).

Figure 9-6
Printed Image



NOTE: The printed image does *not* include the toolbars, status bar, coordinates bar, menu items, the caption (title bar), or Object Manager.

See "Printing Meter Data" on page 10–9 for instructions on printing meters.

9.6 Controlling Simulation Accuracy

Working Model 3D provides you with complete control of the numerical algorithms used in creating simulations.

For More Information

Section "Accuracy Dialog and Simulation Parameters" on page A–9 provides more information on more advanced options.

Simulation Accuracy Modes

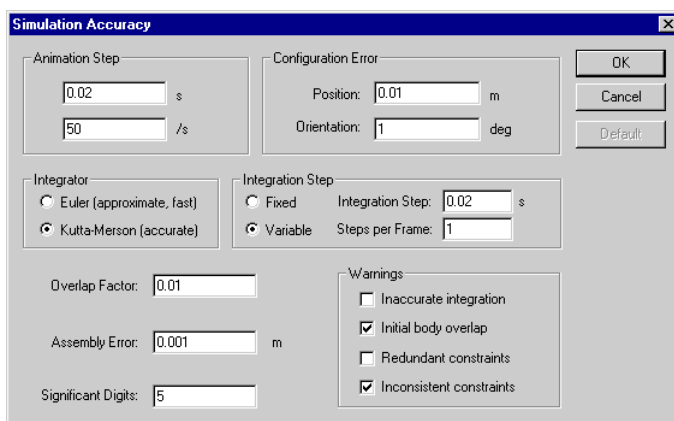
To choose a simulation mode:

1. Choose Accuracy from the World menu.

The Simulation Accuracy dialog appears (Figure 9-7).

2. Set the desired properties.

Figure 9-7
Simulation Accuracy Dialog



You can restore Working Model 3D's default simulation accuracy values at any time by clicking the Default button.

Animation Step

The Animation Step box determines the time interval between animation frames. *This box does not always represent the integration time step.*

By default, Working Model 3D automatically sets the Animation Step to 0.02 second. You can override the automatic animation step decision and set your own time step size. See “Integration Time Step” on page A-12 for a discussion on setting the integration time step.

Configuration Error

The Configuration Error (position and orientation error) controls an error bound for various aspects of simulation. In general, smaller values of Configuration Error (which translates to a tighter error bound) generate more accurate results at the expense of the simulation speed. The value of both Configuration Error parameters must be greater than zero. By default, Working Model 3D automatically chooses appropriate position and orientation tolerances for Configuration Error. For more information, please see “Configuration Error” on page A-11.

Integrator

Euler Mode

The Euler mode is approximate and fast, and employs a simple integration method which optimizes the simulation for speed. The mode is appropriate if you want a rough prototype of your model.

Kutta-Merson Mode

The Kutta-Merson mode is more accurate and employs a more sophisticated integration method. Working Model 3D may require more time to compute simulation results, but they are much more accurate than the ones computed with Fast mode.

Tips for Faster Simulation

Here are some suggestions to improve the simulation speed.

- Use a hardware-accelerated OpenGL video card. Because Working Model 3D relies on OpenGL for most of the graphics, rendering performance will be substantially increased with hardware acceleration.
- Minimize the number of bodies that are set to collide. The less collisions to look for, the faster simulation runs.
- Minimize the number of collisions. The less collisions the model has, the faster simulation runs. For example, instead of using an anchored body to block the motion of another body, consider using a separator (see “Separator” on page 8–34).
- If you are using polyhedra in your model, decrease the complexity of the polyhedra (through your modeling package, or in ACIS Converter), or replace them completely with simpler Working Model 3D’s geometric primitives (e.g., spheres, cylinders, and boxes).
- Minimize the number of constraints.
- Make sure that rigid joints are set to “optimized” unless you need to measure reaction forces and torques at the joints (see “Rigid Joint” on page 8–5).
- Use smaller window size to minimize redraw time.
- Try running the simulation with a large Animation Step first, then cut it down to obtain desired accuracy. Smaller time steps yield more accurate results, but require more computing time per unit simulation time.
- Do not increase the values of Configuration Error arbitrarily. Start with the default values, and increase the tolerances as necessary. You may be able to obtain sufficiently accurate results with relatively large error tolerance. See the example described in Appendix A (“How Working Model 3D Bounds Errors” on page A–5).
- If the hard disk is accessed frequently, the Windows system may be “thrashing”—accessing the hard disk for virtual memory. Quit other applications, or, if possible, expand your system’s physical memory.
- Try to avoid moving the mouse while the simulation is running. Working Model 3D would need to spend its resources during the simulation to track the moving mouse (for rotating the view, for example).

C H A P T E R 10

Measuring Simulation Data

This chapter will show you how to:

- create meters
- customize meters
- print meter data

10.1 Creating Meters

Meters allow you to extract data from your simulation and display them in numerical or plotting format. You can not only measure almost any physical property in a Working Model 3D simulation, but also customize any meter to measure, display, or evaluate arithmetic and mathematical expressions using the versatile formula language available in Working Model 3D.

Working Model 3D also allows you to export the meter data to a text file. You can use a spreadsheet program or other applications to import the meter data for further analysis. For instructions on exporting the meter data, please see “Exporting Meter Data” on page 11–9.

What Can Be Measured with Meters

When you select a set of objects, Working Model 3D automatically presents a menu of quantities that are available for measurement in the Measure menu.

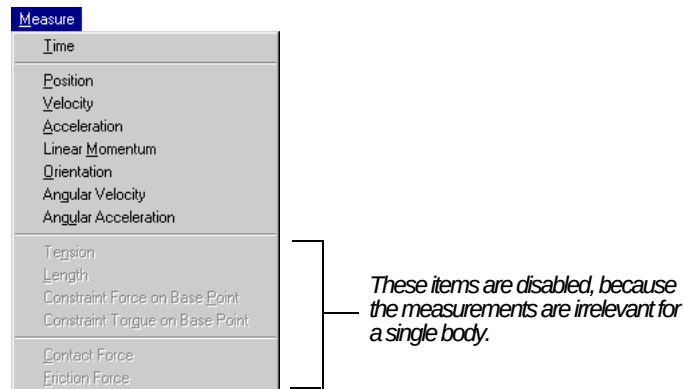
► *To see which properties can be measured for any object:*

1. Select the object.
2. Open the Measure menu.

The menu presents you with available options.

Figure 10-1 shows an example of the Measure menu when a body is selected.

Figure 10-1
Measure Menu when One Body is Selected



Depending on which object you select, Working Model 3D presents appropriate options. Note that when two objects are selected, you can measure a quantity that pertains to the interaction of the two objects, such as a contact force.

The table below shows the available meter options for selected objects.

When you select...	you can measure...
(nothing)	time
a body	position, velocity, acceleration, momentum, orientation, angular velocity, angular acceleration
two bodies	contact force (see note below), friction force
Coord-on-Coord constraints	constraint force on base point, constraint torque on base point
point-to-point constraint	tension, length

NOTE: Contact force meters do NOT measure impact forces or impulses and should NOT be used for determining impact loads. Contact force meters report non-zero values only when two bodies are in steady contact. If two bodies are colliding and/or have periods of separation, you will notice the contact force meter measuring zero.

10.2 Customizing Meters

In addition to the predefined meters discussed in the previous section, you can customize not only the appearance, display format, but also the measured quantities.

Switching Between Digital and Graph

You can switch a meter between digital and graph format by adjusting its appearance properties.

► **To switch the meter format:**

1. **Double-click the meter.**

The Properties window for the meter appears.

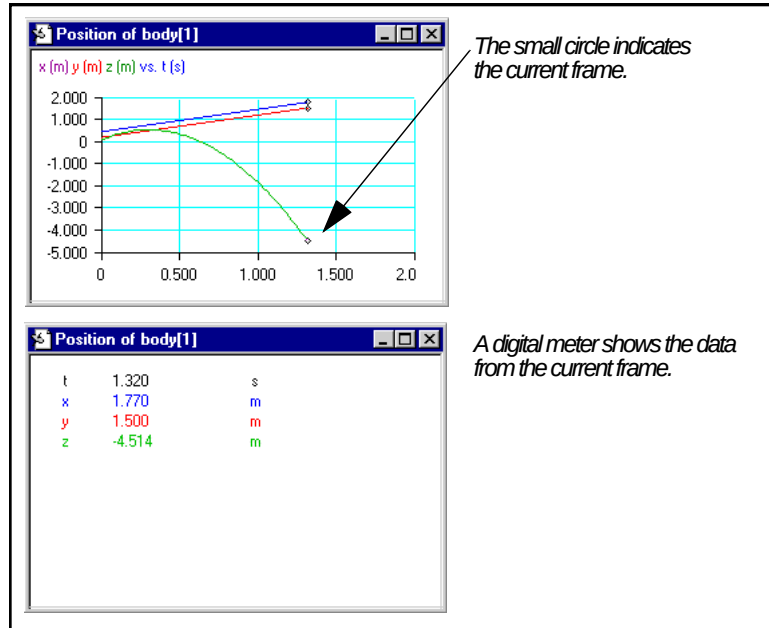
2. **Click the Appearance tab.**

3. **Select the Digital or Graph radio button.**

The meter changes its format.

NOTE: The digital meter shows data from the current frame only. You can advance or recede the frame counter to see data from other frames.

Figure 10-2
Digital and Graph Meter Format

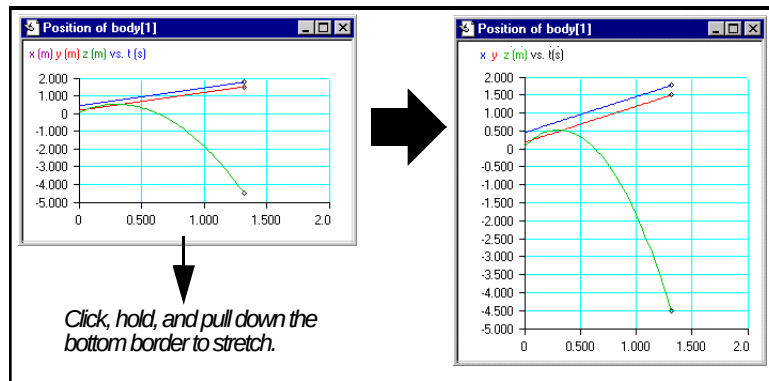


Modifying Meter Position and Size

Meters can be located anywhere within the Working Model 3D window. To move the meter, drag the title bar of the meter.

To modify the meter size, drag one of the corner handles. The meter plot adjusts and scales itself automatically. Figure 10-3 shows an example where the meter is stretched vertically.

Figure 10-3
Resizing a Meter

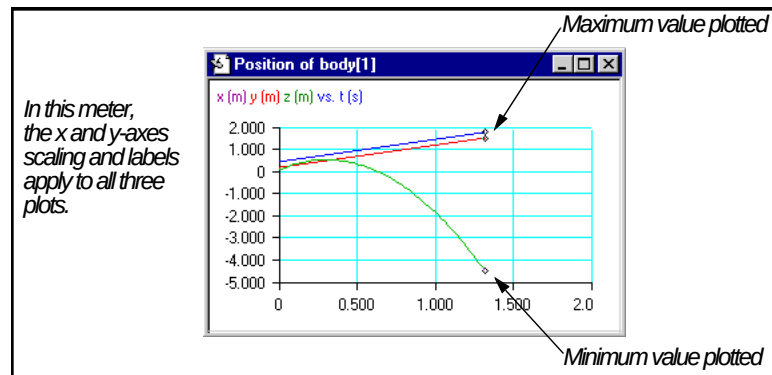


Adjusting Scale of Meter Axes

The Working Model 3D meters allow you to customize the meter's axis scales.

By default, the meter axes are scaled dynamically so that the plotted graph fits in the meter window. For meters with multiple plots (as with most default meters), Working Model 3D ensures the correct comparative analysis because each meter plots all the values within the same minimum and maximum boundaries and scales the plots with the same ratio (Figure 10-4).

Figure 10-4
Meter Properties Window



In the default mode, Working Model 3D controls the axis scale by taking the maximum and minimum values from all the plots, and applies them to set the y-axis so that all the plots fit in the window with a slight margin (Figure 10-4). The same principle applies to the x-axis (although the x-axis—the abscissa—is limited to display one variable).

You can customize the axis scaling by providing your own minimum and maximum values for any of the plots.

► To adjust the meter scaling:

1. **Create a meter (e.g., position of a body) and run the simulation to plot some data (so that you can observe the changes made by the following steps).**
2. **Double-click anywhere within the meter window (do not double-click the title bar, or the meter window will be maximized).**

The Properties window is displayed, with the Axes panel at the front.(Figure 10-5).

Figure 10-5*Properties Window for Meter*

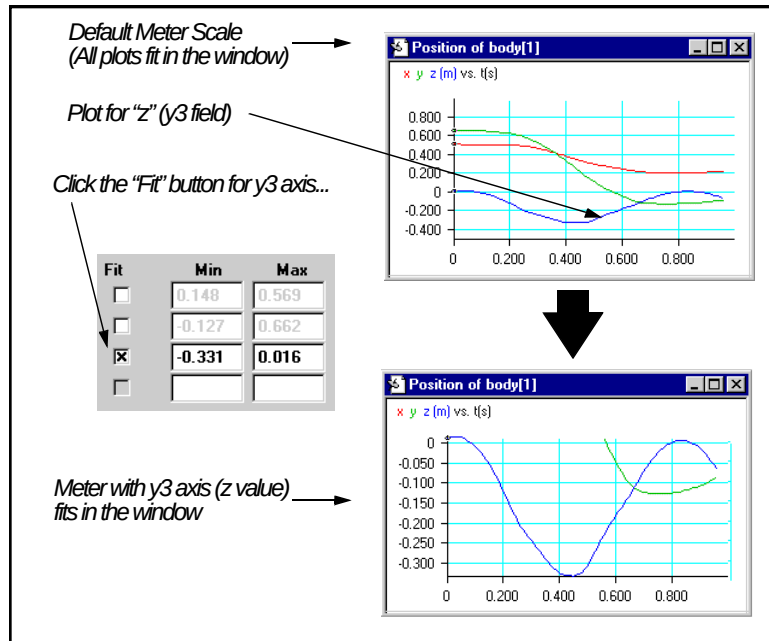
	Label	Fit	Min	Max
y1	x	☐	0.607	1.37
y2	y	☐	0.113	0.255
y3	z	☐	-2.89	-0.135
y4		☐		
x	t	☐	0	0.51

3. Check one of the boxes labeled “Fit”.

The minimum and maximum values become black, and the meter is automatically scaled so that the corresponding plot fits nicely in the meter window (Figure 10-6). The y-axis uses the minimum and maximum values provided in the Meter Properties window.

Figure 10-6 shows an example with a graph meter plotting x, y, and z position of a body. The three plots correspond to the fields y1, y2, and y3 (as shown in the Meter Properties window), respectively. When you click the Fit button for the y3 field (which plots the z position), the graph meter scales automatically according to the minimum and maximum value attained thus far.

Figure 10-6
Forcing Scale Fit



Function of the Fit Option

When none of the Fit options are checked:

- Working Model 3D automatically adjusts upper and lower boundaries of each plot to accommodate its minimum and maximum values and scales the meter accordingly.

When one or more Fit options are checked:

- In the Meter Properties window, the corresponding minimum and maximum values become black, indicating that you can edit them. Working Model 3D will no longer update the minimum and maximum values.
- Working Model 3D uses the values entered in the minimum and maximum values to scale the graph plot. If multiple "Fit" boxes are checked, Working Model 3D determines the maximum and minimum boundaries from all the values entered.
- You can disable the Fit option again to allow Working Model 3D to scale the graph automatically.

Modifying Meters to Display Customized Properties

You can fully customize meters through the use of the Working Model 3D formula language. When you create a meter with the Measure menu, Working Model 3D simply creates a blank meter, and fills its fields with predefined formula language expressions. Once the meter is created, you have complete control over the expressions. You can modify meters so that you can:

- change x-axis from time (default) to frame numbers, or another property of the simulation
- refer to the meter as a variable to be used in another formula expression

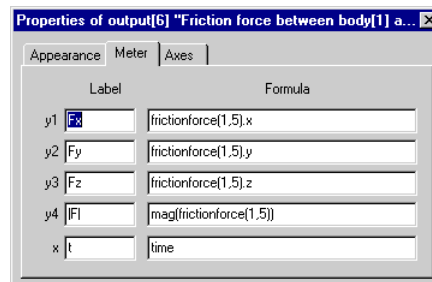
For example, Figure 10-5 shows the meter with body positions by using expressions such as:

`body[1].p.x, body[1].p.y`

whereas Figure 10-7 shows the Properties window for a meter that measures the friction force between two bodies (the components are returned in the world coordinates) with the expressions such as:

`frictionforce(1,5).X`

Figure 10-7
Properties Window for Meter
Measuring Friction Force



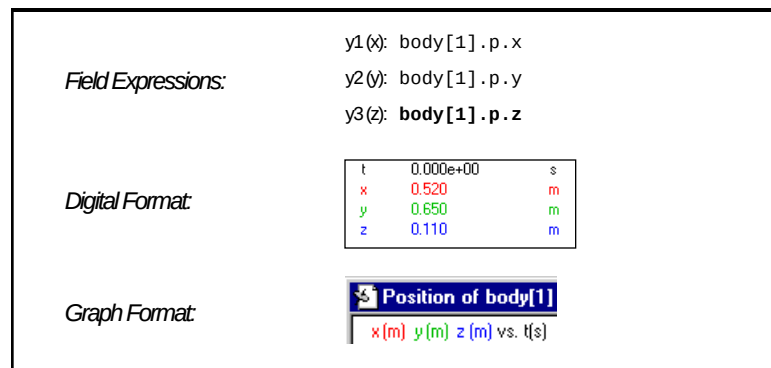
The meter window automatically detects the units for the formula expression entered, and displays them accordingly. See below for more information.

The section "Customizing Meters" on page 12–6 provides several examples. For the complete information on the formula language, please see Appendix B, "Formula Language Reference".

Meters and Unit Labels

Meters display each field with appropriate unit labels. For example, Figure 10-8 illustrates a meter that traces a body's x, y, and z position. Each expression (such as `body[1].p.x`) is associated with the distance unit, and each meter label is shown with its units.

Figure 10-8
Meters and Units



10.3 Printing Meter Data

You can print the meter data, either in graph format or in digital format.

► To print the meter data:

1. Select the meter window or select the meter in the Object Manager.
2. Change the graph format to digital or graph as desired.

See "Switching Between Digital and Graph" on page 10-3 for instructions.

3. Choose Print from the File menu.

The standard Windows print dialog appears. Set the desired option and proceed to print.

Note: You may want to preview how the meter will print, using the File menu's Print Preview command.

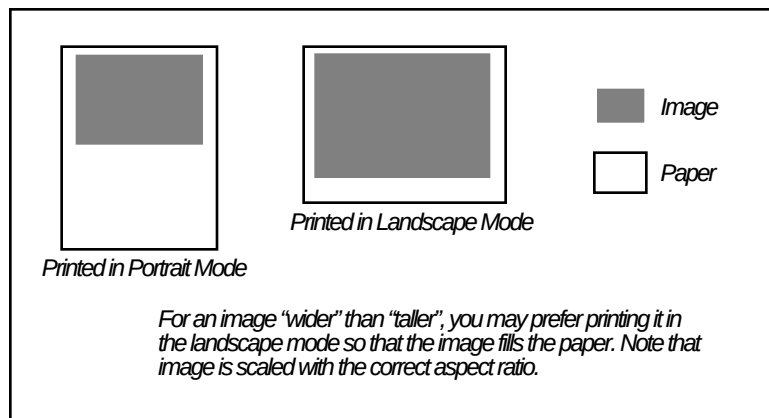
Working Model 3D prints the currently active meter. If the meter is in the graph format, the printed image bears the graph plot. If the meter is in the digital format, the printed image shows data from the current frame.

If the meter is in the graph format, the printed image is maximized on the printed page, while the image's aspect ratio is maintained. The orientation of the printed page (i.e., whether in the portrait mode or landscape mode) follows the Windows page setup settings.

For example, if your graph plot is rectangular, you may prefer printing the image in the landscape mode so that the image spreads over the paper nicely (see Figure 10-9).

See "Printing Simulations" on page 9-8 if you wish to print the simulation model instead of meters.

Figure 10-9
Printed Image



10.4 Deleting Meters

You can delete the meter data.

► ***To delete the meter:***

1. **Select the meter window, or select the meter from the Object Manager list.**
2. **Press the DELETE key.**

NOTE: You can also hide the meter, rather than deleting it, by clicking the close box.

CHAPTER 11

Import and Export

11.1 Available Import/Export Options

Working Model 3D has the following import/export options.

Import

Working Model 3D can import the following files:

- ACIS files
- Working Model 3D files

ACIS Files

ACIS is an industry-wide file format that is used to store complete information of three-dimensional solids. Using an ACIS Converter tool (installed with Working Model 3D), you can convert an ACIS geometry and bring it into a Working Model 3D document.

Importing Working Model 3D Files

The feature allows you to bring in another Working Model 3D file into the existing document. For example, you can store an archive of component files in your library, and import them as necessary to build a composite model.

Export

Working Model 3D can export the animation results in Video for Windows format. You can also export the meter data.

Video for Windows

Video for Windows is a standard animation file format for the Windows operating system. You can specify various parameters such as frame rate, compression method, and compression rates to match your needs.

NOTE: In order to save or play back the Windows for Video files, your system must have the Video for Windows Runtime Library.

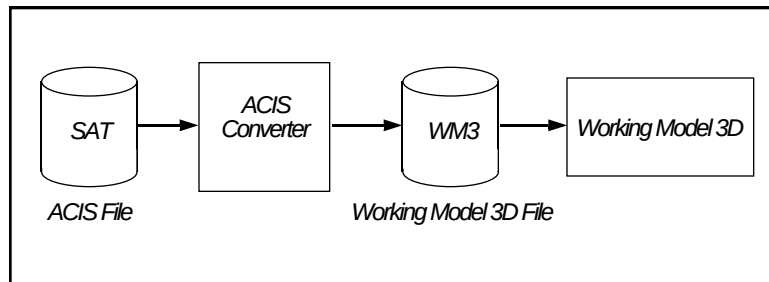
Meter Data

You can export the meter data taken during the simulation to a tabulated text file which can be opened with most data processing programs or spreadsheet applications.

11.2 Importing ACIS Files

The Working Model 3D installation comes with a utility called **ACIS Converter** that converts an ACIS file (which typically has the extension “.SAT”) to a Working Model 3D file (with the extension “.WM3”). Working Model 3D does not directly import an ACIS file, but you can use this utility to convert existing ACIS files to Working Model 3D format. Figure 11-2 below shows the data flow of ACIS import.

Figure 11-1
Data Flow of ACIS File Import



Reading ACIS Files

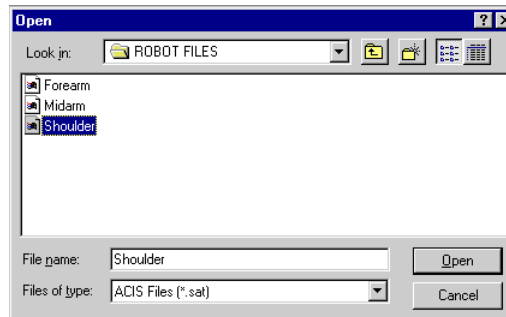
The application icon for ACIS Converter is in the Working Model 3D program group. Otherwise, you can find the file ACIS2WM.EXE in the Working Model 3D directory.

► **To convert an ACIS file to a Working Model 3D file:**

1. **Launch the ACIS Converter (ACIS2WM.EXE).**

The File Open dialog appears (Figure 11-2).

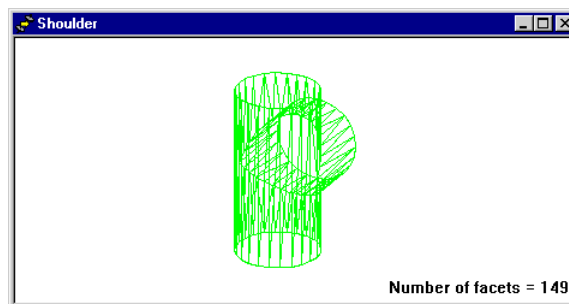
Figure 11-2
ACIS Converter Import Dialog



2. **Select the ACIS file you wish to import.**
3. **Click Open.**

After reading the file, ACIS Converter displays a wireframe rendering of the geometry (see Figure 11-3 for an example). The window also shows the number of facets used in the wireframe model.

Figure 11-3
Sample Result of Imported Geometry in ACIS Converter



To ensure that the entire object is visible, ACIS Converter automatically scales the geometry to fit in the window, regardless of its dimension or the unit system. The geometry is shown with a predefined viewing direction in isometric projection.

The number of facets represents the geometric complexity of the model. In general, fewer facets yield a faster simulation, whereas more facets increase the accuracy of the model. The ACIS Converter allows you to control the complexity of the model before it creates the Working Model 3D file (see below).

4. **Choose Export Options from the Options menu to modify the facet resolution, if so desired (see “Tuning Facet Resolution” on page 11–4 for more information).**
5. **Choose Export in the File menu.**

The File Save dialog appears (similar to Figure 11-2).

6. **Type the output filename, and click Save.**

The output file is in Working Model 3D format, and by default has a .WM3 extension.

7. **Quit the ACIS Converter, and launch Working Model 3D.**
8. **Set the unit system to match the ACIS file (see below).**
9. **Import the converted Working Model 3D file (see “Importing Working Model 3D Files” on page 11–6).**

Notes on the Unit System

When ACIS Converter generates the WM3 file, the unit information is discarded. Therefore, the Working Model 3D file generated by the ACIS Converter is unitless, unlike the files generated by Working Model 3D.

For example, suppose you had an ACIS file with a part that is 22.3 meters long. If you converted the file to the WM3 format using ACIS Converter, and imported into a Working Model 3D document whose current distance unit is set to inches, the part will be 22.3 inches long.

Therefore, if you wish to integrate multiple ACIS models that are created under different units into a single Working Model 3D document, make sure to change the unit system of the document (the document into which other files are imported) every time before importing each file. See “Numbers and Units” on page 5–1 for more information on changing the unit system.

Tuning Facet Resolution

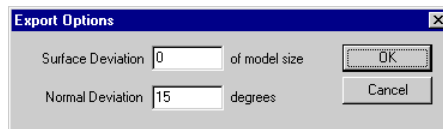
While Working Model 3D represents a body as a polyhedron (a solid bounded only with polygons), geometry represented in ACIS can contain polyhedra *and* solids bounded with curves and curved surfaces. Therefore, ACIS Converter must approximate nonlinear features with planar surfaces before exporting the data to a Working Model 3D file.

The approximation, often called *faceting*, involves converting curves and curved surfaces with a set of polygon. Through the Export Option dialog (Figure 11-4), ACIS Converter allows you to control the resolution on the facets—that is, how fine or coarse the surface would appear after the file is imported to Working Model 3D.

You can use these parameters to fine-tune geometric complexity of the objects to be imported in Working Model 3D. As you change these values, the “number of facets” shown in the ACIS Converter’s document window reflects the complexity of the facetization (the higher the number of facets, the greater the complexity of the object becomes).

As much as possible, you should fine-tune the complexity of the object to improve Working Model 3D’s performance (which favors less geometric complexity) without sacrificing realistic simulation results (which favors greater geometric complexity).

Figure 11-4
ACIS Export Option Dialog



ACIS Converter provides the following export options for ACIS conversion:

- Surface Deviation
- Normal Deviation

Surface Deviation

The *surface deviation* specifies the maximum distance between a facet and the true surface as a ratio compared to the size of the model. The default value is 0, meaning that ACIS Converter automatically computes the proper value.

Normal Deviation

The *normal deviation* specifies the maximum deviation (in degrees) allowed between the normal vectors of any two adjacent facets. In general, a smaller normal deviation results in more facets. The default value is 15 degrees. The value zero, or any value greater than 15 does not have any effect.

11.3 Importing Working Model 3D Files

You can import another Working Model 3D file into your document. When you import a file, the objects in the file are added to the current document, with all the existing objects intact.

The import feature proves useful in the following situations, for example:

- You can keep a library of parts and components as Working Model 3D files. When you put together a new design or mechanism, simply import these parts and assemble them.
- Subdivide an entire design into small components. Build and test (in Working Model 3D) each component separately to make sure that the specifications are correct, then bring the components together by importing the individual files.

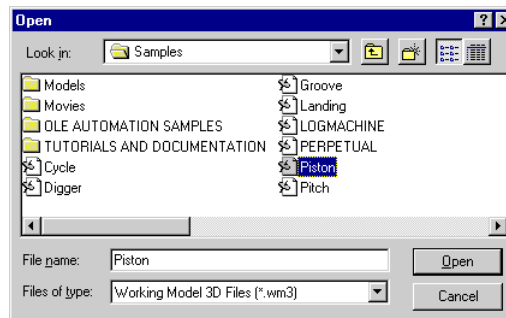
In any case, we recommend you start by building and testing small components first, and then putting them together.

► **To import another Working Model 3D file:**

1. **Choose Import in the File menu.**

The Import dialog appears, as shown in Figure 11-5. By default, the dialog only shows files with a .WM3 extension.

Figure 11-5
Import Dialog



2. **Specify the file you wish to import.**
3. **Click Open.**

The current document contains all the objects that existed in the imported file.

You can repeat this process to build a large model progressively.

Working Model 3D automatically changes the unit system of the imported file (the “importee” file) to align with the existing file (the “importer” file). For example, suppose that the importee had a part that is 52.4 meters long, and the importer file has the distance unit set to inches. When the file is imported, the part will be 2063 inches (= 52.4 meters) long in the importer file.

NOTE: Working Model 3D files generated by the ACIS Converter are *unitless*. Therefore, when the file is imported, all dimensions are interpreted in the current unit system. See "Importing ACIS Files" on page 11-2 for more information.

11.4 Exporting Video for Windows

Video for Windows is a standard animation data format used on Windows-based PCs. Working Model 3D simulations will play back more quickly as Video for Windows movies.

NOTES:

- In order to export, you must have the Video for Windows Runtime Library installed on your machine. **The Video for Windows system is included in Windows 95 and Windows NT.**
- To play back the exported Video for Windows files (typically called AVI files), you also need the Video for Windows Runtime Library. Please contact Microsoft to obtain a copy.

► **To export a Video for Windows movie:**

1. **Run the simulation.**

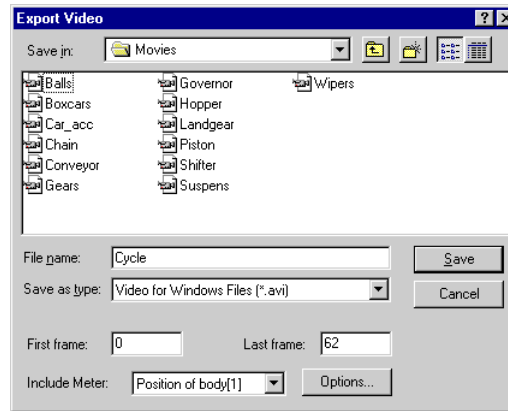
You MUST have a history.

2. **Choose Export Video from the File menu.**

The Export Video dialog appears (Figure 11-6).

3. **Choose which meter you want to export (if any) from the Include Meter popup menu.**

Figure 11-6
Export Video Dialog



4. Click the Options button to set Video Compression parameters as necessary (see “Video Compression Options” on page 11-8).
5. Click Save.

NOTE: Working Model 3D exports the image of your simulation window. Toolbar, Coordinates bar, menu items, document name, and the Tape Player Controls do *not* appear in the exported video file.

Video Compression Options

Working Model 3D provides default values for the image compression options that are sufficient for most purposes. If you wish to customize the image quality and storage requirements, Working Model 3D provides these advanced options.

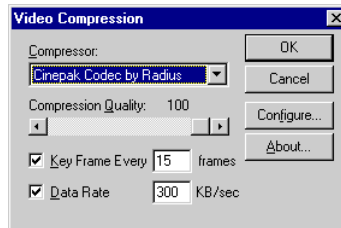
► **To set the compression option parameters:**

1. Choose Export Video from the File menu.
2. Click the Options button in the Export Video window.

The Video Compression dialog appears (Figure 11-7).

For options given in the Video Compression dialog, please refer to the documentation on Video for Windows provided by Microsoft Corporation.

Figure 11-7
Video Compression Dialog



11.5 Exporting Meter Data

Working Model 3D allows you to export the data from meters in a tab-delimited text file. Since the file is in the ASCII text format, you can open the exported file with any word processor or spreadsheet program. This feature allows you to use another application for further analysis of the simulation data obtained by Working Model 3D.

The feature exports data from all the existing meters or from the meter you chose. Data from each meter axis is printed in columns, whereas each data row shows the output from every frame. The numerical format of meter data is taken from the current settings in the Numbers & Units dialog (see “Numbers and Units” on page 5–1). For example, a document with a single position meter would produce the following file:

Data from C:\MYFILES\TEST.WM3

at Mon Mar 11 15:32:50 1996

Position of body body[1]

t	x	y	z
0.000	0.540	0.460	0.270
0.010	0.550	0.470	0.280
0.020	0.560	0.480	0.288

0.030	0.570	0.490	0.296
0.040	0.580	0.500	0.302
0.050	0.590	0.510	0.308
0.060	0.600	0.520	0.312
.			
.			
.			

When the document contains multiple meters, columns are appended to the right to accommodate the data. Therefore, each data row always contains the output from every meter axis at every frame.

Working Model 3D exports meter data in the current unit system.

NOTE: If the document has no meter, the Export Meters menu item is disabled. See Chapter 10, “Measuring Simulation Data”, for information on how to create meters.

► **To export meter data:**

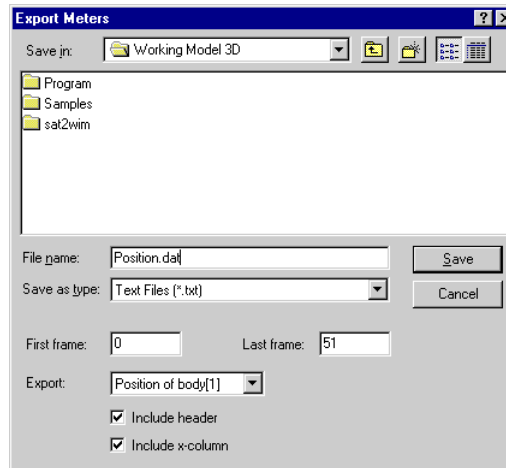
1. **Run your simulation.**

You MUST have a history.

2. **Choose Export Meters... in the File menu.**

The Export Meter Data dialog appears (Figure 11-8).

Figure 11-8
Export Meter Data Dialog



3. Type the name for the output data.
4. Set export options as necessary (see below for descriptions).
5. Click Save.

Working Model 3D saves the data file.

Export Options

The Export Meter Data dialog provides the following options:

First Frame, Last Frame

Specify the first and last frame numbers for data export. By default, Working Model 3D assigns the span of the time history as the First Frame and Last Frame.

Export Meter

Choose all meters or a particular meter to export.

Include X-Column

When checked, this option includes the x-column data of each meter as a column. All the default meters have time as the x-axis. You can specify another variable (such as the frame number) and make it to be the independent variable. See “Modifying Meters to Display Customized Properties” on page 10–8 for more information.

This option is on by default.

Include Header

When checked, this option includes the file name, date, meter name, and column names (labels) before all the numerical data.

This option is on by default.

Printing the Tabulated Meter Data

Since the exported meter data file is a tab-delimited text file, you can open the file with any word processor or spreadsheet application, format the data as you like, and print it from the application.

You can also print the graph plot or a digital meter readout directly from Working Model 3D by selecting the meter and choosing the Print from the File menu. See “Printing Meter Data” on page 10–9 for more detail.

C H A P T E R 12

Using Formulas

Working Model 3D has an embedded formula language featuring a full set of mathematical and arithmetic functions. The language follows a simple syntax, resembles a high level programming language, and enormously expands the capabilities of Working Model 3D by allowing you to customize various aspects of your simulation model.

This chapter provides examples and information on using formulas. For the complete reference of the formula language, please refer to Appendix B, “Formula Language Reference”.

12.1 Units and Formulas

A *constant* is defined as a literal number. Everything else is considered a formula, including expressions that evaluate to constants. For example, 3.12 is a constant, whereas $3+2$ is a formula expression.

- All constants are associated with the current unit system. Changing the current settings in the Numbers & Units dialog box affects the values displayed.
- Formulas are always expressed SI units (meters/kilograms/seconds). Changing the current settings in the Numbers & Units dialog has no effect on formulas you have defined.

Constants

Length: 5 [in feet]

Length: 60 [in inches]

Length: $\text{time} + 5$ [in feet]

Length: $((\text{time}/60)+5)*0.3048$ [in meters]

- For example, suppose you created a time meter while the current unit for time is seconds. If you change the unit system to minutes, the meter “knows” that the unit change has taken place, and multiplies values by an internal conversion factor of 1/60 to display the data properly in minutes.

Formula references to a Meter object are always associated with SI units, even though the Meter displays its data in the current unit system.

12.2 Customizing Constraints

Perhaps the most powerful application of the formula language lies in its capability to customize objects in Working Model 3D simulation. This section provides but a few examples.

Creating a Torque-Limiting Velocity Motor

Working Model 3D allows you to create a velocity motor, which, given an angular velocity specification, applies whatever torque is necessary to maintain the angular velocity. While this feature simplifies your mechanism design effort tremendously, chances are that your physical model features a motor that can only apply a finite amount of torque. To model such a realistic motor, the formula language provides a very simple solution.

The example below creates a motor that can sustain the torque of 1.0 Nm. Suppose you would like to maintain the angular velocity to be 500 radian per second. The motor has a simple proportional difference control system.

► **To create a torque-limiting motor:**

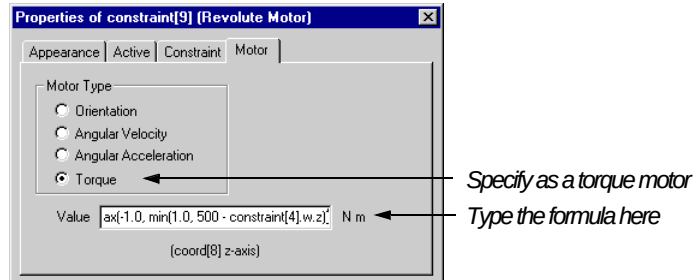
1. **Create a motor in your model.**

For the construction of constraints, see "Creating Constraints" on page 7–7.

2. **Double-click the motor in the model window or the Object Manager.**

The Properties window appears (Figure 12-1).

Figure 12-1
Properties Window for Motor



3. Select Torque as the motor type.

Although you are about to create a velocity-controlled motor, the torque motor allows you to specify the amount of torque based on the current angular velocity.

4. In the Value field of the Properties window, type the following formula:

```
max(-1.0, min(1.0, 500 - constraint[4].w.z))
```

The formula expression instructs the motor to exert at most 1.0 Nm in either direction in an attempt to maintain the angular velocity of the motor to be 500 (radians per second). Note that Working Model 3D interprets these numbers in the SI unit system.

Instead of a rudimentary control system like the one above, you could implement an advanced control system by entering a more complex formula.

Constraint that Breaks at Certain Load

By default, all constraints in Working Model 3D are absolute. For example, a revolute joint never breaks no matter how much force acts on the bodies connected by the joint.

To simulate a constraint that breaks, you can take advantage of the Active When field Active page of the Properties window. For example, suppose you wish to create a revolute joint that breaks after the force magnitude exceeds 2 newtons.

You will approach the solution in two steps as follows:

- First, create a meter that measures the maximum force applied to the constraint.
- Second, specify the revolute joint so that it is only active when the maximum force is under the tolerance (2 newtons).

Suppose you already have a revolute joint in your model.

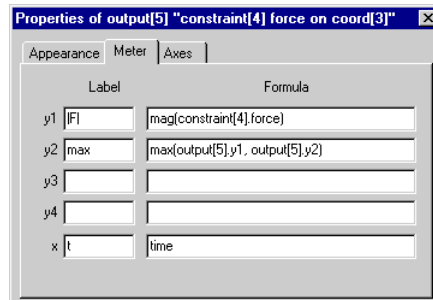
1. **Select the revolute joint, and choose Constraint Force on Base Point in the Measure menu.**

The force meter appears.

2. **Double-click the meter window to open the Properties window for the meter.**
3. **Click the Meter tab.**
4. **Modify the meter fields to record the maximum force magnitude.**

See Figure 12-2 as an example. The figure assumes that the constraint ID of the revolute joint is `constraint[4]`, and the output ID is `output[5]`.

Figure 12-2
Measuring the Maximum Force



The formula expression `max ( a, b )` takes the two expressions and returns the maximum value. Since the meter fields are evaluated from top to bottom in sequence, the y2 field will always record the maximum value returned by the y1 field, or the magnitude of the reaction force at the joint.

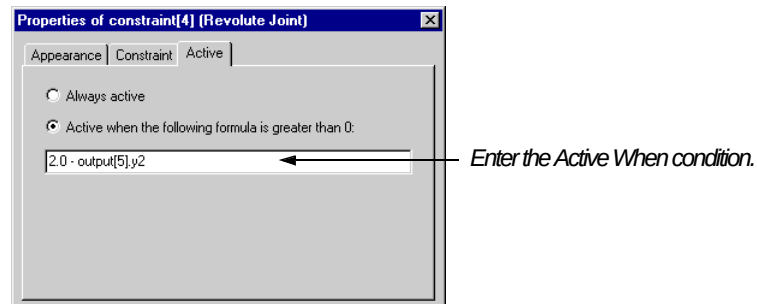
5. **Double-click the revolute joint.**

The Properties window appears.

6. **Select the Active panel.**

See Figure 12-3.

Figure 12-3
Specifying Active When Condition



7. Enter this formula that describes when the constraint should be active (use the output ID for the meter you created in Step 1):

$2.0 - \text{output}[5].y2$

The constraint is now specified to be only active when the maximum force reading is under 2.0 newtons. The bodies bound by the revolute joint will fall apart when the force exceeds 2 newtons during the simulation.

12.3 Customizing Meters

Working Model 3D features meters to measure various simulation properties. In addition to default meters discussed in Chapter 10, “Measuring Simulation Data”, you can create customized meters, both by modifying the existing default meters, or creating an entirely new meter from scratch. Shown below are some examples.

Changing X-axis

All the default meters have time (simulation time clock) as the x-axis.

► *To modify the x-axis of the meter:*

1. Create any simulation meter. For example, create a block, and choose Position in the Measure menu.

The corresponding meter appears.

2. Double-click the meter.

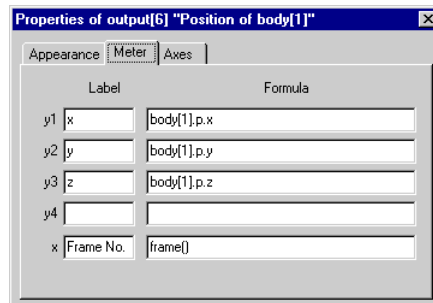
The Properties window appears.

The window presents the default settings for each field of the meter. In particular, note that the x-axis bears the formula expression `time`, which returns the simulation clock reading. You will change the x-axis reading so that it reads the frame number.

3. To change the x-axis measurement from time to frame number, delete the expression `time` the x-axis field, and type `frame()` instead.

See Figure 12-4.

Figure 12-4
Customizing X-axis Field



4. Close the Properties window.

5. Click the Run button on the Tape Player Control

Note that the meter's x-axis reads the frame number instead of the simulation clock.

Finding Patterns in Nonlinear Systems

Finding solutions to a nonlinear system is often a daunting task, if not an impossible one. Working Model 3D can be an effective tool to perform a qualitative analysis.

For example, a common method of nonlinear system analysis is to observe system response as depicted in state plane plots. Such analyses are useful for detecting stability of equilibrium points, limit cycles,

dependence of response on initial conditions, and chaos. In addition to being a valuable analytical tool, state plane plots can be intriguing means to characterize a dynamical system.

Shown below is a simple double-pendulum, consisting of two bodies connected in series with revolute joints. The instructions below show how to create a custom meter that serves as a state plane plot to trace the x-coordinates of the two bodies.

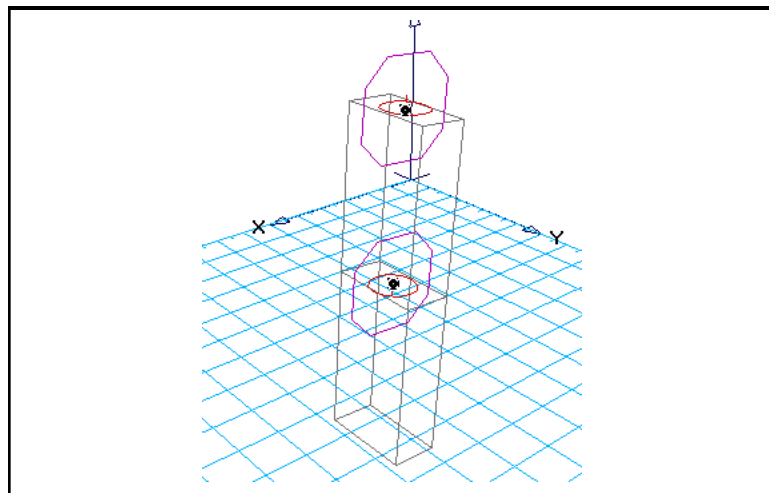
► **To create a custom meter for the double-pendulum system:**

1. **Create a simple double pendulum system using blocks and revolute joints.**

The model may appear as shown in Figure 12-5.

Figure 12-5

A Double Pendulum System



2. **Select either of the two bodies.**
3. **Choose Position in the Measure menu.**

The position meter appears.

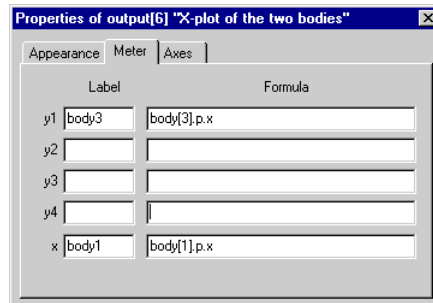
4. **Double-click the meter window.**

The Properties window appears.

5. **In the X-axis field of the Properties window, type the formula expression for the x-position of the other body.**

See Figure 12-6 for an example.

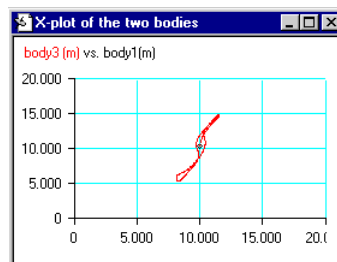
Figure 12-6
Modifying the X-axis



6. Run the simulation.

The plot shows an interesting trace of the x-positions of the two bodies (Figure 12-7).

Figure 12-7
Plot of the x-position of Double
Pendulum Bodies



Using Meters as Variables

As you attempt to simulate more and more complex models, the formula expressions may become quite long and complicated. The string length of the expression may even exceed 255 characters, the limit set by Working Model 3D. In such cases, you can simplify the expressions by typing often-used expressions into a meter, and refer to the meter in another expression.

For example, suppose you frequently wish to use the expression:

$$|\text{body}[3].p - \text{body}[4].p|$$

which defines the distance between two bodies. Instead of entering this expression everywhere, you can define a meter and use it as a variable, or intermediate placeholder.

► **To define a meter and use it as a variable:**

1. Under the Measure menu, choose Time and create a time meter.

In fact, any meter can be used as a variable. For now, you will just use a time meter and modify it.

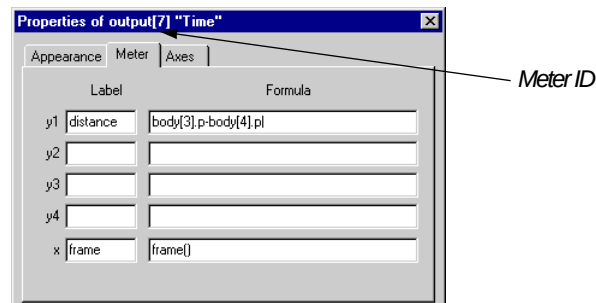
2. Double-click on the time meter.

The Properties window appears (Figure 12-8).

3. Overwrite the field y1 with the formulas shown in Figure 12-8 below.

In fact, you can use any or all of the output fields (y1 through y4) to define the formula.

Figure 12-8
Using Meter Fields as Variables



4. Make a note of the meter ID; you will refer to the expression through the meter ID from here on.

The meter ID appears at the top of the Properties window and in the Object Manager list. In the example shown in Figure 12-8, the meter ID is output[7]. You can rename this meter to a more convenient name on the Appearance page of the Properties window

You can now refer to the distance quantity by simply typing:

`output[7].y1`

(or y2 through y4, depending on which field you used as a variable)
instead of spelling out:

```
|body[3].p - body[4].p|
```

For example, in order to take the square-root distance between the two
bodies, you can type:

```
sqrt(output[2].y1)
```

instead of:

```
sqrt(|body[3].p - body[4].p|)
```


A P P E N D I X A

Technical Information

This chapter describes technical information that may help you better understand the algorithms employed by Working Model 3D. The last section in Appendix A also provides literature references for users who are interested in the subject matters of mechanics and numerical analysis.

A.1 Numerical Methods

Overview

Working Model 3D solves problems using a variety of sophisticated numerical methods. A problem is time-discretized so that Working Model 3D computes motion and forces, while making sure that all the constraints are satisfied. With its systematic approach, Working Model 3D can model a wide variety of problems.

Numerical methods is an area of ongoing research in science and engineering. Vast amounts of literature are available on this topic. Interested readers are strongly encouraged to refer to more advanced literature. The “Technical References” on page A–19 may serve as a beginning for interested readers.

In describing numerical methods, let's take the example of a ball traveling in projectile motion. For simplicity, consider gravity as the only force acting on the ball.

Analytical Method ¹

An analytical solution for the x, y and z position of the mass center a ball seen in most elementary physics texts is:

$$x = x_0 + v_{x_0} t$$

¹. Also called *symbolic method* or *closed form solution*

$$y = y_0 + v_{y_0} t$$

$$z = z_0 + v_{z_0} t + \frac{1}{2} g t^2$$

where $g = -9.81 \text{ m/sec}^2$.

With these formulas, one can find the position of the ball at any moment in time by simply plugging in the correct initial values

$$(x_0, y_0, z_0, v_{x_0}, v_{y_0}, v_{z_0}).$$

If projectile motion were the only type of problem one needed to solve, there would be no need for the many numerical methods employed by Working Model 3D.

However, for most physical problems, it is impossible to find exact solutions like those for projectile motion. For example, no analytical solution exists for the equations of motion for three particles (or stars) all acting under gravitational forces among themselves.¹

Numerical Integration

Instead of analytical methods, Working Model 3D uses numerical methods to allow the solution of the motion of mechanical systems which are governed by differential equations arising from mechanics principles. In 3-D, these principles can be expressed with the familiar equations:

$$\mathbf{F} = m\mathbf{a}$$

$$\mathbf{T} = \mathbf{I} \boldsymbol{\alpha} + \boldsymbol{\omega} \times \mathbf{I} \boldsymbol{\omega}$$

$$\mathbf{a} = \frac{d\mathbf{v}}{dt}$$

$$\mathbf{v} = \frac{d\mathbf{x}}{dt}$$

$$\boldsymbol{\alpha} = \frac{d\boldsymbol{\omega}}{dt}$$

¹. As a matter of fact, no analytical solutions exist for 2 bodies, either, unless they are perfect spheres.

where $\mathbf{F}$ is force, m is mass (scalar quantity), $\mathbf{a}$ is acceleration, $\mathbf{T}$ is torque, $\mathbf{I}$ is mass moment of inertia dyadic, α is angular acceleration, ω is angular velocity, and $\mathbf{v}$ is linear velocity.

Working Model 3D uses these equations in its solution of dynamics problems. The solution is carried out by a process known as *numerical integration*. Let us illustrate the process with a one-dimensional case.

For example, integrating both sides of the equation $\mathbf{a} = \frac{d\mathbf{v}}{dt}$ results in

$\mathbf{v} = \int \mathbf{a} dt + \mathbf{v}_0$, which may be approximated as $\mathbf{v} = \mathbf{a} \Delta t + \mathbf{v}_0$. The accuracy of this approximation improves with smaller Δt , called *time step* (see “Time Step and Performance” on page A-4 for more discussion). The heart of numerical integration lies in approximating the problem by subdividing the problem into small, discrete time steps, and incrementally computing the result at each time step.

More specifically, Working Model 3D finds the current acceleration of an object, and uses this acceleration to compute a velocity (and position) one time step later. Furthermore, Working Model 3D incorporates a scheme to check and correct its prediction. This process is then used again to find a new velocity and position.

For example, numerical integration with Euler’s method for a linear motion proceeds as follows:

1. At $t = 0$, calculate $\mathbf{a}$ using equations of motion.
2. Use $\mathbf{a}$ to calculate the new velocity $\mathbf{v}$ at $t = \Delta t$.

$$v(\Delta t) = a(t)\Delta t + v(0) \quad (\text{approximating } v = \int a dt + v_0)$$

3. Use $\mathbf{v}$ to calculate the new position $\mathbf{x}$ at $t = \Delta t$.

$$x(\Delta t) = x(0) + v(0)\Delta t$$

While many numerical integration methods exist, the simplest is Euler integration. Working Model 3D features another, more accurate numerical integrator called the *Kutta-Merson* integration. See “Integrators” on page A-9 for more discussion on numerical integration.

A.2 Time Step and Performance

The size of a time step is a critical parameter in *fixed* step numerical integrators, because it affects the speed and accuracy of the results significantly. In general, a small time step produces more accurate simulation results, but requires more computational effort per given time period than a larger integration time step. If you are using variable step integration, which is the default option, picking a time step is much less critical.

Choosing a Proper Time Step

Choosing a time step for a particular problem may be difficult. The following ad-hoc rules can assist you in selecting a time step:

Small time step = improved accuracy

Large time step = improved computing speed

Usually, you do not have to choose an extremely small time step because you are concerned about accuracy. More often than not, a reasonable step size produces a sufficiently accurate simulation result.

Fortunately, Working Model 3D does a good job at shielding you from needing to pick precise time steps. Working Model 3D has a variable step numerical integrator which automatically adjusts the time step and monitor simulation errors of various types (see “Variable Time Step” on page A-4 for more details).

For example, an ideal time step should be variable during the course of a simulation, adapting to the complexity of the problem in order to get the most out of the speed-accuracy trade-off discussed previously. If you are simulating an automobile collision, the numerical integrator will quickly approach the critical part of the experiment with a large time step while the car is nearing an obstacle, and subsequently use a smaller time step during the collision. If the small time step was used the whole time, the computer might unnecessarily spend a long time to compute an acceptable solution to the problem.

Variable Time Step

A variable time step integrator is useful in getting accurate results relatively quickly. When acceleration changes rapidly during a portion

of a simulation, the integrator internally reduces the integration time step.

When possible, the integrator will increase the integration time step to improve computational performance.

In variable time step mode, Working Model 3D estimates a numerical error at every integration step (“How Working Model 3D Bounds Errors” on page A-5 discusses how the error checking is done), and if the error exceeds a certain tolerance, the current *integration* time step (which is set in the accuracy dialogue box) is considered to be too large for the particular frame. In such cases, Working Model 3D cuts the time step in half and attempts to compute a new result with the smaller time step. This result is subject to the same error checking.

Working Model 3D repeats the error-checking process until the error is less than the user-specified tolerance. This process effectively breaks one animation frame into multiple intermediate frames (that will not be displayed on the screen) in order to ensure accuracy. As a result, you may notice that some frames require more time to display than others.

If accelerations drastically change their magnitude during a simulation, the integrator may be unable to find a time step small enough to meet the accuracy criteria. In such cases, a warning will be given to indicate that the integrator cannot converge on a solution (see “Warnings” on page A-13).

A.3 How Working Model 3D Bounds Errors

As discussed in “Time Step and Performance” on page A-4, Working Model 3D adjusts the integration time step by estimating the integrity of the calculation results. This section explains how Working Model 3D checks its own results against user-defined *error bounds*.

In principle, Working Model 3D makes an error estimate on the calculation results. The method of estimating the error is discussed in “Integrators” on page A-9. Working Model 3D compares the magnitude of the estimate to the user-defined *error bounds*.

Fundamentally, the integration error bound is defined as the greater of the two criteria discussed below: Relative Acceptable Error and Absolute Acceptable Error. These error criteria are based on two parameters (both can be customized): *relative error* (denoted ϵ_r) and *absolute error* (denoted ϵ_a).

Relative Acceptable Error

Relative error is a positive number used to relate the relative acceptable error to $|Y|$, the estimate of the absolute value of Y , the variable being integrated. The Relative Acceptable Error is defined as:

$$\text{Relative Acceptable Error} = \epsilon_r |Y|$$

The value of ϵ_r is chosen to give a specific number of significant digits of accuracy. A useful rule is to set $\epsilon_r = 10^{-n}$, where n is the desired number of significant digits for Y . The value of n can be set in the Accuracy dialog as “Significant Digits” (see “Accuracy Dialog and Simulation Parameters” on page A–9).

The *upper limit* on n depends on the precision of the numbers used for calculations. In Working Model 3D, n should not be greater than 15. The default value of 5 yields highly accurate results in nearly all cases.

The relative error criterion works best when the value of Y retains the same order of magnitude during the integration. If the value of Y varies over many orders of magnitude—especially if Y changes sign or approaches 0—the relative error criterion above would cause the simulation to slow down dramatically.

For example, suppose Y takes on values between -1000 and +1000, and ϵ_r is set to 10^{-8} (8 significant digits). When $Y=1000$, the relative acceptable error is 10^{-5} ($= \epsilon_r |Y|$). However, when $Y=.001$, the relative acceptable error would be 10^{-11} . As a result, when the value of Y is small, Working Model 3D would waste computing time trying to meet this exceedingly small error criterion. To avoid this problem, Working Model 3D uses a second error bound called Absolute Acceptable Error.

Absolute Acceptable Error

Absolute error, ϵ_a , is a positive number which designates a “small” absolute value of Y . That is, when $|Y|$ is less than ϵ_a , integration continues without cutting the integration step. Therefore, in effect, this criterion accelerates the integration process for small values of $|Y|$. The Absolute Acceptable Error is simply defined as:

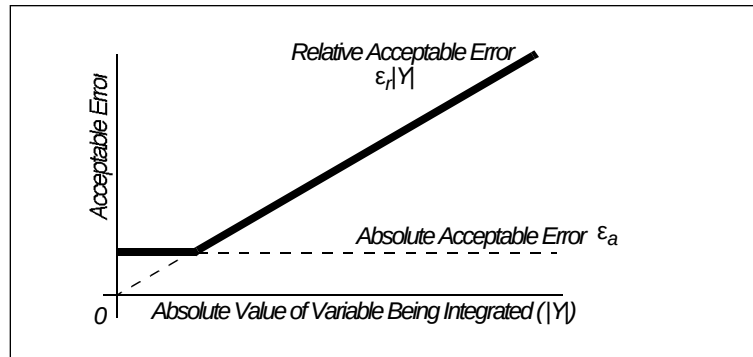
$$\text{Absolute Acceptable Error} = \epsilon_a$$

You can specify the magnitude of ϵ_a as the Configuration Error in the Accuracy dialog. Working Model 3D provides you with controls on the error bounds both in terms of position and orientation. See “Accuracy Dialog and Simulation Parameters” on page A-9 for more information.

Acceptable Error

The acceptable error is the maximum of Absolute Acceptable Error and Relative Acceptable Error. In Figure A-1, the acceptable error is denoted as a thick line. When the error estimate exceeds this line, Working Model 3D proceeds to cut the time step in half (see “Variable Time Step” on page A-4).

Figure A-1
Acceptable Error



Note that the Absolute Acceptable Error remains constant regardless of the value of $|Y|$, whereas the magnitude of the Relative Acceptable Error is proportional to $|Y|$.

Choosing an Acceptable Error for Your Simulation

Now that you have two types of error bounds to control, how can you optimize the simulation performance in terms of its speed and accuracy? The answer really depends on what you are interested in getting out of Working Model 3D.

Note, however, that Working Model 3D's default option automatically chooses the error parameters which suffice for most purposes. This section is for those who wish to fine-tune their simulation runs.

Case Study: a Space Probe

As a case study, consider simulating a space probe that is launched from the Earth.

If you are interested in the position of the space probe after one year from launch, and if you can afford an error of ± 1 mile, you should set the Absolute Acceptable Error to 1 mile. On the other hand, you should keep the Relative Acceptable Error low (e.g. 10^{-6} , which translates to setting Significant Digits to 6), because ± 1 mile of an error is a small fraction of the overall displacement of the probe, which probably extends to millions of miles.

Calculating Probe's Orientation

However, suppose you are interested in keeping the probe oriented in a certain direction in space, and assume that the magnitude of the error must stay within 1° .

Working Model 3D provides an independent tolerance for orientation error in the Accuracy dialog. In the box labeled Configuration Error, simply type a desired value to control the orientation error. By default, Working Model 3D provides a 1° orientational error tolerance. The actual error observed may vary between 0° and 1° , although in most simple simulations, the orientation error observed is very close to 0° . See "Configuration Error" on page A-11 for more information.

Summary

Shown below are general rules of thumb in fine-tuning the acceptable errors:

- You should almost always keep the Relative Acceptable Error low by choosing a value (of at least 3) for Significant Digits in the Accuracy dialog.
- You may need to control the Absolute Acceptable Error (defined as Configuration Error in the Accuracy dialog) depending on the magnitude of the error in the solution you are interested in.

The following section provides more information on how to control simulation parameters.

A.4 Accuracy Dialog and Simulation Parameters

Working Model 3D's Accuracy dialog provides controls over various simulation parameters.

► **To open the Accuracy dialog:**

1. **Choose Accuracy in the World menu.**

The Simulation Accuracy dialog appears.

For most purposes, you can simply use the default simulation accuracy parameters. For those interested in intimately controlling the simulation performance, see below for more information.

Figure A-2
Simulation Accuracy dialog

Integrators

Integration is the mathematical process that Working Model 3D uses to calculate bodies' positions and velocities. The following integrators are available in Working Model 3D:

- Euler Integration

- Kutta-Merson Integration

In order to illustrate the differences between these two methods, let's examine how each integrator solves the following first order differential equation:

$$\dot{y} = f(y, t)$$

We are interested in finding $y(t + h)$, the value of y at time $t+h$, given $y(t)$, the value of y at t and $\dot{y}(t)$, the derivative of y at t . The quantity h is call the *integration step*.

Euler

Euler integration is one of the fastest and simplest—but least accurate—integrators available. Euler integration may suffice in giving you a rough idea of the motion. The Euler method solves the above differential equation in a single step:

$$y(t + h) = y(t) + hf(y(t), t)$$

Working Model 3D does not provide a variable time step option for the Euler method.

Kutta-Merson

Kutta-Merson integration is an integration scheme which can yield highly accurate results fairly quickly. The method solves the above differential equation in the following fashion:

$$y_0 = y(t)$$

$$y_1 = y_0 + \frac{1}{3}hf(y_0, t)$$

$$y_2 = y_0 + \frac{1}{6}hf(y_0, t) + \frac{1}{6}hf(y_1, t) + \frac{1}{3}h\ddot{y}_0$$

$$y_3 = y_0 + \frac{1}{8}hf(y_0, t) + \frac{3}{8}hf(y_2, t) + \frac{1}{3}h\ddot{y}_0$$

$$y_4 = y_0 + \frac{1}{6}hf(y_0, t) - \frac{3}{2}hf(y_2, t) + \frac{1}{3}h\ddot{y}_0 + 2hf(y_3, t) + \frac{1}{2}h\ddot{y}_0$$

$$y_5 = y_0 + \frac{1}{6}hf(y_0, t) + \frac{2}{3}hf(\frac{2}{3}y_3, t + \frac{1}{2}h) + \frac{1}{6}hf(y_4, t + h)$$

$$y(t + h) = y_5$$

The Kutta-Merson integrator approximates the integration error by:

$$\frac{1}{5}|y_4 - y_5|$$

to the Acceptable Error (Please see “How Working Model 3D Bounds Errors” on page A-5).

Information about the Kutta-Merson algorithm is available in L. Fox “Numerical Solutions of Ordinary and Partial Differential Equations,” Palo Alto: Addison-Wesley, 1962, pp.24-25.

Animation Step

The value of Animation Step determines the time between frames of animation. Data from the simulation is presented on the screen as a new frame at this interval. *This value usually (but not necessarily) represents the integration time step.*

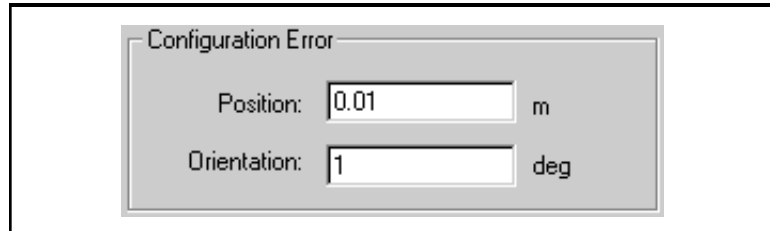
By default, Working Model 3D sets the size of Animation Step to 0.02 second. You can choose an Animation Step more appropriate for your simulation. See “Integration Time Step” below for discussion on how to set the integration time step separately from Animation Step.

Configuration Error

The Configuration Error provides an absolute error bound for variable step numerical integration. The value corresponds to the Absolute Acceptable Error, or the parameter ϵ_a discussed in “How Working Model 3D Bounds Errors” on page A-5. During any integration step, the error as determined by the integrator cannot exceed this absolute error.

Working Model 3D provides independent controls for position and orientation error tolerances. Figure A-3 shows the portion of the Accuracy dialog that controls the tolerances.

Figure A-3
*Configuration Error Settings
(Located in the Accuracy Dialog)*



By default, Working Model 3D sets the position error tolerance to one hundredth of the Grid Extents (see “Properties of the Edit Grid” on page 3–9) and sets the orientation error tolerance to one degree.

Integration Time Step

You can specify the integration time step in two ways; by directly entering the time step, or by entering the number of integration steps per animation frame.

Fixed

In Fixed mode, the integration time step is locked. The default integration step value is equal to the animation step. You can make the integration time step smaller than the animation step by typing the desired step size in the text box. By doing so, you are packing multiple integration steps into one animation frame.

Variable

In Variable mode, the integrator automatically adjusts the time step throughout the simulation to optimize computational performance. The integration step may become smaller than the one specified in Animation Step but will never be greater. Please see “Variable Time Step” on page A–4 for more information.

Simulation Error Tolerances

During the course of a simulation, Working Model 3D is constantly monitoring various errors such as interpenetrating bodies and constraint violations. At each integration step, Working Model 3D checks its computation results to see if the model satisfies the error bounds. You can fine-tune the following parameters to optimize simulation performance.

By default, Working Model 3D computes a value for these error criteria based on the positional error.

If necessary, you can override the default and specify the value on your own (*it must be greater than zero*). Specifying a small error tolerance results in more computational time. On the other hand, an excessively large tolerance may produce inaccurate simulation results.

Overlap Factor

The amount of interpenetration, or overlap, between two colliding bodies is determined from the geometry of the bodies and the *overlap factor*. In Working Model 3D, each body has a bounding box whose dimensions are specified in the Geometry category of the Properties dialog. When a collision occurs, response impulses and/or forces prevent the interpenetration from exceeding the product of the overlap factor and the minimum bounding box dimension for the two bodies.

Assembly Error

Assembly error is used to bound the numerical error in the Join operation during closed-loop assembly. For example, when you create a four-bar mechanism, an iterative process assembles the mechanism to within the assembly error tolerance.

NOTE: While computing simulations, Working Model 3D uses the parameters specified as Configuration Error to monitor errors (see “Configuration Error” on page A-11). The Assembly Error parameter is only used during assembly processes.

Significant Digits

The value given in the *Significant Digits* box corresponds to the number of digits that are accurate during numerical integrations. The number given in the Significant Digits field sets the relative error, 10^{-n} (see “How Working Model 3D Bounds Errors” on page A-5 for details).

Warnings

When a warning occurs, the simulation pauses and a dialog box appears. The simulation can then be stopped or continued.

Inaccurate Integration

Warns when the mass center of a body has a large velocity or acceleration. This warning can be overridden, but the remainder of the simulation may be inaccurate and/or unstable.

Initial Body Overlap

Warns when two or more bodies initially overlap by more than the overlap tolerance (see “Simulation Error Tolerances” on page A–12), and the bodies are designated as penetrable. See “6.6 Specifying Collisions Among Bodies” for more information.

When two bodies are initially overlapping and are set to collide, simulation results may be inaccurate. See “Preventing Unstable Simulations” on page A–17 for more information.

Redundant Constraints

Warns when more constraints than are necessary exist to constrain an object or system of objects. For example, a body connected to the background with several rigid joints has redundant constraints.

Inconsistent Constraints

Warns when the constraints in the model become physically inconsistent: e.g., when an object driven by a constant velocity motor hits a second, anchored body. *This warning can be overridden to continue the simulation if desired, but the results of the simulation results may be spurious thereafter.*

A.5 Simulating Collisions

This section provides a summary of how collisions are modeled in Working Model 3D. You may find this section helpful in understanding the measurement data obtained from Working Model 3D simulations.

How Working Model 3D Simulates Collisions

Detecting Overlaps

Since Working Model 3D numerically integrates in discrete steps, bodies may overlap by a small amount. For instance, two bodies may be close but not overlapping at time t , but their velocities may cause them to overlap each other in the next frame, at time step $t+h$.

Working Model 3D detects collisions geometrically by finding intersections between bodies. In particular, Working Model 3D reduces all body-to-body collisions to one or more point-to-point contacts. For example, if a vertex of a body collides with a face of another body, the contact point can be defined on the face with a simple geometric calculation. If an edge of a body collides with an edge of another body, the contact point can be defined on each edge as well.

When bodies collide, Working Model 3D computes the forces and/or impulses necessary to prevent interpenetration and applies these forces at the contact points. Based on these forces, Working Model 3D calculates the new velocities of the bodies and continues the simulation.

In variable time step mode, you can define the error tolerance to bound the amount of overlap (see “Simulation Error Tolerances” on page A-12). Working Model 3D automatically uses appropriately small integration steps near collisions and maintains the overlap to be within the tolerance, which is specified by the Overlap Factor in the Accuracy dialog. The Overlap Factor is used to detect interpenetration of colliding bodies in the fixed time step mode as well, but collisions may exhibit overlap greater than the tolerance since the time step is not automatically adjusted.

NOTE: Due to various optimization techniques implemented in Working Model 3D for collision detection, bodies with extremely narrow sections may exhibit excessive overlaps when they are undergoing rapid motions, even with the variable time step. In such cases, simply decrease the time step; for example, simply cut the time step in half.

Collision Impulse

Once a collision is detected, Working Model 3D computes impulses and forces¹ to repel the bodies. Working Model 3D employs an impulse-momentum collision model which uses the coefficient of restitution.

For example, if two particles (of mass m_1 and m_2) are colliding head on with velocities v_1 and v_2 , respectively, then their velocities after the collision (call them v_1' and v_2') can be computed from a pair of linear equations. Note that two unknowns v_1' and v_2' exist in two linearly independent equations:

$$m_1 v_1 + m_2 v_2 = m_1 v_1' + m_2 v_2' \quad (\text{conservation of linear momentum})$$

$$e = \frac{v_2 - v_1}{v_1' - v_2'} \quad (\text{definition of the coefficient of restitution})$$

¹. Working Model 3D computes forces as opposed to impulses once bodies achieve relatively stable contact.

where e is the coefficient of restitution.

Computing Collision Forces

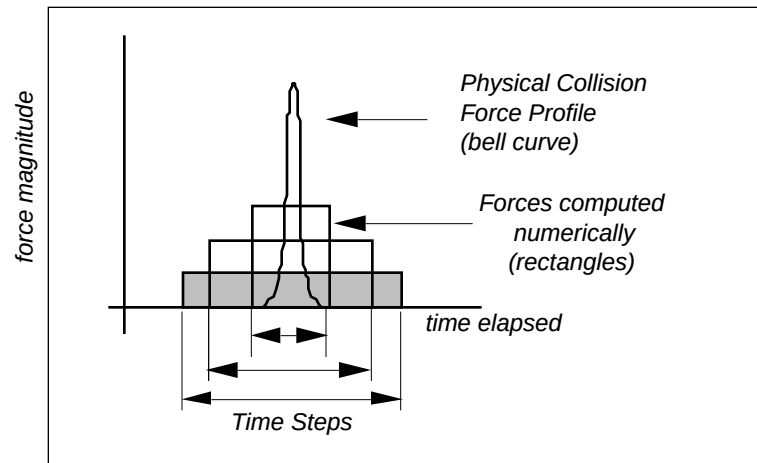
Using the notations provided in the previous section (“How Working Model 3D Simulates Collisions” on page A-14), Working Model 3D reports the collision force f_1 acting on the mass m_1 as:

$$f_1 = m_1 e \frac{v_1' - v_1}{\Delta t} \quad (\text{collision force reported in Working Model 3D})$$

where Δt is the animation step. Therefore, note that the measurement of the collision force in Working Model 3D *depends on the size of the animation step*, whereas the collision impulse $f_1 \Delta t$ does not.

In physical experiments, a collision force profile typically resembles a spike-shaped bell curve whose *support* (where the function is non-zero; i.e., the physical duration of the collision) is often much smaller than a typical time step used in numerical simulations (see Figure A-4).

Figure A-4
TimeStepNumericalPeakValue,
and the Physical Peak Value



The area underneath the bell curve is called an *impulse*, and this quantity is independent of the time step size. In Figure A-4, rectangles of various sizes represent the “shape” of impulse computed in Working Model 3D. Because the exact duration of the collision nor the shape of the collision

force profile is rarely known even in physical experiments, Working Model 3D reports the collision force as the impulse divided by the animation time step.

Collision Force in Meters

The collision force can be displayed using the contact force meter in Working Model 3D.

► ***To create a meter that measures the collision force:***

- 1. Select the two bodies that will collide**
- 2. Choose Contact Force in the Measure menu**

The contact force meter appears in the document.

The contact force meter reports the sum of the normal force and the collision force. When one body is resting on top of another, the contact force reports non-zero values because of the normal force acting from one body to the other. During collisions, the contact force meter reports the sum of the normal force and collision force.

A.6 Hints for Controlling Simulation Accuracy

Preventing Unstable Simulations

An unstable simulation is visibly apparent as bodies move in random directions at high speeds.

Unstable simulations can occur when two or more unconnected bodies that initially overlap are allowed to collide. If you overlap two bodies without using the Penetrate (in the Object menu) command, large forces will be generated to move the bodies apart when the simulation starts.

Instabilities usually indicate the need for a smaller time step. If an object is moving large distances in a small time and interacting with another object through a joint or contact, the simulation may yield incorrect results and/or instabilities.

A good rule of thumb is that the time step must be small enough to capture small motions that occur in the system. In particular, if you are simulating vibrations, you may need a very small time step. For example, if the highest frequency of vibration is 100 times per second (100 Hz), then you need to pick a time step of 1/400 second, or 4 time steps per vibration cycle.

Other systems requiring a small time step include one in which the mass ratio of interacting bodies is very large (e.g., on the order of 1,000 or more).

In most cases, the variable time step (default in the Accurate mode) automatically adjusts the *internal* time step used in integration. However, a large Animation step may yield an *aliasing* effect, in which a rapid vibration may appear as a low frequency motion, thereby producing a deceiving visual effect. Through a bit of trial and error, you will get a feel for how the time step affects the simulation. See “Time Step and Performance” on page A-4 for more information.

In addition, reducing the value of the Configuration Error increases the accuracy. See “Accuracy Dialog and Simulation Parameters” on page A-9 for more information.

Fine-tuning the Simulation Accuracy

The method used to increase simulation accuracy is similar to that used with finite element analysis packages. Run your simulation several times, increasing the accuracy each time, until the results asymptotically converge.

You can increase accuracy in two ways: With fixed step integration, decrease the time step. With variable step integration, decrease the value of the Configuration Error.

Quantities which serve well as a check on numerical accuracy are the positions of the mass center of bodies which are moving rapidly. For example, when simulating a vehicle suspension system, use one of the fast moving suspension components to check for accuracy. When simulating a swinging chain, use one of the outermost links.

To check for accuracy with a specific component, create position and velocity meters. Record the meter values at a specific time near the end of the simulation. Reset the simulation and cut the time step in half if using a fixed time step method, or cut the Configuration Error in half if

using a variable time step method. Run the simulation again and see if the metered values change significantly. By running with increasing accuracy, you should see any value that you measure converge from simulation to simulation.

If you do not see convergence in a measured simulation variable, there is a good chance that the system is highly dependent on initial conditions or is unstable in some way.

Correcting Systems that Appear to Gain Energy

Systems that gain energy usually need to be simulated more accurately. If a simple pendulum begins to swing higher and higher, the system is gaining energy. If a system is gaining energy, switch to the Kutta-Merson integrator, reduce the time step, or try using a variable time step.

Chaotic Systems in Working Model 3D

Simulation accuracy depends to a large extent on the physical system being modeled. Certain physical systems, such as a four bar linkage with a single driving force, lend themselves to very accurate simulation. On the other hand, some physical systems are very sensitive to initial conditions, mass distribution, applied forces, and/or geometrical effects.

If your system would never produce the same results twice in a row in the real world, you can only expect a simulation to give you insight into possible behaviors of the system. Dropping a linked human figure down a flight of stairs is such an example.

If your system is reproducible in the real world, you should be able to get simulation accuracy to any degree you choose.

A.7 Technical References

Interested readers may wish to refer to more comprehensive documentation on numerical methods and mechanics. The following are but a few books of the vast array of literature devoted to the subject matters.

On Numerical Methods

Leslie Fox: *Numerical Solution of Ordinary and Partial Differential Equations*, Addison-Wesley, 1962.

Robert W. Hornbeck: *Numerical Methods*, Prentice-Hall, 1961.

Gene H. Golub and James M. Ortega: *Scientific Computing and Differential Equations: An Introduction to Numerical Methods*, Academic Press, Inc., 1992.

C. William Gear: *Numerical Initial Value Problems in Ordinary Differential Equations*, Prentice-Hall, 1971.

Germund Dahlquist and Åke Björck: *Numerical Methods*, Prentice-Hall, 1974.

Lawrence F. Shampine: *Numerical Solution of Ordinary Differential Equations*, Chapman & Hall, 1994.

On Mechanics

Thomas R. Kane and David A. Levinson: *Dynamics: Theory and Applications*, McGraw-Hill Publishing Company, 1985.

Ferdinand P. Beer and E. Russell Johnston, Jr.: *Vector Mechanics for Engineers*, McGraw-Hill Book Company, 1977.

Nicholas P. Chronis: *Mechanisms & Mechanical Devices Sourcebook*, McGraw-Hill, Inc., 1991.

Edward J. Haug: *Computer-Aided Kinematics and Dynamics of Mechanical Systems*, Allyn and Bacon, 1989.

A. Bedford and W. Fowler: *Engineering Dynamics—Mechanics*, Addison-Wesley Publishing Company, 1995.

R. E. Roberson and R. Schwertassek: *Dynamics of Multibody Systems*, Springer-Verlag, 1988.

R. C. Hibbeler: *Engineering Mechanics—Dynamics*, Macmillan Publishing Co., Inc., 1983.

J. L. Meriam and L. G. Kraige: *Engineering Mechanics, Volume 2, Dynamics*, John Wiley and Sons, 1987.

David J. McGill and Wilton W. King: *Engineering Mechanics: An Introduction to Dynamics*, Brooks/Cole Engineering Division, 1984.

John J. Craig: *Introduction to Robotics*, Addison Wesley, 1989.

A P P E N D I X B

Formula Language Reference

Working Model 3D has an embedded formula language. The language follows standard rules of mathematical syntax and strongly resembles the language used in high-level programming languages and spreadsheet programs.

B.1 Overview

The formula language is case-insensitive. A set of *identifiers*, *operators*, and/or *functions* constitute an expression that has a return value.

The following shows several examples:

- `body[5].p.x`
Returns the x-position of body #5.
- `if(time > 5, vector(2,3,3.2), coord[3].v)`
When time is greater than 5, returns a vector (2, 3, 3.2), otherwise returns the velocity of Coord #3.
- `max(constraint[33].tension, constraint[21].tension)`
Returns the greater of tensions on constraint #33 and constraint #21.

B.2 Scalars and Vectors

Identifier fields (see “B.3 Identifiers”), mathematical functions (“B.4 Mathematical Functions”), and simulation functions (“B.5 Simulation Functions”) in Working Model 3D returns scalar and/or vector quantities. A scalar identifier field or function evaluates to a numerical constant, whereas a *vector* quantity has the three scalar fields .x, .y, and .z.

For example, the position field of a body:

`body[3].p`

is a vector quantity. This vector has the following three scalar fields:

`body[3].p.x`, `body[3].p.y`, `body[3].p.z`

Likewise, the function `vector()` takes three scalar quantities and returns a vector. Therefore:

`vector(1, 2, 3).x`

returns 1.

In this section, assume that all functions/fields return scalar quantities unless otherwise noted.

B.3 Identifiers

Identifiers are symbols to represent an object in Working Model 3D, such as a Body, Coord, or Constraint. The following table lists a complete set of identifiers. All fields return scalar quantities in the current set of units unless otherwise noted.

Body[n]

An identifier `body[n]` represents a body. The following fields are available.

Field	Description
<code>p</code>	Returns a vector describing the position of the body in World coordinates. For example, <code>body[3].p.y</code> returns the y-position of the body[3].
<code>r</code>	Returns a vector describing the orientation of the body in body-XYZ angle.
<code>v</code>	Returns a vector describing the velocity of the body's origin in World coordinates. For example, <code>body[3].v.x</code> returns the x-component of the velocity of body[3]'s origin.

Field	Description
w	Returns a vector describing the angular velocity of the body in World, expressed in the body's coordinate axes. For example, <code>body [7] . w . z</code> returns the angular velocity of <code>body [7]</code> about its z-axis.
a	Returns a vector describing the acceleration of the body's origin.
a lf	Returns a vector describing the angular acceleration of the body in World, expressed in the body's coordinate axes. For example, <code>body [7] . a lf . y</code> returns the angular acceleration of the <code>body [7]</code> about its y-axis.
mass	Returns the mass of the body.
restitution	Returns the coefficient of restitution.
friction	Returns the coefficient of friction.
ke	Returns the kinetic energy of the body.
pe	Returns the potential energy contribution due to a uniform gravitational field.

Constraint [n]

An identifier constraint `[n]` represents a constraint. The following fields are available.

NOTE: Almost all the constraint fields return a property only meaningful to a pair of Coords. Some constraints (e.g. applied forces and applied torques) have only one Coord associated with them; all the constraint field except `p1` return zero or the zero vector for single coord constraints.

Field	Description
p	Returns the position vector from the first Coord to the second Coord, expressed in the first Coord.
v	Returns the velocity of the second Coord relative to the first Coord, expressed in the first Coord.
a	Returns the acceleration of the second Coord relative to the first Coord, expressed in the first Coord.
r	Returns the orientation of the second Coord relative to the first Coord measured in XYZ Euler angles.
w	Returns the angular velocity of the second Coord relative to the first Coord, expressed in the first Coord.
alf	Returns the angular acceleration of the second Coord relative to the first Coord expressed in the first Coord.
force	Returns the force acting from the second Coord on the first Coord, expressed in the first Coord.
torque	Returns the torque acting from the second Coord on the first Coord, expressed in the first Coord.
length	Returns the distance between the two Coords. The return value is identical to the magnitude of the vector constraint[n].p.
tension	For Coord-to-Coord constraints (such as joints and motors), tension returns the magnitude of vector returned by the force vector. For point-to-point constraints (such as spring dampers and linear actuators), tension returns the signed magnitude of the force, where the return value is positive or negative when the constraint is in tension or compression, respectively.
p1	Returns the first Coord constituting the constraint. For example, constraint[3].p1.p returns the x-position of the first Coord (in World coordinates).
p2	Returns the second Coord constituting the constraint. For Force constraints, the p2 field is void.

Coord[n]

An identifier `coord[n]` represents a Coord. The following fields are available.

Field	Description
<code>p</code>	Returns the global position of the Coord expressed in World coordinates.
<code>offset</code>	Returns the position of the Coord from its body or igin expressed in its body's coordinate system. If the Coord is attached to the background, <code>constraint[n].offset</code> is equivalent to <code>constraint[n].p</code> .
<code>v</code>	Returns the velocity of the Coord in World expressed in World coordinates.
<code>a</code>	Returns the acceleration of the Coord in World expressed in World coordinates.
<code>r</code>	Returns the orientation of the Coord in the World, expressed in XYZ Euler angle.
<code>w</code>	Returns the angular velocity of the Coord in World, expressed in its own Coord.
<code>alf</code>	Returns the angular acceleration of the Coord in World, expressed in its own Coord.

Output[n]

An identifier `output[n]` represents a meter. The following fields are available.

Field	Description
x	Returns the value evaluated from the expression specified in the x-axis of the meter.
y1, y2, y3, y4	Each field returns the value evaluated from the expression specified in corresponding column of the meter. Each meter has four columns.

B.4 Mathematical Functions

Working Model 3D formula language provides the following mathematical functions. Also refer to “**B.5 Simulation Functions**” for Working Model 3D-specific functions.

abs

Syntax	<code>abs (number)</code>
Description	Takes a number and returns the absolute value of the number. Example: <code>abs(body[3].p.x)</code> returns the absolute value of body #3's x position.

acos

Syntax	<code>acos (number)</code>
Description	Takes a number and returns the inverse cosine of the number. Values are returned in the range $[0, \pi]$ regardless of the current unit system.

and

Syntax	<code>and (number, number)</code>
Description	Logical AND operation. Takes two numbers and returns the value 1 if both numbers are not 0. Otherwise, returns the value 0. Example:

```
and(time>1 , body[2].v.y>10)
```

returns the value 1 if time is greater than 1 *and* body #2's y velocity is greater than 10.

asin

Syntax `asin(number)`
Description Takes a number and returns the inverse sine of the number. Values are returned in the range $[-\pi/2, \pi/2]$ regardless of the current unit system.

atan

Syntax `atan(number)`
Description Takes a number and returns the inverse tangent of the number. Values are returned in the range $[-\pi/2, \pi/2]$ regardless of the current unit system.

atan2

Syntax `atan2(x, y)`
Description Takes two numbers (*x*, *y*) and returns the inverse tangent of *y/x*. This function is useful because unlike the `atan` function, it can generate an angle in the correct quadrant. Values are returned in the range $[-\pi, \pi]$ regardless of the current unit system.

ceil

Syntax `ceil(number)`
Description Takes a number and returns the smallest integer no smaller than the number.

cos

Syntax `cos(number)`
Description Takes a number and returns the cosine of the number. The parameter *number* must be in radian regardless of the current unit system.

exp

Syntax	<code>exp(<i>number</i>)</code>
Description	Takes a number and returns the exponential of the number. (<i>e</i> raised to the value of the number.)

floor

Syntax	<code>floor(<i>number</i>)</code>
Description	Takes a number and returns the largest integer no larger than the number.

if

Syntax	<code>if(<i>a</i>, <i>b</i>, <i>c</i>)</code>
Description	Takes three numbers. If the value of <i>a</i> is not equal to 0, then returns <i>b</i> . Otherwise, returns <i>c</i> . Example:

`if(time>1, 20, 0)`

returns the value 20 if time is greater than 1, otherwise returns the value 0.

Typically, the first argument of an `if` function is a relation (such as `x > y`) or a logical operation (such as `and(a, b)`). You can write nested if-statements by recursively using other `if()` functions as its own arguments.

For example, shown below is a somewhat naive C-code segment which returns the maximum of three numbers *a*, *b*, and *c*:

```

{
  if (a > b) {
    if (a > c) return a ;
    else return c ;
  }
  else {
    if (b > c) return b ;
    else return c ;
  }
}

```

In the formula language of Working Model 3D, the above segment can be translated into a single line as follows:

```
if(a>b, if(a>c, a, c), if(b>c, b, c))
```

ln

Syntax	<code>ln(<i>number</i>)</code>
Description	Takes a number and returns the natural logarithm of the number.

log

Syntax	<code>log(<i>number</i>)</code>
Description	Takes a number and returns the base 10 logarithm of the number.

mag

Syntax	<code>mag(<i>vector</i>)</code>
Description	Takes a vector and returns the magnitude of the vector.

max

Syntax	<code>max(<i>x</i>, <i>y</i>)</code>
Description	Takes two numbers (<i>x</i> , <i>y</i>) and returns the greater of the two. For example:

```
max(body[1].a.x , body[2].a.x)
```

returns the larger x acceleration of either body #1 or body #2.

If you wish to find the maximum of three numbers *a*, *b*, and *c*, you could recursively use the `max()` functions as follows:

```
max(max(a,b),c)
```

min

Syntax `min(x, y)`
Description Takes two numbers (*x*, *y*) and returns the smaller of the two. Example:

```
min(body[1].v.x , body[2].v.x)
```

returns the smaller x velocity of either body #1 or body #2.

As in the `max()` function, you could find the minimum of three numbers *a*, *b*, and *c* as:

```
min(min(a,b),c)
```

mod

Syntax `mod(x, y)`
Description Takes two numbers (*x*, *y*) and returns the remainder when the first value is divided by the second. For example, `mod(8,7)` returns 1, `mod(8,7.2)` returns 0.8, `mod(8.2, 7)` returns 1.2, `mod(-8.2, 7)` returns -1.2, `mod(8.2, -7)` returns -1.2.

not

Syntax `not(number)`
Description Logical NOT operation. Takes a number and returns the value 0 if the number is *not* 0. Otherwise, returns the value 1.

or

Syntax `or(x, y)`

Description Logical OR operation. Takes two numbers (x, y) and returns the value 1 if at least one of the numbers is not 0. Returns 0 if and only if both numbers are 0. Example:

```
or(time>1 , body[2].v.r>10)
```

returns the value 1 if time is greater than 1 *or* body #2's angular velocity is greater than 10.

pi

Syntax `pi()`

Description Returns the value of π .

pow

Syntax `pow( $x, y$ )`

Description Takes two numbers (x, y) and returns the value of x raised to the power of y ; i.e., returns x^y .

rand

Syntax `rand()`

Description Returns a random value between 0 and 1. Do not omit the trailing pair of blank parentheses; the omission causes a syntax error.

sign

Syntax `sign( $number$ )`

Description Takes a number and returns the value 1 if the number is greater than zero, returns -1, if the number is less than zero, otherwise returns zero.

sin

Syntax `sin( $number$ )`

Description Takes a number and returns the sine of the number. The parameter *number* must be in radian regardless of the current unit system.

sqr

Syntax 1	<code>sqr (number)</code>
Description	Takes a number and returns the square of the number.
Syntax 2	<code>sqr (vector)</code>
Description	Takes a vector and returns its magnitude squared. In other words:
	$\text{sqr}(v) = v.x^2 + v.y^2 + v.z^2$

sqrt

Syntax	<code>sqrt (number)</code>
Description	Takes a number and returns the square root of the number.

tan

Syntax	<code>tan (number)</code>
Description	Takes a number and returns the tangent of the number. The parameter <i>number</i> must be in radian regardless of the current unit system.

vector

Syntax	<code>vector (x, y, z)</code>
Description	Takes three numbers (x, y, z) and returns a vector composed of them. The first number (x) becomes the .x field of the vector. The second number (y) becomes the .y field of the vector. The third number (z) becomes the .z field of the vector.

B.5 Simulation Functions

Working Model 3D provides the following functions that are specifically designed for the formula language. These functions return information taken from Working Model 3D simulations. All functions return scalar quantities in the current set of units unless otherwise noted.

frame

Syntax	<code>frame()</code>
Description	Returns the current frame number. The initial conditions are defined as being frame zero.

frictionforce

Syntax	<code>frictionforce(<i>bodyID1</i>, <i>bodyID2</i>)</code>
Description	Takes the ID numbers of two bodies and returns the vector describing the friction force applied from body[<i>bodyID1</i>] on body[<i>bodyID2</i>]. The vector components are expressed in world coordinates.

normalforce

Syntax	<code>normalforce(<i>bodyID1</i>, <i>bodyID2</i>)</code>
Description	Takes the ID numbers of two bodies and returns the vector describing the normal force or collision force applied from body[<i>bodyID1</i>] on body[<i>bodyID2</i>] while they are in contact. The vector components are expressed in world coordinates.

When bodies are at rest, `normalforce` returns the normal force vector acting on body[*bodyID2*]. During collisions, Working Model 3D employs an impulse-momentum method to compute the collision force, and `normalforce` returns the resulting value. See “A.5 Simulating Collisions” for more information.

length

Syntax	<code>Length(<i>bodyID1</i> or <i>CoordID1</i>, <i>bodyID2</i> or <i>CoordID2</i>)</code>
Description	Takes the ID numbers of two bodies or Coords, and returns the length of the line connecting their coordinate origins.

B.6 Operators

Working Model 3D provides operators, including all of the common arithmetic operations (+, -, >, =). The following operators require one or two numbers. The letters “a” and “b” are used as place holders for any number or formula that evaluates to a number.

Operator Precedence

Use parentheses to set the order of equation evaluation. All equations are normally evaluated from left to right. Precedence is given to operators in the following order. Operators listed in the same row have equal precedence:

() [] . - (unary minus)	highest precedence
* / ^ %	
+ - (binary operators)	
< <= > >=	
== !=	lowest precedence

Arithmetic Operators

Operators with the highest precedence are applied first. For example, the following formula:

$$3 + 2 * 4$$

is evaluated as $3+(2*4)$ instead of as $(3+2)*4$. This is because the multiplication (*) operator has a higher precedence than the addition (+) operator.

Use parentheses to change the order of evaluation, or to assure yourself of the order of evaluation if you're not quite sure of the precedence of various operators. In the above example, you could enter the formula as

$$(3 + 2) * 4$$

to force evaluation of the addition before the multiplication.

You can nest parentheses, as in the formula

$$((3 + 2) * 4 + 10) / 2$$

Be sure to use parentheses, and not brackets ([]) or braces ({}).

Note on Inequalities

Although the inequality operators have the same precedences, the return value of the formula:

```
if (0 <= t <=1, 50, 100)
```

is actually equivalent to:

```
if ((0 <= t) <= 1, 50, 100)
```

since the chain of the binary operators is evaluated from left to right. As a result, the above formula *always* returns 50 regardless of the value t (since $(0 \leq t)$ returns 1 or 0, the entire first argument is always 1, or true). If you want the effect of “return 50 when t is between 0 and 1, or else return 100”, you should type:

```
if(and(0 <= t, t <= 1), 50, 100).
```

Please refer to “**B.4 Mathematical Functions**” for detailed discussions on each function.

- (negate)

Syntax1	- <i>number</i>
Description	Takes a single number and returns the negative of the number.
Syntax 2	- <i>vector</i>
Description	Takes a single vector and returns the vector with equal magnitude and opposite direction (i.e., the vector with all three components negated).

- (subtract)

Syntax 1	$x - y$
Description	Takes two numbers (x, y) and returns the result of subtraction.
Syntax 2	$v - w$
Description	Takes two vectors (v, w) and returns the vector difference.

+ (add)

Syntax 1	$x + y$
Description	Takes two numbers (x, y) and returns the sum.
Syntax 2	$v + w$

Description Takes two vectors (v, w) and returns the vector sum.

*** (multiply)**

Syntax 1 $x * y$

Description Takes two numbers (x, y) and returns the product.

Syntax 2 $number * vector$

Description Takes a number and vector, and returns the scaled vector (i.e., each vector component is multiplied by the scalar quantity).

Syntax 3 $vector * number$

Description Takes a number and vector, and returns the scaled vector (i.e., each vector component is multiplied by the scalar quantity).

/ (divide)

Syntax x / y

Description Takes two numbers (x, y) and returns the quotient.

% (modulus)

Syntax $x \% y$

Description Takes two numbers (x, y) and returns the remainder of the first value divided by the second. Same as `mod(x,y)`.

^ (power)

Syntax $x ^ y$

Description Takes two numbers and returns the first value raised to the power of the second value (i.e., returns x^y). Note, operators evaluate left to right so $2^3^2 = 8^2 = 64$.

> (greater than)

Syntax $x > y$

Description Takes two numbers (x, y) and returns the value 1 if the x is greater than y . Otherwise, returns the value 0.

< (less than)

Syntax $x < y$

Description Takes two numbers (x, y) and returns the value 1 if the x is less than y . Otherwise, returns the value 0.

>= (greater than or equal to)

Syntax $x >= y$

Description Takes two numbers (x, y) and returns the value 1 if the x is greater than or equal to y . Otherwise, returns the value 0.

<= (less than or equal to)

Syntax $x <= y$

Description Takes two numbers (x, y) and returns the value 1 if the x is less than or equal to y . Otherwise, returns the value 0.

== (equal)

Syntax $x == y$

Description Takes two numbers (x, y) and returns the value 1 if the two values are equal. Otherwise, returns the value 0. This operator *does not* assign any value to the left side of the equation. The formula:

`body[3].p.y == 3`

returns 1 if body #3's y position equals 3.0. This formula does not set any values of body #3's position.

!= (not equal)

Syntax $x != y$

Description Takes two numbers (x, y) and returns the value 1 if the two values are not equal. Otherwise, returns the value 0.

B.7 Predefined Variables

Working Model 3D's formula language provides the following system variables:

time

Returns the current time in the simulation. Time always begins at 0.0 in frame 0.

t

Same as time.

A P P E N D I X C

Keyboard Shortcuts and Troubleshooting

This chapter provides references to keyboard shortcuts and troubleshooting tips.

C.1 Keyboard Shortcuts

In addition to standard Windows key shortcuts (such as Alt+F4 for quit), Working Model 3D provides a convenient set of keys to speed up your work.

Shortcut Key	Function
F1	Help
F2	Pan
F3	Zoom In/Out
F4	Rotate Around
F5	Run
F7	Join/Create Constraint
F8	Split
F9	Attach Coordinate System to Selection
F10	Reset Coordinate System
F	Front View
B	Back View

Shortcut Key	Function
T	Top View
L	Left View
R	Right View
M	Bottom View
X	X-Grid
Y	Y-Grid
Z	Z-Grid
A	Automatic Grid
S	Set Home
H	Go Home
P	Previous View
N	Next View
V	View All

C.2 Troubleshooting

This section provides a few tips and solutions that may help you work with Working Model 3D.

Simulations with Polyhedra Run Very Slow

Working Model 3D automatically detects collisions and produces responses based on dynamics principles. However, when the complexity of the bodies increase drastically, you may observe simulations slowing down. In general, you should be aware of the following limitations:

- Simulations with 2,000 or more facets (of bodies that collide) take a long time for the simulation engine to pre-process before calculating frame 1.
- Bodies with more than 1000 facets collide very slowly.

- Working Model 3D may not completely render bodies with more than 3,000 facets.
- Models with more than 20 constraints may assemble extremely slowly.

Simulation with Excessive Body Overlap

Since Working Model 3D employs a discrete time integration method, your simulation may exhibit excessively large overlaps among the bodies in fixed time step mode. Switch to the variable time step mode. See “A.2 Time Step and Performance” for more information.

If your simulation involves complex concave polyhedra, bodies may overlap even with variable time step mode. In such cases, decrease the time step. See “A.5 Simulating Collisions” for more information.

A Coord Does Not Attach Correctly to a Body Surface

When you attach a Coord to a body surface, Working Model 3D automatically positions the Coord so that it is directly on the surface of the body; not away from it, or not inside of it, regardless of the body geometry. Therefore, if you attach a Coord to a cube with 1 meter on each side, then at least one of the x-, y-, or z-coordinate of the Coord should always be 0.5, because the coordinate origin of the cube is located at its geometric center.

However, suppose that the cube has a slight irregular feature; for example, one of the 12 sides is filleted with a fixed radius. Then the body’s coordinate origin is shifted—due to the feature—from what would be otherwise the geometric center of the original cube. Then you cannot guarantee that the x-, y-, or z-coordinate of any Coord attached on the body’s surface would always be 0.5.

See “Local Coordinates of a Body” on page 6–20 for more information.

Menu Items are Disabled

If Working Model 3D exhibits disabled menu titles in the menu bar, reboot your system and restart Working Model 3D. Low memory situations cause the symptom after other Windows application(s) crashes. Rebooting the system properly reclaims the memory/resource.

Index

Note: Index entry with `courier` typeface are formula language functions or expressions.

A

- `abs` B-6
- Absolute Acceptable Error A-7
- acceleration actuator 8-28
- acceleration motor 8-31
- Acceptable Error A-7
- Accuracy
 - Accuracy dialog A-9
 - controlling accuracy 9-9
 - menu command 2-5
- ACIS 6-10, 11-1
 - ACIS Converter 11-2
 - ACIS2WM.EXE 11-2
 - importing 11-2
- `acos` B-6
- Active
 - Properties window 8-3
- actuator 7-17, 8-27
- aliasing effect A-18
- analytical methods A-1
- anchoring bodies 6-24
- `and` B-6
- Animation Step 9-10, A-11
- Appearance
 - Properties window 2-3, 6-19, 7-6, 8-3
- Arrow
 - toolbar button 2-11
- `asin` B-7
- assembly
 - closed- and open-loop 7-33
- assembly behavior
 - Create Constraint window 7-15
- Assembly error A-13

- Assembly Progress 7-34
- `atan` B-7
- `atan2` B-7
- Attach Coord to Body
 - menu command 2-6
- Attach Coordinate System
 - menu command 2-7
- Attach to Selection
 - menu command 2-7
- attachment
 - modifying position for constraints 7-21
- attachment targets
 - Create Constraint window 7-14

B

- background
 - modifying the color 4-11
- Background Color
 - menu command 2-4
- body
 - anchoring 6-24
 - appearance 6-26
 - box 6-2
 - changing geometry 6-25
 - changing position and orientation 6-21
 - changing velocity 6-22
 - collisions 6-32
 - creating 6-5
 - cylinder 6-2
 - definition 6-1
 - deleting 6-18
 - extruded polygon 6-4
 - mass properties 6-28, 6-30

- moving and rotating 6-16
- naming 6-26
- polymesh 6-5
- rigid 6-1
- selecting 6-13
- showing and hiding 6-28
- sphere 6-3
- types 6-2
- body[n]** B-2
- Box**
 - toolbar button 2-12
- box 6-2
- Box Zoom
 - toolbar button 2-10
- box-select 6-14
- C**
- Cascade
 - menu command 2-8
- ceil** B-7
- Center of Mass
 - changing 6-30
 - Properties window 2-3, 6-19
- Central Inertia
 - Properties window 2-3, 6-19, 6-28
- central inertia matrix 6-29
- central inertia panel 6-30
- Close
 - menu command 2-2
- closed-loop assembly 7-33
- coefficient
 - of friction 6-32
 - of restitution 6-31
- Collide
 - menu command 2-6
- collision 6-32
 - computing forces A-16
 - force A-14
 - force meters A-17
 - impulse A-15
 - normalforce** B-13
 - simulation of A-14
- conditional statement
 - if** function B-8
- Configuration Error 9-10, A-11
- constraint
 - appearance 7-25, 8-3
 - changing types 7-24
 - classes 7-18
 - Coord-on-Coord 7-15
 - Create Constraint window 7-13
 - creating 7-7
 - creating multiple 7-12
 - definition 8-1
 - deleting 7-27
 - modifying 7-18
 - modifying attachment positions 7-21
 - naming 7-25, 7-26
 - point-to-point 7-15
 - properties 7-18, 7-22
 - Properties window 8-3
 - selecting 7-19
 - showing and hiding 7-27
 - splitting and joining 7-30
 - tracking position 7-27
 - types 7-20, 8-3
- Constraint and Coord Appearance 3-11
- constraint type
 - Create Constraint window 7-14
- constraint[n]** B-3
- Coord
 - definition 7-2
 - toolbar button 2-13
- coordinate system 3-1
 - attaching to objects 3-2
- Coordinate System dialog 3-5
- Coordinates Bar 2-15
- Coord-on-Coord constraint 7-15, 7-18
- Coords
 - attaching and detaching from a body 7-4
 - creating and attaching 7-4
 - modifying properties 7-5
- coord[n]** B-5
- Copy

- menu command 2-3
 - toolbar button 2-10
- cos** B-7
- Create Connecting Body
 - menu command 2-6, 6-11
- Create Constraint window 7-13
- current length
 - of spring damper 8-23, 8-33, 8-35, 8-37
- current rotation
 - of spring damper 8-26
- Cut
 - menu command 2-3
 - toolbar button 2-9
- Cylinder
 - toolbar button 2-13
- cylinder 6-2

D

- Delete
 - menu command 2-3
- density 6-31
- Detach Coord from Body 7-5
- document
 - window 2-1
- Document Properties
 - menu command 2-2
- documentation 1-5
- Drag
 - toolbar button 2-11
- Drag-Z
 - toolbar button 2-12

E

- Edit Grid 3-7
 - changing properties 3-9
 - specifying orientation 3-8
- Edit menu 2-3
- Edit Toolbar 2-11
- Erase History 9-4
 - menu command 2-5
- errors A-5

- Euler integration 9-10, A-10
- Exit
 - menu command 2-2
- exp** B-8
- export
 - available formats 11-1
 - exporting meter data 11-9
 - Video for Windows 11-7
- Export Meters
 - menu command 2-2
- Export Video
 - menu command 2-2
- Extruded Polygon
 - toolbar button 2-12
- extruded polygon 6-4

F

- faceting 11-5
- face-to-face join 7-16
- File menu 2-2
- files
 - opening, saving, and closing 9-1
- fixed point 5-3
- fixed time step A-12
- fixing body in world 6-24
- floating point display
 - automatic 5-3
 - fixed point 5-3
 - floating point 5-3
- floor** B-8
- Force
 - toolbar button 2-13
- force 7-17
 - applied 8-38
 - properties 8-39
- force actuator 8-28
- formula language B-1
- formulas
 - customizing constraints 12-3
 - unit conversions 12-2
 - units 12-1
- frame

- moving to a specific 9-7

- frame B-13

- friction

- coefficient of 6-32

- friction force B-13

- frictionforce B-13

G

- Geometry

- Properties window 2-3, 6-19

- global coordinate system

- attaching to a body 3-4, 7-6

- attaching to a Coord 3-5, 7-6

- effects of attaching 3-3

- effects of making permanent 3-4

- resetting 3-6

- global coordinates 2-7

- current coordinate system 3-3

- Go Home

- menu command 2-4

- Gravity 5-3

- controlling the force 5-4

- menu command 2-5

- none 5-3

- vertical 5-3

- Grid Extents 3-10

- Grid menu 2-7

- Grid Orientation

- menu command 2-7

- Grid Settings

- menu command 2-7

- Grid Snap 6-10

- snap size 3-11

- Ground Plane 5-4

- menu command 2-5

H

- Help Menu 2-8

- Home View 4-6

I

- if B-8

- impact *See* collision

- Import

- menu command 2-2

- import

- ACIS 11-2

- available formats 11-1

- Working Model 3D files 11-6

- impulse

- collision impulse A-15

- Inaccurate integration A-13

- Inconsistent constraints A-14

- inertia

- central inertia matrix 6-29

- initial condition

- setting 9-5

- Initial mass overlap A-14

- installation 1-4

- installing Working Model 3D 1-4

- integration

- numerical integration A-2

- time step A-12

- locked A-12

- variable A-4, A-12

- Integrator 9-10, A-9

- Euler A-10

- Euler Mode 9-10

- Kutta-Merson A-10

- Kutta-Merson integration 9-10

- Isometric

- menu command 2-4

- isometric projection 4-2

J

- Join

- menu command 2-6

- joining constraints 7-30

- Join/Create Constraint

- toolbar button 2-12

K

keyboard shortcuts C-1
Kutta-Merson integration A-10
Kutta-Merson Mode 9-10

L

length B-13
linear actuator 7-17, 8-27
 assembly behavior 8-27
 properties 8-27
linear spring/damper 7-18, 8-21
 assembly behavior 8-22
 creating 7-10
 properties 8-22
linkage 6-11
ln B-9
load
 attaching force 8-38
 attaching torque 8-39
Local Coordinates 6-20
log B-9
Look At
 menu command 2-4

M

mag B-9
mass 6-31
Material
 Properties window 2-3, 6-19
max B-9
Measure menu 2-7
measurement
 changing units 5-1
menu commands 2-1
meter
 creating 10-1
 customizing 10-3, 12-6
 deleting 10-11
 digital and graph 10-3
 effects of unit change 12-2
 exporting data 11-9

position and size 10-4
printing 10-9
Properties window 10-5, 10-6, 10-8
scaling 10-5
unit labels 10-9

min B-10

mod B-10

Models

 assembling 7-1
 Coords and constraints 7-2

motor 8-29

 assembly behavior 8-29
 properties 8-30
 revolute motor 7-17

N

name

 customizing 6-26, 7-25

natural angle 8-26

natural length

 of spring damper 8-23

New

 menu command 2-2
 toolbar button 2-9

New Window

 menu command 2-8

Next View 4-6

 menu command 2-4

nonlinear systems

 finding patterns 12-7

normal deviation 11-5

normal force B-13

normalforce B-13

not B-10

Numbers & Units 2-5

 menu command 2-5

Numerical Display Format 5-3

numerical integration A-2

numerical method A-1

 references A-20

O

- Object Manager 2-14
 - Connections list 2-14
 - Object list 2-14
- Object menu 2-6
- objects
 - viewing all 4-7
- on-Coord constraint 7-18
- Open
 - menu command 2-2
 - toolbar button 2-9
- open-loop assembly 7-33
- operator
 - arithmetic B-14
- or B-10
- output[n] B-5
- Overlap error A-13

P

- Pan
 - toolbar button 2-10, 4-4
- Paste
 - menu command 2-3
 - toolbar button 2-10
- Penetrate
 - menu command 2-6
- Penetrates All
 - menu command 2-6
- Perspective
 - menu command 2-4
- perspective projection 4-2
- pi B-11
- playback 9-5
 - backward 9-7
 - frame by frame 9-6
 - skipping frames 9-6
 - specifying a frame 9-7
- point-select 6-13
- point-to-point constraint 7-15, 7-18
- polyhedron 6-5
- polymesh 6-5

Position

- Properties window 2-3, 6-19, 7-6
- pow B-11
- precedence
 - of operators B-14
- pre-loading
 - linear spring 8-23, 8-26
- Previous View 4-5
 - menu command 2-4
- Print
 - menu command 2-2
- Print Preview
 - menu command 2-2
- Print Setup
 - menu command 2-2
- printing
 - meters 10-9
 - simulation model 9-8
 - tabulated data 11-12
- Properties
 - menu command 2-3
- Properties window
 - opening 6-20

R

- radius
 - of cylinder body 6-2
 - of sphere body 6-3
- rand B-11
- Redo
 - menu command 2-3
 - toolbar button 2-10
- Redundant constraints A-14
- reference body
 - in a closed-loop assembly 7-35
- Relative Acceptable Error A-6
- rendering
 - shaded 4-9
 - wireframe 4-8
- Reset
 - menu command 2-5
 - toolbar button 2-13

Reset Coordinate System
 menu command 2-7

Resize
 toolbar button 2-12

restitution
 coefficient of 6-31

revolute joint 8-11
 assembly behavior 8-11
 creating 7-7
 properties 8-13

revolute joint on slot 8-13
 assembly behavior 8-14
 properties 8-15

revolute motor 8-29
 assembly behavior 8-29
 properties 8-30

revolute spring/damper 8-24
 assembly behavior 8-24
 properties 8-25

rigid joint 8-5
 assembly behavior 8-5
 properties 8-7

rigid joint on slot 8-8
 assembly behavior 8-8
 properties 8-10

rod 8-36
 assembly behavior 8-36
 properties 8-36

rope 8-32
 assembly behavior 8-32
 properties 8-32

Rotate
 toolbar button 2-12

Rotate Around
 toolbar button 2-10, 4-5

rotating
 body 6-17

rotation motor 8-31

Run
 menu command 2-5
 toolbar button 2-13

Run Toolbar 2-13

S

Save
 toolbar button 2-9

saving documents 2-2

scaling
 meter axes 10-5

Select
 toolbar button 2-11

Select All
 menu command 2-3

selecting objects
 box-select 6-14
 multiple objects 6-14
 point-select 6-13

separator 8-34
 assembly behavior 8-34
 properties 8-34

Set Home
 menu command 2-4

Shaded
 menu command 2-4

Shaded Mode 4-9

Show Grid
 menu command 2-7

show or hide the body 6-28

show or hide the center-of-mass 6-28

sign B-11

simulation
 controlling accuracy 9-9, A-17
 engine 1-2
 erasing history 9-4
 history 9-3
 parameters A-9
 playing back 9-5
 playing backward 9-7
 printing 9-8
 recording 9-3
 running 9-1
 setting initial conditions 9-5
 skipping frames 9-6
 stopping 9-2
 tips for faster 9-11

Simulation Accuracy Dialog A-9
 $\sin$ B-11
 Sketch Toolbar 2-12
 Skip Frames
 menu command 2-5
 Skipping
 Skip Frames 9-6
 skipping frames 9-6
 sleeve joint 8-8
 Snap To Grid
 menu command 2-7
 Snap to Grid 3-11
 Sphere
 toolbar button 2-12
 sphere 6-3
 spherical joint 8-16
 assembly behavior 8-16
 properties 8-18
 spherical joint on slot 8-18
 assembly behavior 8-19
 properties 8-20
 Split
 menu command 2-6
 toolbar button 2-12
 split
 leave constraint split 7-15
 splitting constraints 7-30
 Spring/Damper
 linear 8-21
 revolute 8-24
 $\sqrt{}$ B-12
 $\sqrt[3]{}$ B-12
 Standard Toolbar 2-9
 status bar 2-16
 stepping through a simulation 9-6
 Stop
 menu command 2-5
 toolbar button 2-13
 surface deviation 11-5

T

t B-18

$\tan$ B-12
 Tape Player Control 2-15, 9-2
 Technical references A-19
 Tile
 menu command 2-8
 Time
 menu command 2-7
 time B-18
 time step 9-6, A-3, A-4
 Toggle Isometric
 toolbar button 2-11
 Toggle Wireframe
 toolbar button 2-11
 Toolbars
 menu command 2-4
 toolbars 2-9
 docking 2-9
 showing or hiding 2-9
 Torque
 toolbar button 2-13
 torque 7-17
 applied 8-39
 properties 8-40
 torque motor 8-31
 track the history 6-28
 translucent 6-28
 troubleshooting C-2

U

Undo
 menu command 2-3
 toolbar button 2-10
 unit system
 changing individual units 5-2
 in ACIS converted files 11-4
 setting up 5-1

V

variable time step A-12
 vector B-12
 Velocity

- Properties window 2-3, 6-19
- velocity 6-22
- velocity actuator 8-28
- velocity motor 8-31
- Vertical Gravity 5-3
- Video for Windows
 - compression options 11-8
 - exporting 11-7
- View All 4-7
 - menu command 2-4
 - toolbar button 2-11
- View menu 2-4
- View Toolbar 2-10
- Viewer's Clipping Plane 3-11
- views
 - Head-On views 4-1
 - Look at Selected Object 4-2
 - manipulating 4-1
 - panning 4-4
 - perspective and isometric 4-2
 - rotating 4-5
 - setting and recalling Home 4-6
 - undoing changes 4-5
 - viewing multiple windows 4-7
 - zooming 4-3
- Visibility Options 6-28

W

- Warnings A-13
- Window menu 2-8
- windows
 - creating 4-8
 - deleting 4-8
- Wire Frame
 - menu command 2-4
- Wireframe Mode 4-8
- Workspace 5-1
- world coordinates 2-7, 3-3
- World menu 2-5

X

- XYZ angles 6-23

Z

- Zoom In/Out
 - toolbar button 2-10, 4-4