

Working Model[®] 3D

Version 2.0

for Windows[®]95 and Windows[®]NT

Tutorial Guide



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.

FlashRender is a trademark of Gnomon, Incorporated.

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

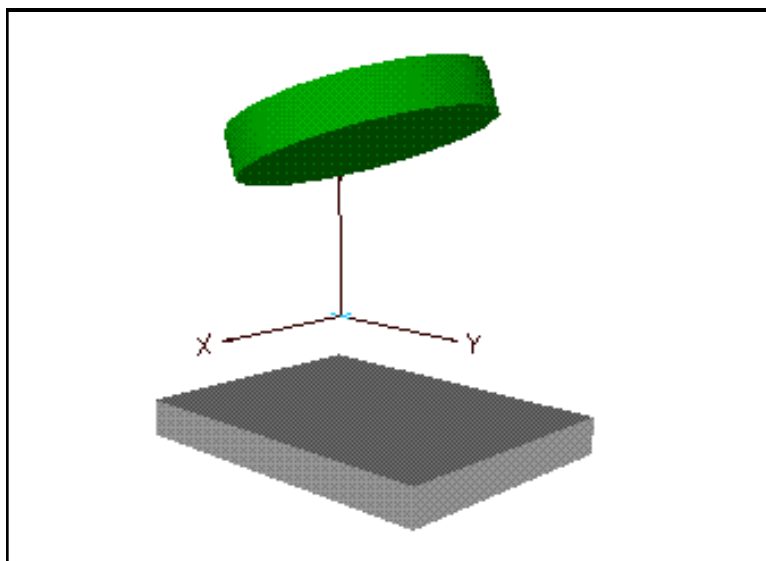
Exercise 1	Coin Drop Simulation.....	1-1
1.1	Starting Working Model 3D.....	1-2
1.2	Drawing the Coin	1-2
1.3	Positioning the coin.....	1-4
	Stepping Back in View	1-6
1.4	Creating a Ground Plane	1-8
1.5	Running the Simulation.....	1-9
1.6	Adding the Shading.....	1-10
1.7	Navigating the World.....	1-11
1.8	Saving File.....	1-13
1.9	Exporting Video for Windows (AVI) File	1-14
1.10	Hints for Faster Animation.....	1-16
 Exercise 2	 Gyroscope Top.....	 2-1
2.1	Setting Up the Workspace.....	2-1
	Changing the Unit System.....	2-2
	Modifying the Edit Grid.....	2-2
	Changing the Animation Step.....	2-4
2.2	Creating the Flywheel.....	2-5
2.3	Creating the Rotation Axle.....	2-6
2.4	Attaching Flywheel to the Rotation Axle.....	2-8
	Coords and Constraints.....	2-8
	Attaching Coord to the Flywheel.....	2-9
	Attaching Coord to the Rotation Axle.....	2-11
	Creating Revolute Joint.....	2-12
2.5	Attaching the Gyroscope to the Ground.....	2-14
	Tilting the Gyroscope.....	2-14
	Creating Attachment Point on the Ground.....	2-16
	Creating Attachment Point on the Gyroscope.....	2-17
	Creating Spherical Joint.....	2-18
2.6	Giving Initial Spin to the Gyroscope.....	2-20
2.7	Running the Simulation.....	2-21

Exercise 3	Piston Engine.....	3-1
3.1	Setting Up the Workspace.....	3-2
	Setting Up the Units.....	3-2
	Setting Up the Edit Grid.....	3-3
3.2	Creating the Crankshaft.....	3-4
	Assigning a Name to the Crankshaft.....	3-7
3.3	Attaching Motor to Crankshaft.....	3-8
	Specifying the Motor Function.....	3-12
3.4	Creating the Connecting Rod	3-14
3.5	Attaching the Connecting Rod to Crankshaft.....	3-15
	Creating Attachment Points.....	3-15
	Creating Spherical Joint.....	3-16
3.6	Testing Your Model	3-18
3.7	Creating the Piston	3-20
	Attaching Piston to the Background.....	3-21
3.8	Attaching Connecting Rod to Piston.....	3-23
	Creating Attachment Point on Connecting Rod.....	3-23
	Creating Attachment Point on Piston.....	3-24
	Creating Spherical Joint.....	3-25
3.9	Running the Simulation.....	3-28
3.10	Taking Measurements	3-29
3.11	Improving the Model.....	3-32
Exercise 4	Robot Manipulator Arm.....	4-1
4.1	Importing ACIS Files.....	4-2
	Converting ACIS Files to WM3 Format.....	4-2
4.2	Setting Up the Workspace.....	4-5
	Setting Up the Units.....	4-5
4.3	Importing WM3 Files.....	4-6
	Naming the Parts.....	4-7
4.4	Arranging the View and Parts	4-8
	Setting the Home View.....	4-8
	Rotating and Positioning Parts.....	4-9
4.5	Creating the Stand	4-12
4.6	Creating the Pedestal.....	4-15
	Creating the Attachment Points.....	4-15
	Creating the Pedestal Cylinder.....	4-16
	Modifying the Pedestal Joints.....	4-19
4.7	Testing Your Model	4-20
4.8	Attaching the Midarm to the Shoulder.....	4-21
	Creating Attachment Points.....	4-21

	Creating a Motor.....	4-23
	Specifying the Motor's Angular Velocity.....	4-26
4.9	Attaching the Forearm to the Midarm.....	4-26
	Creating Attachment Points.....	4-26
	Creating the Motor.....	4-28
	Adjusting the Part Configuration.....	4-29
	Specifying the Motor Characteristics.....	4-30
4.10	Running the Simulation.....	4-31
	Taking Measurements.....	4-31
	Rescaling Meter Axis.....	4-34
Exercise 5	Working Model for Mechanical Desktop.....	5-1
	5.1 Importing a Mechanical Desktop Drawing	5-1
	5.2 Running the Model.....	5-3
	5.3 Improving the Model.....	5-3
	5.4 Testing the Improvements.....	5-7
Exercise 6	Working Model for Solid Edge.....	6-1
	6.1 Importing a Solid Edge Drawing.....	6-1
	6.2 Running the Model.....	6-3
	6.3 Improving the Model.....	6-4
	6.4 Testing the Improvements.....	6-7
Exercise 7	Working Model for SolidWorks.....	7-1
	7.1 Importing a SolidWorks Drawing.....	7-1
	7.2 Running the Model.....	7-2
	7.3 Improving the Model.....	7-3
	7.4 Testing the Improvements.....	7-5

EXERCISE 1

Coin Drop Simulation



This very first exercise in Working Model 3D introduces you to its most fundamental features: modeling and running.

Exercise 1 Concepts:

- Creating bodies
- Running and stopping a simulation
- Manipulating views
- Saving file
- Exporting Video for Windows file

1.1 Starting Working Model 3D

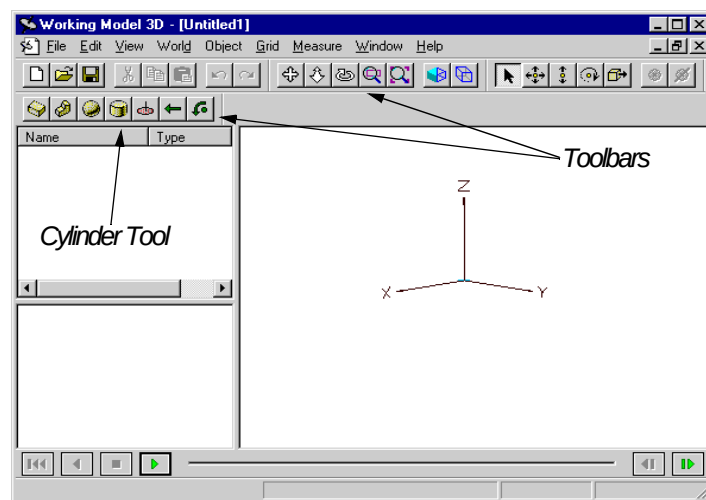
To start Working Model 3D:

1. **Launch the Working Model 3D program.**

Working Model 3D starts and presents a blank document window as seen in Figure 1-1.

Figure 1-1

Working Model 3D with a Blank Document Window



2. **Choose Show Grid in the Grid menu.**

The Edit Grid appears in the window.

1.2 Drawing the Coin

You will create a coin in the document.



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

The Cylinder tool icon appears indented to show that the tool is currently selected.

2. **Position the pointer anywhere on the Edit Grid and click the mouse button. Then move the mouse to expand the circle.**

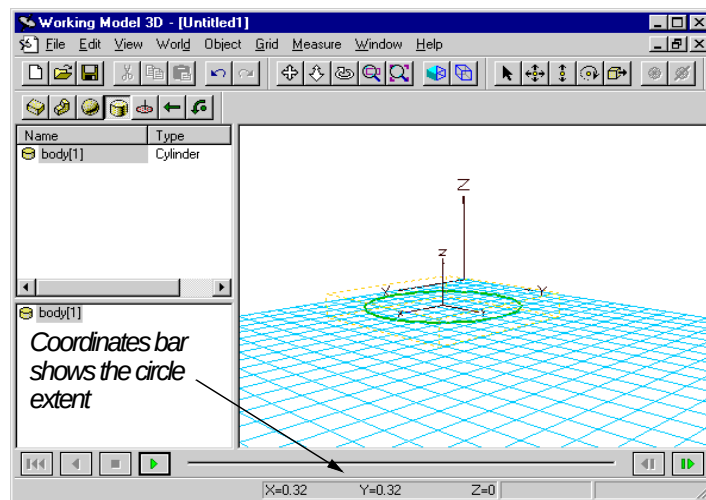
As you move the mouse, a circle appears on the Edit Grid. Note that the circle expands from the center, and the dimension of the circle “snaps” to the grid units.

3. **Move the mouse around until the diameter of the circle reaches 0.2 or 0.3 meters (see Figure 1-2). Click the mouse button.**

Notice how the coordinates bar shows the extent of the circle.

By clicking a second time you have completed the bottom surface of the cylinder. You are just about to extrude the circle up or down to complete the cylinder.

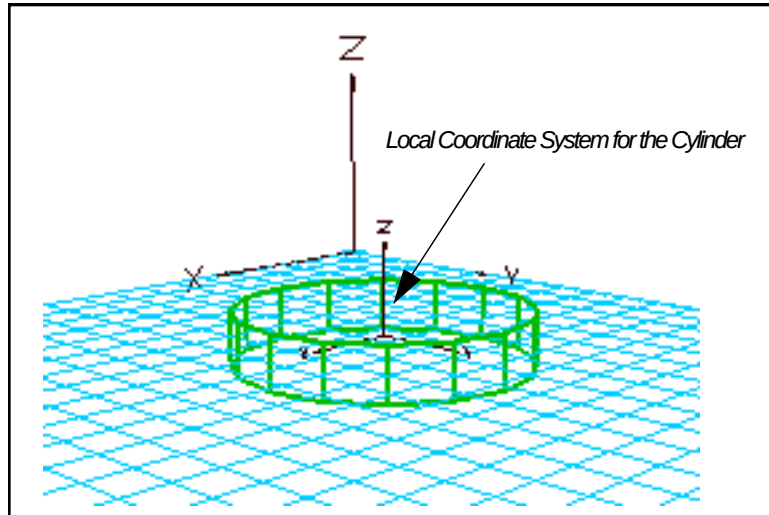
Figure 1-2
Drawing a Coin



4. **Move the mouse upwards slightly to extrude the circle. Click the mouse button once again to complete.**

See Figure 1-3 for example. Again, notice how the mouse movement snaps to the grid units, resulting in a “jumpy” shift of the cylinder height.

Figure 1-3
Completed Coin



Also, note that the coin has a set of axes to represent the local coordinate system (see Figure 1-3). The coordinate origin coincides with the geometric center of the body.

1.3 Positioning the coin

Initially the coin rests on the xy plane. You will move the coin away from the plane, and rotate it slightly.

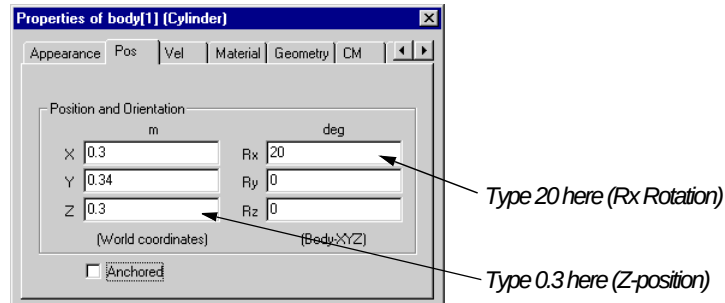
1. Double-click the coin.

The Properties window appears (Figure 1-4). This window “floats” above other windows and displays information about the selected objects.

2. Change the z-position of the coin by typing “0.3” into the “Z” edit box (as shown in Figure 1-4). Press Enter.

Notice that the coin moves upwards immediately.

Figure 1-4
Properties Window



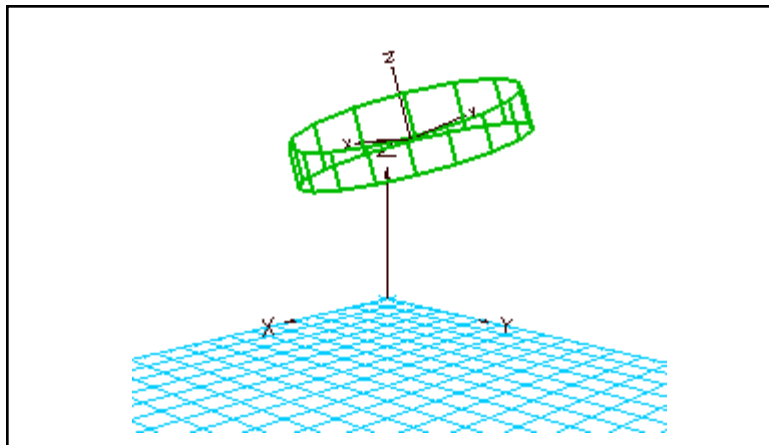
The body position is represented as the position of the coordinate origin of the cylinder in the global coordinates. The coordinate origin of the cylinder coincides with the geometric center of the cylinder.

3. **Change the orientation of the coin by typing 20 into the Rx edit box. Press Enter.**

The coin rotates by 20 degrees (see Figure 1-5) about the body's x-axis.

Working Model 3D expresses the orientation of the body in the local coordinate system, using *body XYZ* angles. Exercise 2 discusses the body XYZ angle representation in more detail. For now, it suffices to say that the body orientation is expressed in terms of a series of rotations about its cardinal axes.

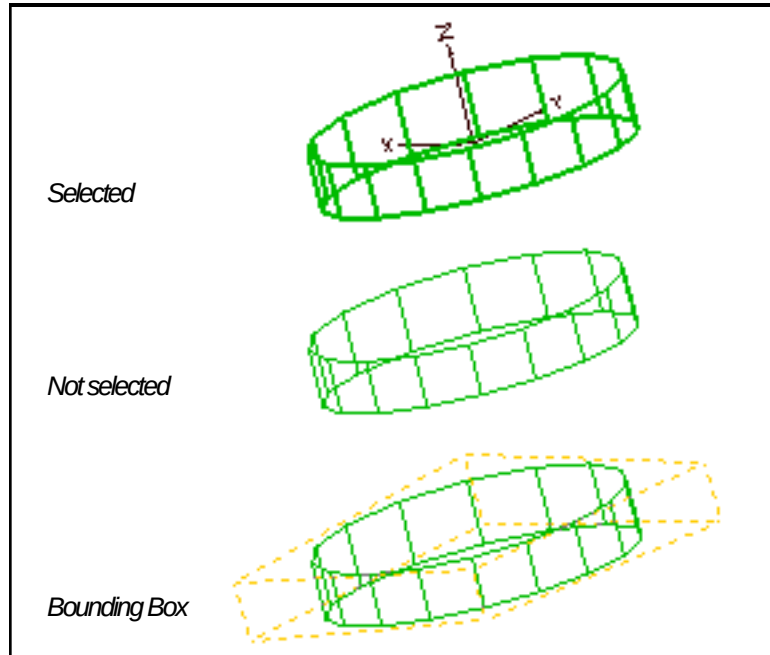
Figure 1-5
Rotated Coin



4. Close the Properties window.
5. Click the blank area of the screen.

The thick edges disappear (Figure 1-6).

Figure 1-6
Selection Handles



When you select a body, the edges thicken (as shown in Figure 1-6) to indicate that the body is selected and the local coordinate system becomes visible.

When you deselect a body, the edges become thin again and the local coordinate system becomes invisible.

When you move the mouse over a body, the bounding box becomes visible to show which body will be selected if you click now.

Stepping Back in View

Currently, your document window shows the coin and a portion of the Edit Grid. To observe the dropping coin later, you will “step back” to get a bigger picture.



1. Click the Zoom In/Out tool on the View toolbar.

The mouse cursor turns into the Zoom icon.

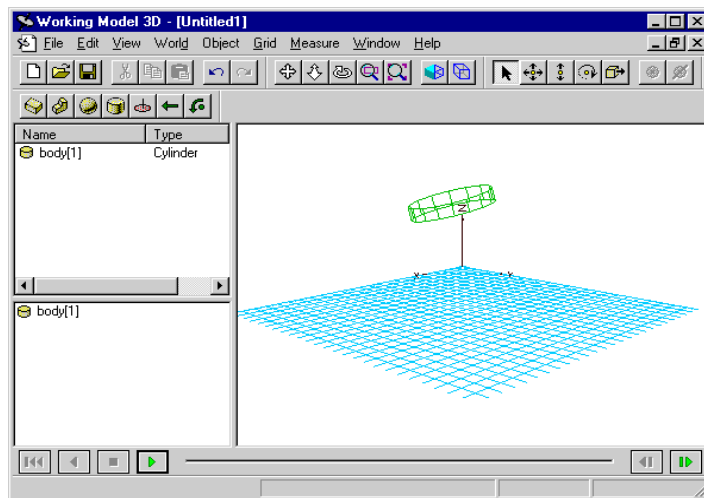
2. Click near the origin of the coordinate axes, hold down the mouse button, then move the mouse up and down.

Observe that the document view changes as if you are stepping backward (as you move the mouse up) and forward (as you move the mouse down).

3. When the entire Edit Grid is visible, release the mouse button.

Your view may resemble Figure 1-7. If the Edit Grid does not appear to be centered, proceed to the next step.

Figure 1-7
Centered Edit Grid



4. If the Edit Grid does not appear to be centered, click the Pan tool.

The mouse cursor turns into the Pan icon.

5. Click anywhere in the document, hold down the mouse button, and drag the mouse.

Observe that the entire “scene” pans in the direction of the mouse movement.

6. Repeat the panning as necessary until your view resembles Figure 1-7.

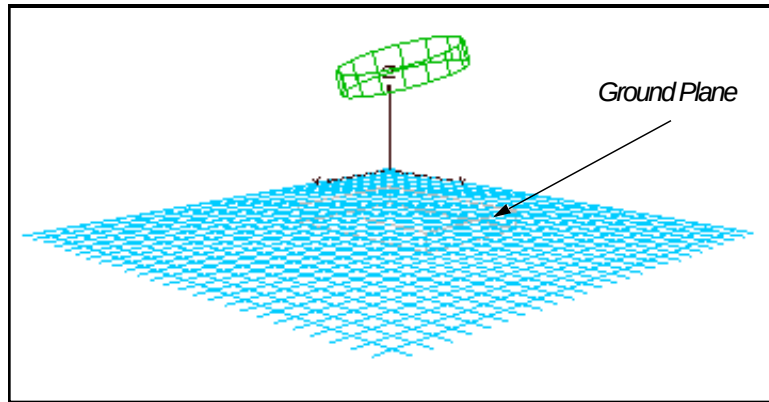
1.4 Creating a Ground Plane

You will now create a simple table-top for the coin to fall on.

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

A large flat body appears just beneath the cylinder (Figure 1-8).

Figure 1-8
Ground Plane



Working Model 3D automatically sizes the Ground Plane based on the maximum (x, y) extent of the bodies in the document. The top surface of the plane is always at $z = 0$. Also, the Ground Plane is not subject to the gravity by default and will not fall during the simulation.

Bodies will pass through each other unless you specify that they should collide. The following steps will make the coin collide with the table.

2. Choose **Select All** in the **Edit** menu.

Both the coin and the ground plane are selected.

3. Choose **Collide** in the **Object** menu.

The selected objects will now collide when you run.

4. Click the blank area of the document to deselect all objects.

1.5 Running the Simulation

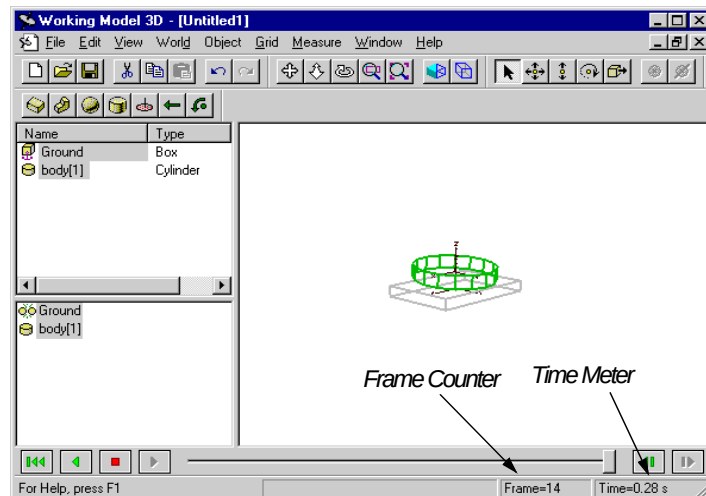
You are now ready to watch the coin drop and hit the table.



1. Click the Run button in the Tape Player Control at the bottom left corner of the document window.

The coin falls, hits the table, and bounces a few times (Figure 1-9). The Frame Counter and Time Meter, also located at the bottom of the window, inform you of the progress.

Figure 1-9
Bouncing Coin



2. Click the Reset button in the Tape Player Control.

The coin returns to its original position.



3. Click the Run button again.

This time the coin will move faster and more smoothly since Working Model 3D is simply playing back the previously calculated simulation history.

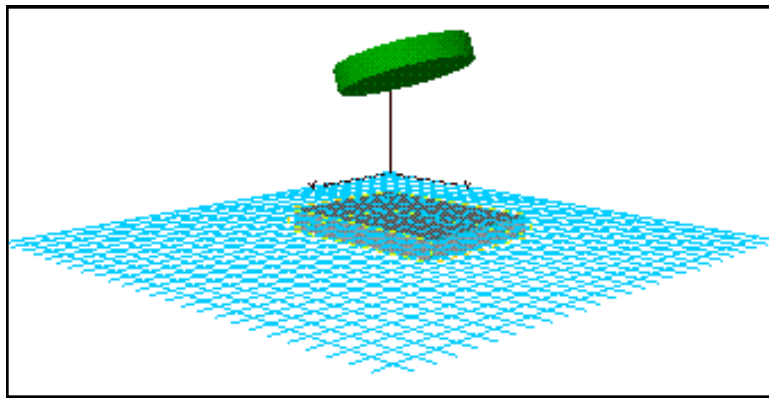
1.6 Adding the Shading

Working Model 3D has a built-in rendering capability to add a far greater realism to the image compared to the wireframe rendering.

1. Choose **Shaded** in the **View** menu.

The green coin and the grey Ground Plane appear shaded.

Figure 1-10
Shaded Image



2. Click the **Run** button in the **Tape Player Control**.

The simulation runs with the shaded coin.



3. Click the **Reset** button after running for a few dozen frames.

NOTE: When two objects become very close to each other, you may notice broken lines or fuzzy edges. Working Model 3D's graphic rendering system is optimized for smooth, rapid animation of shaded objects so that it works on both 16-bit and 32-bit systems. The graphic irregularity is a byproduct of the optimization and does not affect the simulation computation at all.

1.7 Navigating the World

This section introduces you to the versatile graphics manipulation capabilities of Working Model 3D. You can gain better understanding of your model when looking at it from various view angles. In addition, the motion associated with the changing view provides a powerful depth cue, especially in the perspective mode, because objects appear to move at different rates depending on where they are located in the three-dimensional space.



1. **Click the Pan tool on the View toolbar, or press F2 on the keyboard.**

The mouse cursor changes to the Pan tool.

2. **Click anywhere in the workspace and drag the mouse to the right.**

As you drag the mouse to the right, the view shifts accordingly.

3. **Press P on the keyboard.**

Pressing P on the keyboard is equivalent to choosing Previous in the View menu. Your view jumps back to the previous position.



4. **Click the Zoom In/Out tool on the View toolbar, or press F3 on the keyboard.**

The mouse cursor changes to the Zoom tool.

5. **Click anywhere on the coin and drag the mouse upwards.**

As you move the mouse down, the view is zoomed in. Moving the mouse up zooms out.



6. **Click the Rotate Around button on the toolbar, or press F4 on the keyboard.**

The Rotate Around tool becomes selected, and a dotted line circle appears around the center of the coin.

7. **Click anywhere inside the circle, and drag the mouse around.**

As you drag the mouse around, your view rotates around the center of the circle. This tool is a very powerful way to change your view and it also provides useful depth cues through motion.

8. Press F on the keyboard.

Pressing F on the keyboard is equivalent to choosing the menu item View->Look At->Front View. Your view jumps to a front view of your model.

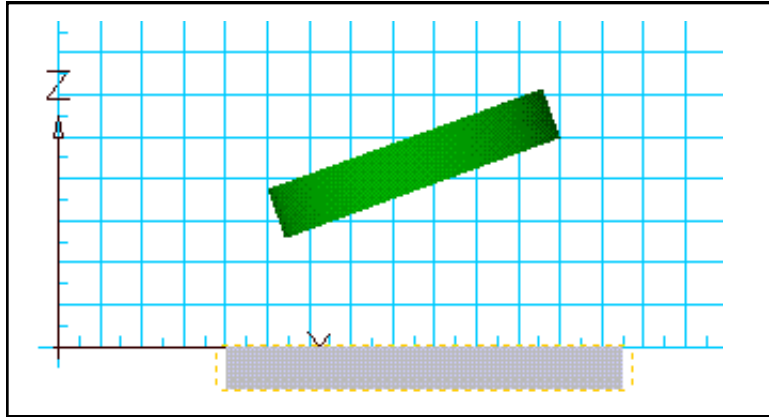
Working Model 3D provides six shortcuts to view the model in predefined directions. You can access the views in the following directions with the corresponding shortcut key.

View from	Shortcut Key
Front (positive X)	F
Back (negative X)	B
Top (positive Z)	T
Bottom (negative Z)	M
Right (positive Y)	R
Left (negative Y)	L

9. Choose Isometric in the View menu.

Although the isometric projection appears to have less depth cues than the perspective projection, you can verify alignments of bodies and components, especially using one of the head-on views, in isometric projection.

Figure 1-11
Isometric Projection



10. Choose Perspective in the View menu.

Your view changes back to a “3D” perspective.



11. Click the Run button on the toolbar and then use the view manipulation tools while the simulation is running.

You can change your view, zoom in on objects, rotate around the world, and so forth while you are running.



12. Click the Reset button after running for a few dozen frames.

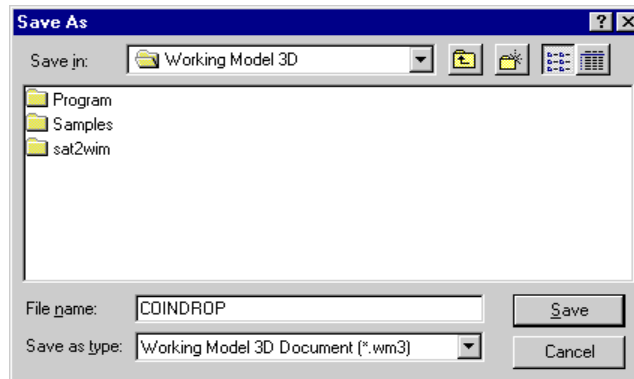
1.8 Saving File

To save your coin drop simulation to a file:

1. Choose Save As in the File menu.

The Save As dialog appears (Figure 1-12).

Figure 1-12
Save As Dialog



2. Choose a directory to which you would like to save the file.
3. Type COINDROP in the File name edit box.

Working Model 3D automatically adds an extension WM3 to the filename. So the file is saved as COINDROP.WM3.

4. Click Save.

The simulation history is saved as well as the model data. When you open the file next time, you can click the Run button to play back the simulation without waiting for Working Model 3D to compute every frame.

1.9 Exporting Video for Windows (AVI) File

Your Working Model 3D simulation is a great way for people to visualize your ideas. To demonstrate the coin drop simulation, you can reopen the file at a later time and play it back; however, you must assume that the Working Model 3D application is available where you want to show the demonstration.

Instead, you can save the simulation file in the Video for Windows format—a common format for animation files under Windows—and distribute the “video” file. Anyone who has a Windows PC can play back the file. In fact, Working Model 3D simulations will play back more quickly as Video for Windows movies.

To export the simulation as a Video for Windows file:

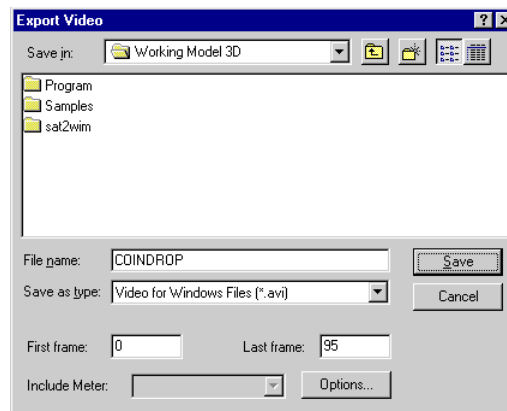
1. **Using the Pan, Zoom In/Out, and Rotate Around tools, arrange the view.**

The Video for Windows file will record the simulation exactly as it appears in the Working Model 3D document window. Make sure to “prepare the scene” before you start exporting.

2. **Choose Export Video in the File menu.**

The Export Video dialog appears (Figure 1-13). Note that Working Model 3D automatically defines the first and last frame numbers according to the existing simulation history.

Figure 1-13
Export Video Dialog



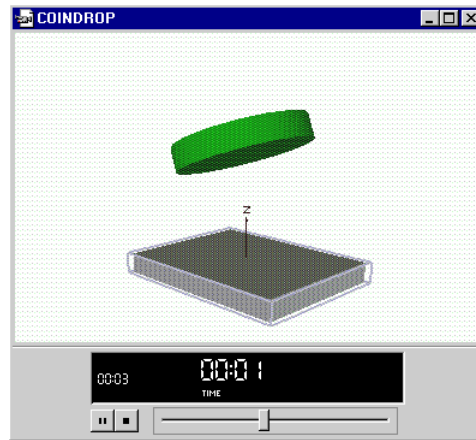
3. **Type COINDROP in the File name edit box.**
4. **Click Save.**

Working Model 3D starts to run each frame to export. After the last frame, a dialog appears to report that exporting has finished.

Working Model 3D saves an AVI file exactly the way the running simulation appears in the document window, including the pixel size of the window. However, the toolbar, menus, tape controls, status bar, and coordinates bar are *not* saved in the AVI file.

To play back the AVI file, simply double click on the AVI file from the Windows File Manager or Explorer.

Figure 1-14
AVI File Playback



NOTE: The Windows Media Player may not handle very large AVI files. If you experience a problem while playing back, try the Video for Windows export again with a reduced window size or number of frames.

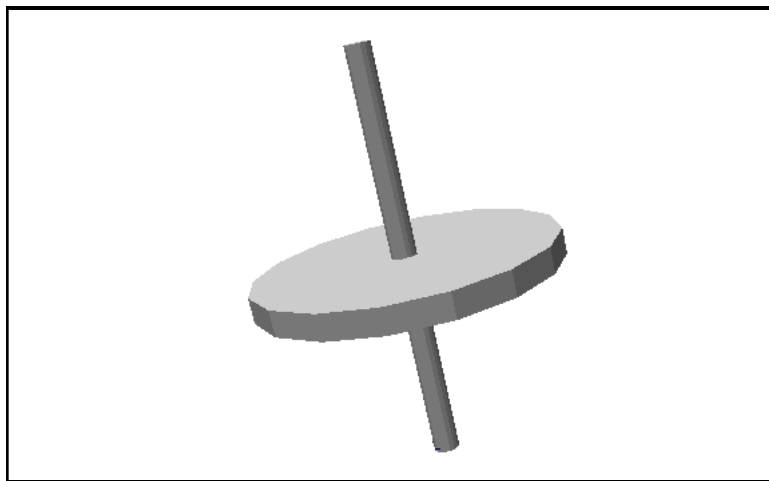
1.10 Hints for Faster Animation

Here are some guidelines to speed up the Working Model 3D animation (playback and editing).

- Use a small window size. The fewer pixels Working Model 3D needs to draw, the faster the rendering becomes. The same goes for the AVI file rendering.
- Use wireframe mode while editing. This option is turned on by default under the View menu. When you drag or resize objects, this option speeds up the editing.

EXERCISE 2

Gyroscope Top



The goals of this exercise are to show you how to create simple constrained bodies in Working Model 3D. The target model is a simple gyroscope top consisting of a flywheel and a rotation axle, fixed to the background with a spherical joint (thus having three rotational degrees of freedom). The gyroscope is given an initial angular velocity to exhibit *precession*, which cannot be demonstrated with a static drawing or two-dimensional modeling.

2.1 Setting Up the Workspace

In this exercise, you will design the gyroscope in the English unit system.

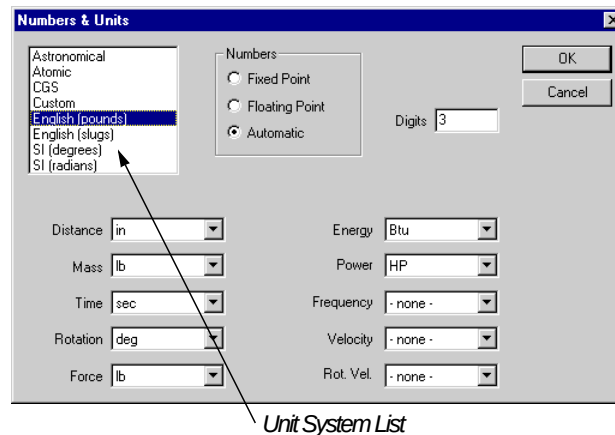
Changing the Unit System

To change the default SI unit system to the English unit system:

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

The Numbers & Units dialog appears (Figure 2-1).

Figure 2-1
Numbers & Units Dialog



2. Select the item “English (pounds)” from the Unit System list
3. Click OK.

All the measurements in the simulation will be done in the English unit system, using inches for distance, pounds for mass, and seconds for time, for the remainder of the exercise.

Modifying the Edit Grid

Although the unit system is changed, the default workspace is scaled rather large for a small mechanism such as a gyroscope top, whose flywheel diameter would be approximately three inches. Each component would appear too small to manipulate.

To modify the scale of the workspace:

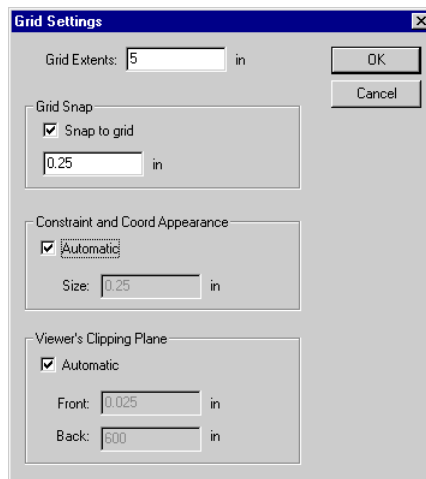
1. Choose **Show Grid in the Grid** menu.

The Edit Grid appears in the document window.

2. Choose Grid Settings in the Grid menu.

The Grid Settings dialog appears (Figure 2-2).

Figure 2-2
Grid Settings Dialog



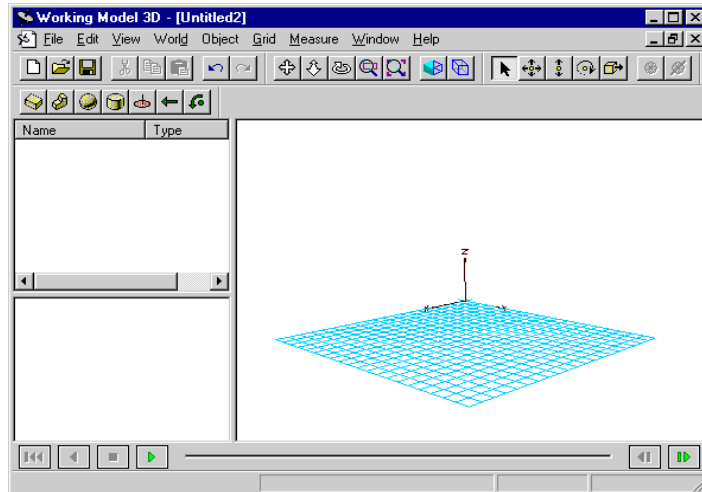
3. Type 5 into Grid Extents and 0.25 into Grid Snap edit boxes.

Note that the units in the dialog are based on inches because you selected the English unit system. Working Model 3D automatically scales the document window to accommodate the smaller Edit Grid.

4. Click OK to close the dialog.

The size of the Edit Grid shrinks on the document as shown in Figure 2-3.

Figure 2-3
Resized Edit Grid



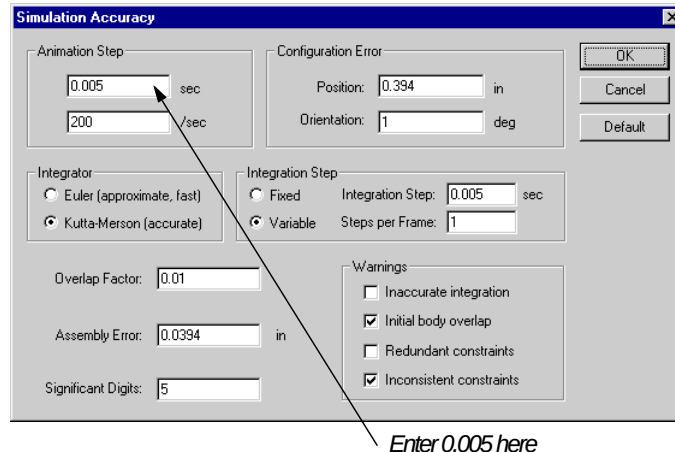
Changing the Animation Step

By default, Working Model 3D sets the Animation Step (that is closely tied to the integration time step) as 0.02 second. In our exercise, you will make the Animation Step smaller than the default value since the gyroscope model in this exercise involves a flywheel that spins very rapidly.

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

The Simulation Accuracy dialog appears (Figure 2-4).

Figure 2-4
Accuracy Dialog



2. Enter the time step as 0.005 seconds as shown in Figure 2-4.

Note that the two edit boxes located together are inverses of each other; i.e., the Animation Step of 0.005 seconds corresponds to the frame rate of 200 frames per second.

2.2 Creating the Flywheel

The simple gyroscope in this exercise consists of a single flywheel and an axle.

To create the flywheel:



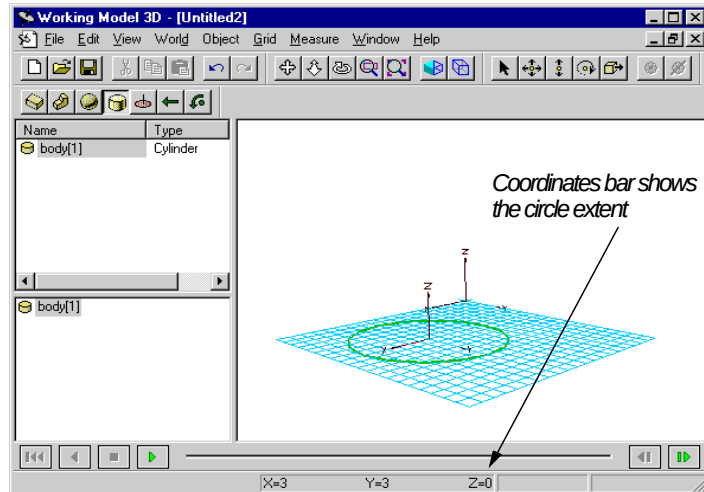
1. Click the Cylinder tool on the Sketch toolbar.
2. Position the pointer anywhere on the Edit Grid (see Figure 2-5), and click the mouse button. Then move the mouse to expand the circle.

As you move the mouse, a circle appears on the Edit Grid.

3. Move the mouse around until the diameter of the circle reaches 3 inches (see Figure 2-5). Click the mouse button.

Notice how the coordinates bar shows the extent of the circle.

Figure 2-5
Drawing the Flywheel



4. Move the mouse upwards slightly to extrude the circle. When the Coordinates bar shows that the thickness reaches 0.25 inches, click the mouse button once again to complete.

2.3 Creating the Rotation Axle

In the coin drop exercise, you created a cylindrical coin by simply sketching on the Edit Grid. This exercise shows you how to specify the body geometry by describing its attributes in the Properties window.



1. Click the Cylinder tool on the toolbar.
2. Click once on the Edit Grid, then press the Enter key on the keyboard.

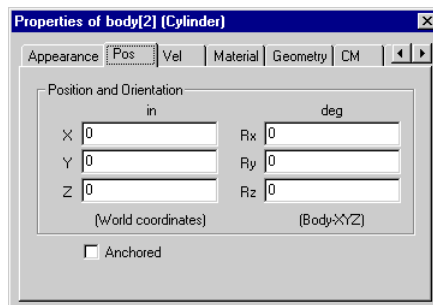
A small cylinder appears on the Edit Grid. This cylinder also appears as body[2] in the Object Manager list, which is located on the left side of the drawing window.

3. Double-click body[2] in the Object Manager list.

The Properties window for the cylinder appears (Figure 2-6).

NOTE: If the small cylinder (body[2]) lies entirely within the larger cylinder, it is only possible to choose this body from the Object Manager list.

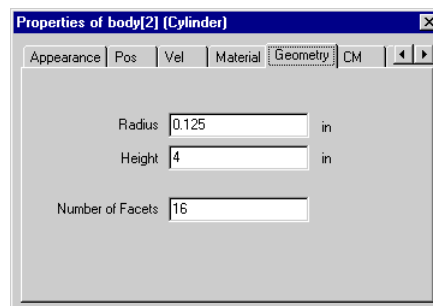
Figure 2-6
*Properties Window:
Position Tab*



4. If needed, click the Position (Pos) tab, then enter the position as (x, y, z) = (0, 0, 0).
5. Click the Geometry tab.

The Geometry tab of the Properties window appears selected (Figure 2-7).

Figure 2-7
*Properties Window:
Geometry Tab*



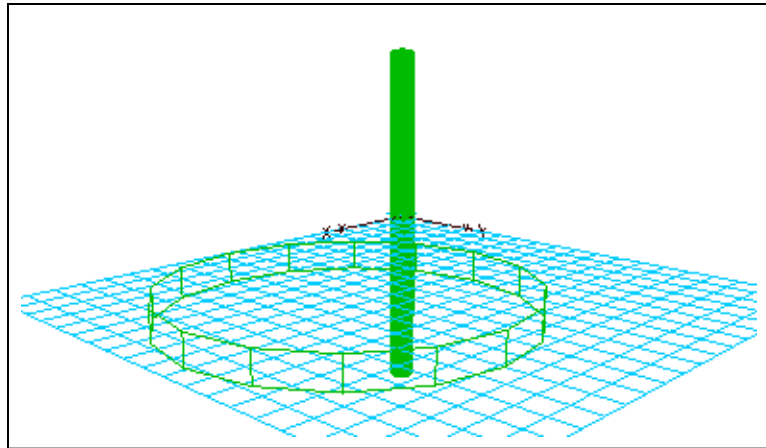
6. Enter 0.125 into the Radius and 4 into the height edit boxes

Note that all the numbers are interpreted in inches.

7. Close the Properties window.

The cylinder is accordingly sized and positioned on the screen (see Figure 2-8).

Figure 2-8
*Rotation Axle Sized
and Positioned*



2.4 Attaching Flywheel to the Rotation Axle

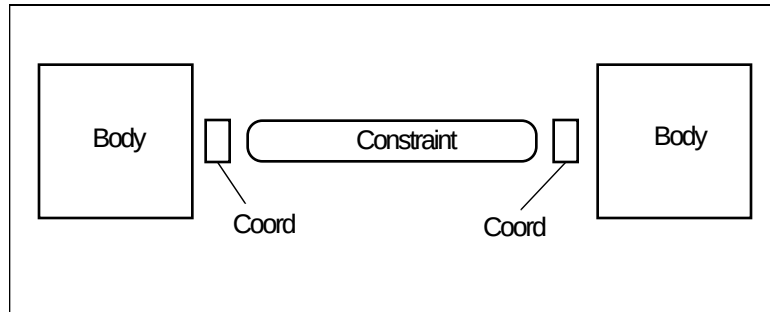
To assemble the gyroscope top, you will use a revolute joint to attach the flywheel to the axle so that the former freely rotates about the latter.

Coords and Constraints

All constraints (except for external forces and torques) in Working Model 3D consist of a pair of objects called *Coords*, each of which are attached to a body or the background. In essence, a *Coord* serves as a medium between a body and the constraint, as illustrated in Figure 2-9.

Figure 2-9

Concept: Construction of a Constraint



In physical terms, a Coord is a point element with a coordinate system attached. Therefore, a Coord has position *and* orientation as properties.

For constraints such as spring dampers and separators, Coords serve as endpoints to those physical constraints. For joints, Working Model 3D uses the orientation of the relevant Coords to determine the constraint conditions.

Exercise 3 “Piston Engine” provides more discussion on Coords and constraints.

Attaching Coord to the Flywheel

You will attach Coords to the flywheel and the rotation axle and create a revolute joint based on the two Coords.

To attach a Coord to the flywheel:



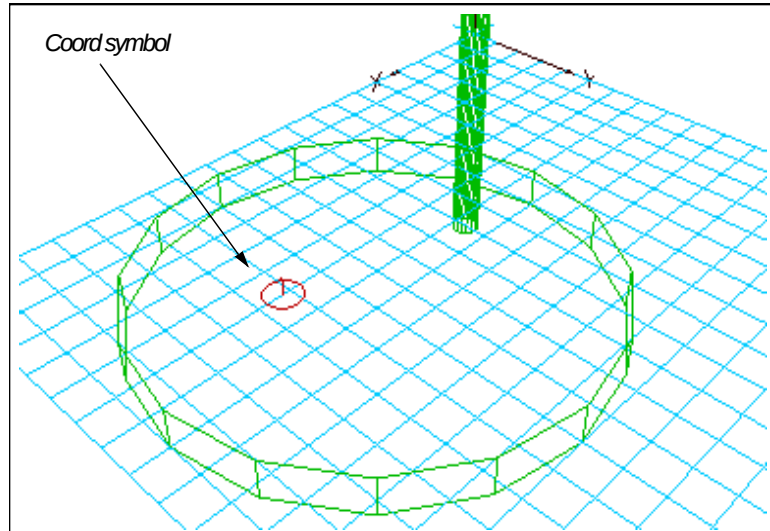
1. Click the Coord tool on the Sketch toolbar.
2. Click anywhere on the flywheel.

A Coord appears with a red circle and the coordinate axes (Figure 2-10). When selected, a Coord displays its triad with labels for x, y, and z axes (Figure 2-10).



To obtain a clear view of the Coord, you may wish to use the Zoom In/Out or Rotate Around tools.

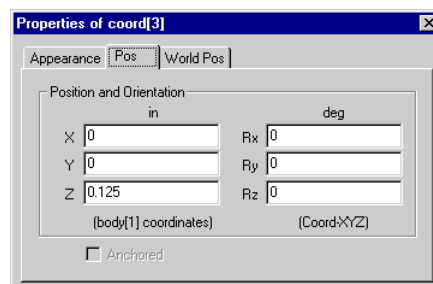
Figure 2-10
Coord on a Flywheel



3. Double-click the Coord in the drawing window or the Object Manager list.

The Properties window appears with the Pos (position) tab selected (Figure 2-11).

Figure 2-11
Properties Window for Coord



Enter these values.

4. Verify that the position and orientation of the Coord are: $(x, y, z) = (0, 0, 0.125)$, and $(R_x, R_y, R_z) = (0, 0, 0)$. If the values appear differently, type these values in the window to position the Coord precisely.

The position and orientation of the Coord are expressed in terms of the coordinate system of the flywheel because the Coord is attached to it.

Attaching Coord to the Rotation Axle

To attach a Coord to the rotation axle:



1. Click the Coord tool on the Sketch toolbar.
2. Click anywhere on the surface of the rotation axle.

A Coord appears with a red circle and the coordinate axes.

3. Double-click the Coord in the drawing window or the Object Manager list.

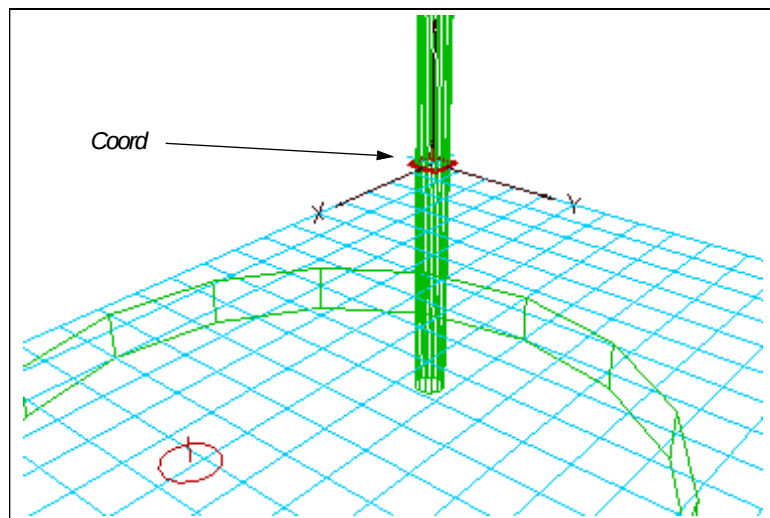
The Properties window appears with the Pos (position) tab selected.

4. Enter the position and orientation of the Coord as follows: (x, y, z) = (0, 0, 0.0), and (Rx, Ry, Rz) = (0, 0, 0).

Again, the position and orientation of the Coord are expressed in the coordinate system of the rotation axle (which, so far, coincides with the global coordinate frame).

The Coord appears “buried” inside the rotation axle (Figure 2-12).

Figure 2-12
Coord Positioned Inside the
Rotation Axle



Creating Revolute Joint

Now that the attachment points are ready, you will proceed to create a revolute joint connecting the two bodies.

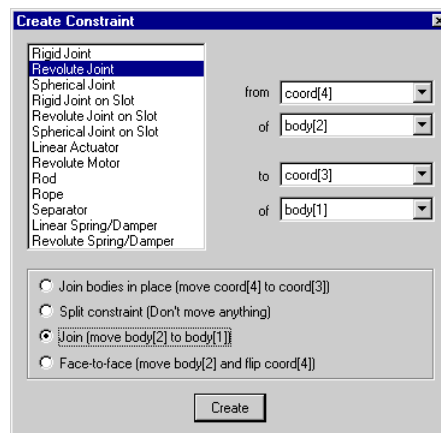
1. Select “coord[4]” (the Coord attached to the rotation axle) in the Object Manager list.
2. Press the Control key on the keyboard and select “coord[3]” (the Coord attached to the flywheel) in the Object Manager list.
3. Click the Create/Join Constraint button on the Edit toolbar.



The Create Constraint window appears (Figure 2-13).

Figure 2-13

Create Constraint Window



4. Select “Revolute Joint” as the joint type in the list of constraints.

The Create Constraint window is a versatile tool to create a constraint between a given pair of Coords. The window presents different options depending on the type of objects that are currently selected.

The window is best understood when you read the first few lines of the window as a complete sentence. In our case, the dialog reads:

“Create a Revolute Joint from coord[4] of body[2] to coord[3] of body[1].”

Therefore, the dialog is set up to create a revolute joint between the two Coords. The variable names such as coord[3] and body[1] appear in the Object Manager list and uniquely identify each object in the Working Model 3D document. Later exercises will show you how you can customize the naming scheme so that you can assign plain English names to these objects.

Meanwhile, the bottom half of the Create Constraint window provides several options. For this exercise:

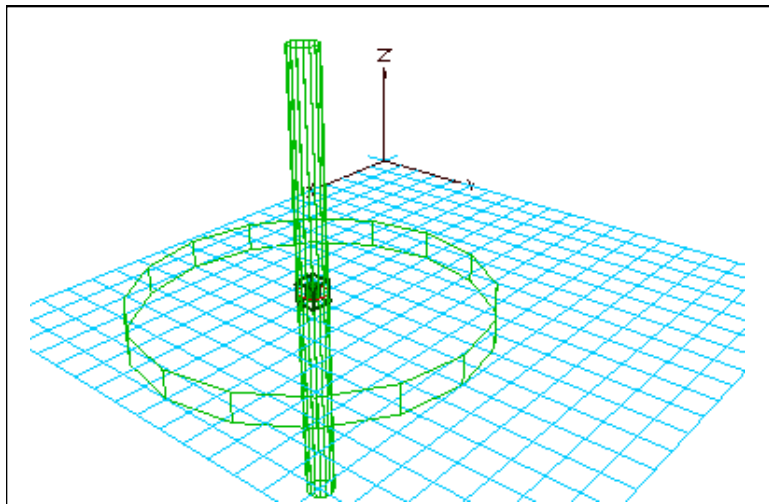
5. Choose the option labeled “Join (move body[2] to body[1])”.

The option indicates that the body[2] (axle) will be moved to the flywheel, so that the two Coords will be aligned (see below for more information).

6. Click the Create button located at the bottom of the window.

The rotation axis is moved to the flywheel, and the revolute joint icon appears at the attachment points (Figure 2-14).

Figure 2-14
Assembled Gyroscope



Note that the rotation axis was moved to the flywheel, because you selected the rotation axis first. For joint constraints, the order of selection thus affects the assembly behavior. Later exercises in this Tutorial Guide discuss more about the assembly behavior.

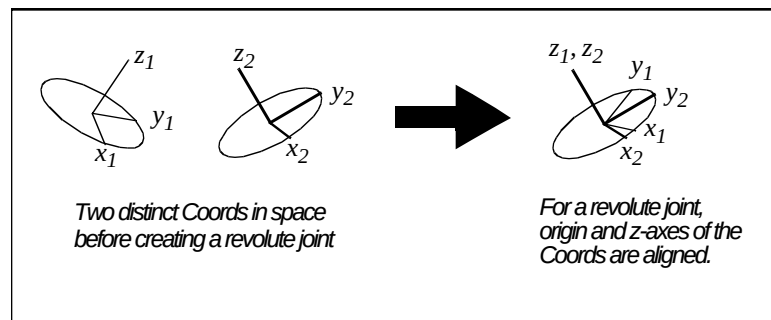
By default, Working Model 3D creates a revolute joint by allowing a single degree of freedom in rotation about the z-axes of the two Coords. Therefore, when you created a revolute joint, note that Working Model 3D aligned the following:

- the origins of the two Coords, and
- the z-axes of the two Coords (in the same direction).

As shown in Figure 2-15, the two Coords will not necessarily be aligned completely, as the revolute joint allows free rotation about the z-axes.

Figure 2-15

Aligning Coords for Revolute Joint



Other joints available in Working Model 3D follow principles similar to the revolute joint. For example, a spherical joint is constructed by aligning the origins of two Coords; since the joint allows three rotational degrees of freedom, the coordinate axes need not be aligned. The next section illustrates how to construct a spherical joint.

2.5 Attaching the Gyroscope to the Ground

The bottom end of the gyroscope axle will be fixed to the ground through a spherical joint to allow three degrees of freedom in rotation.

Tilting the Gyroscope

The gyroscope is currently in an upright position. To observe precession, you will tilt the gyroscope slightly.

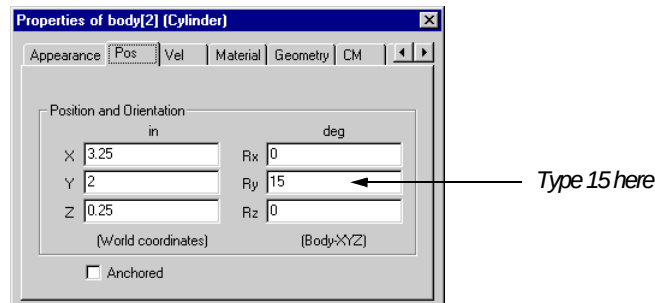
1. **Double-click anywhere on the rotation axle (make sure you do not click the flywheel) or on “body[2]” in the Object Manager list.**

The Properties window appears with the Pos (position) tab selected (Figure 2-16).

2. In the field Ry (orientation about the y-axis), enter 15.

The gyroscope tilts by 15 degrees about the y-axis of the axle.

Figure 2-16
Properties Window
for the Rotation Axle



Note that the flywheel also tilted by 15 degrees and preserved the revolute joint constraint connecting the axle and the flywheel. As a result, the entire gyroscope tilted by 15 degrees.

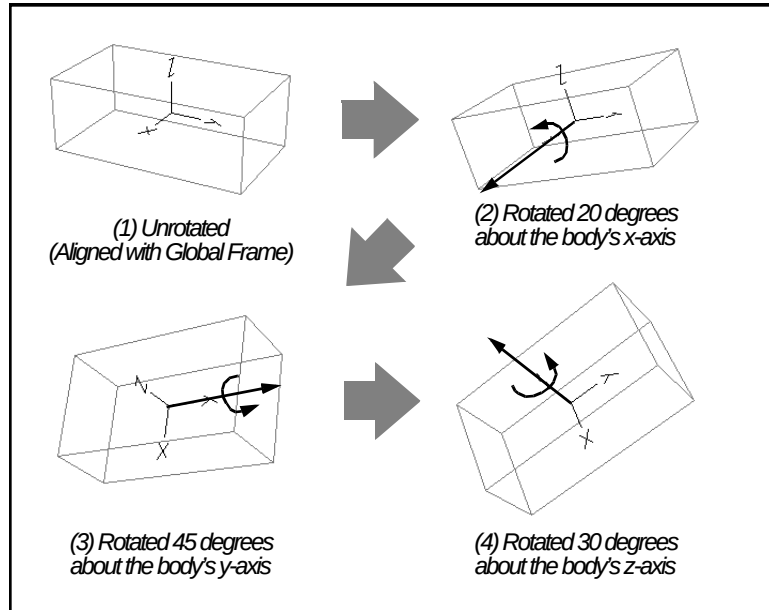
Notes on Body XYZ Angle

It is important to understand that the body was rotated about the y-axis of the body's frame, not of the global frame because Working Model 3D expresses the orientation of a body in *Body XYZ* angles. Body XYZ angles represent the orientation of a body with three parameters (X, Y, and Z angles). Initially, the body is oriented so that its coordinate system orientation matches that of the global frame.

Given the X, Y, and Z angles, the body is rotated first by X, with the body's x-axis fixed in space. Then the body is rotated by Y, with the body's y-axis fixed in space. Finally, the body is rotated by Z, with the body's z-axis fixed in space.

For example, if the Properties window of a box has a parameter (Rx, Ry, Rz) = (20, 45, 30), then Working Model 3D represents the orientation of the box as shown in Figure 2-17.

Figure 2-17
Body XYZ Angle Example



Creating Attachment Point on the Ground

You will attach the gyroscope assembly to the ground using a spherical joint. To create the joint, you will take steps similar to the ones taken when creating the revolute joint.



1. Click the **Coord** tool on the **Sketch** toolbar.
2. Click anywhere on the **Edit Grid**, away from the bodies.

A Coord automatically attaches to the xy plane.

3. Double-click the **Coord** in the drawing window or the **Object Manager** list.

*The Properties window appears with the **Pos** (position) tab selected.*

4. Enter the values to locate the **Coord** at the global origin. That is, $(X, Y, Z) = (0, 0, 0)$ and $(R_x, R_y, R_z) = (0, 0, 0)$.

The Coord immediately moves to the global origin.

Creating Attachment Point on the Gyroscope

To attach another Coord at the bottom of the rotation axle, you will need to look at the bottom.

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

The second window appears, presenting the view of the gyroscope model from the same view angle.

Working Model 3D allows you to open multiple windows for a single document. As shown below, individual windows can have completely independent view angles, yet any changes to the model are updated on all windows simultaneously.

- 2. Choose Tile in the Window menu.**

The two windows are repositioned and appear side-by-side.

You can resize the Object Manager pane of either or both windows to make more room available for the drawing pane by dragging the vertical line that separates the panes.



- 3. Click the Rotate Around button on the View toolbar.**

The mouse pointer changes its shape accordingly. Alternatively, you can select the Rotate Around tool by pressing F4 on the keyboard.

- 4. On the right window, click anywhere on the rotation axle and hold the mouse button.**

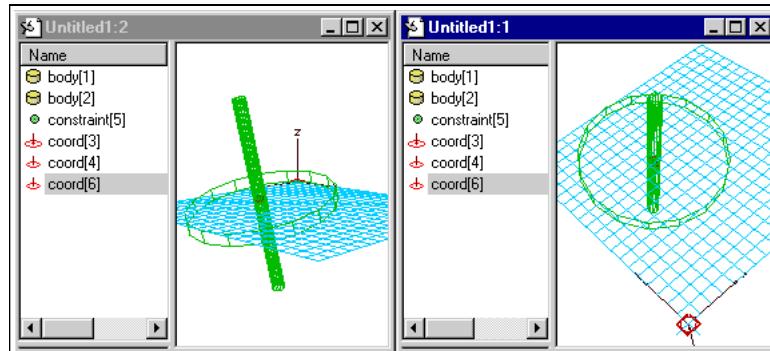
Note that the view in the left window remains unchanged.

- 5. Drag the mouse so that the bottom end of the rotation axle is facing you.**

See Figure 2-18 as an example.

Figure 2-18

Multiple Window Placed Side-by-Side Showing Different Angles



6. Click the Coord tool on the Sketch toolbar.
7. Click at the bottom end of the rotation axle.

Note that left window also shows that the Coord is attached.

8. In the Properties window, verify that the position and orientation of the Coord are $(X, Y, Z) = (0, 0, -2.0)$ and $(R_x, R_y, R_z) = (0, 180, 0)$.

Creating Spherical Joint

You will connect the last two Coords with a spherical joint.

1. In either window, select coord[7] (the Coord attached to the bottom end of the rotation axle)

Note that the Coord appears selected on both right and left windows.

2. In either window, hold the shift key down and select coord[6] (the Coord attached to the background at the global origin).

Note that the two Coords are selected on both windows.

3. Click the Join/Create Constraint button on the Edit toolbar.

The Create Constraint window appears.

4. Select "Spherical Joint" as the constraint type.



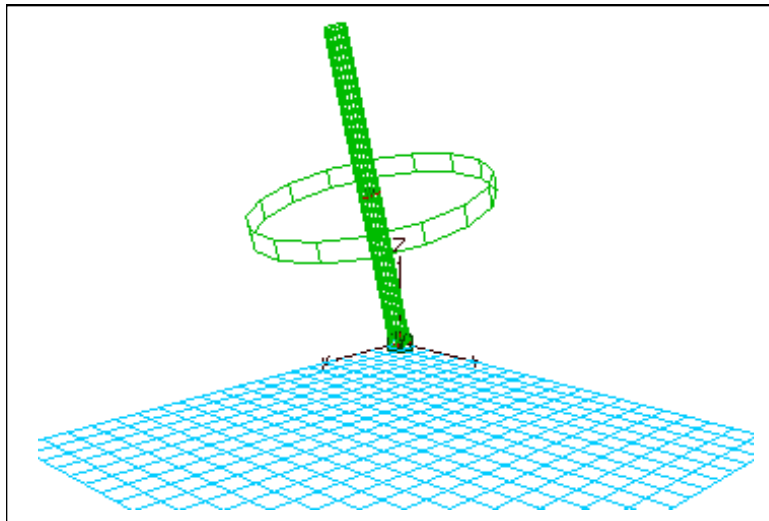
5. Choose the option labeled “Join (move body[2] to coord[6])”.

The option indicates that the body[2] (axle) will be moved to the Coord, so that the two Coords will be aligned (see below for more information).

6. Click the Create button located at the bottom of the window.

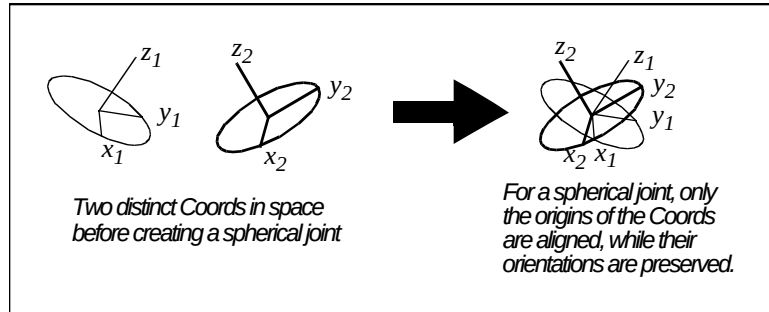
The gyroscope is moved, and the spherical joint icon appears at the attachment point (Figure 2-19).

Figure 2-19
*Gyroscope Attached
to the Ground*



When creating a spherical joint, Working Model 3D makes an effort to align the coordinate origins of the two Coords. However, since a spherical joint allows all three degrees of rotation, the orientation of each Coord remains unchanged before and after the joint is created.

In the same light, note that the entire gyroscope assembly is *translated* so that the attachment points are aligned, but it was not *rotated*. It was only necessary to align the position of the Coords, but not their orientations. Figure 2-20 illustrates this principle. Compare it with Figure 2-15 which describes the principles of the revolute joint.

Figure 2-20*Aligning Coords for Spherical Joint*

2.6 Giving Initial Spin to the Gyroscope

To observe precession, the gyroscope must have an initial angular velocity.

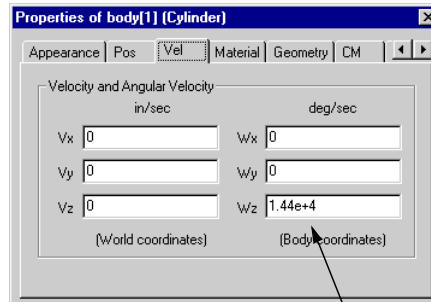
1. **Double-click the flywheel, body[1]**
1. **Click the Vel tab in the Properties window.**

The Properties window shows the initial velocity and angular velocity of the flywheel.

2. **To specify the angular velocity about the z-axis of the flywheel, enter Wz = 14400 (see Figure 2-21).**

The angular velocity of a body is specified in terms of the body's coordinate axes. Since the flywheel is to spin about its z-axis, you are specifying the angular velocity in z-axis (ω_z). In this case, the angular velocity of 14400 degrees per second corresponds to 2400 rpm.

Figure 2-21
Assigning Initial Angular Velocity



Type 14400 and press Enter. The number format is automatically converted to 1.44e+04.

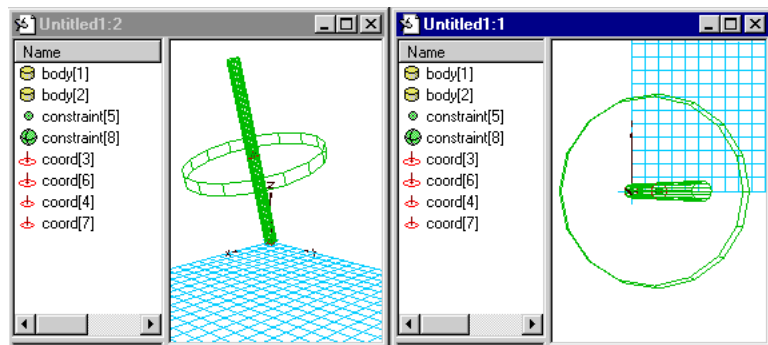
2.7 Running the Simulation

To better observe the precession (in which the axle slowly rotates about the global z-axis), you will prepare another view angle to the model.

1. Click once in the blank area of the right window.
2. Press T on the keyboard.

The window provides the “top view” of the model (Figure 2-22). The keyboard shortcut “T” stands for “top view.”

Figure 2-22
Getting the Top View



3. Click the Run button in the Tape Player control.

Run the simulation for approximately one hundred frames. Observe that the gyroscope exhibits the precession while the flywheel undergoes a rapid rotation.



4. **After running for about 100 frames, click the Reset button.**

Working Model 3D resets the simulation to the initial condition, but the simulation data is still stored in memory.



5. **Click the Run button again.**

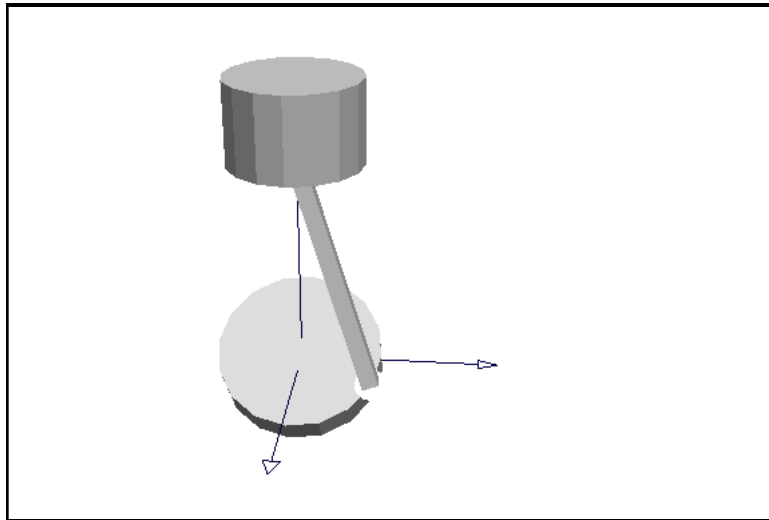
Note that the simulation seems to run much faster the second time around. Working Model 3D stored the simulation history from the previous run, and it is simply playing back the data. As soon as the tape player control's frame indicator reaches the rightmost edge, Working Model 3D will need to recompute, and the apparent speed of the simulation slows down.



6. **After running for about 100 frames, click the Reset button.**

EXERCISE 3

Piston Engine



The goals of this exercise are to show you how to create constraints and assemble bodies in Working Model 3D and how to take data from the simulation through meters. The target model is a simple piston engine model consisting of a crankshaft, a connecting rod, and a piston. The engine has a skewed crankshaft (i.e., the rotation axis is not orthogonal to the translation axis of the piston); therefore, the model can only be illustrated in a three-dimensional simulator.

In a conventional internal combustion engine, a gas explosion in the chamber pushes down the piston to drive the crankshaft. For simplicity, however, this tutorial exercise calls for a mechanism that is driven by rotating a crankshaft with a motor; you can think of the model as a rudimentary compressor, or a piston engine being started by a crank motor.

3.1 Setting Up the Workspace

You will first set up the unit system in the Working Model 3D document in order to accommodate small mechanisms such as a piston engine. The default unit system employs the SI unit system (where distance is measured in meters). You will change the distance unit to millimeters, while leaving all other units (such as time and mass) intact.

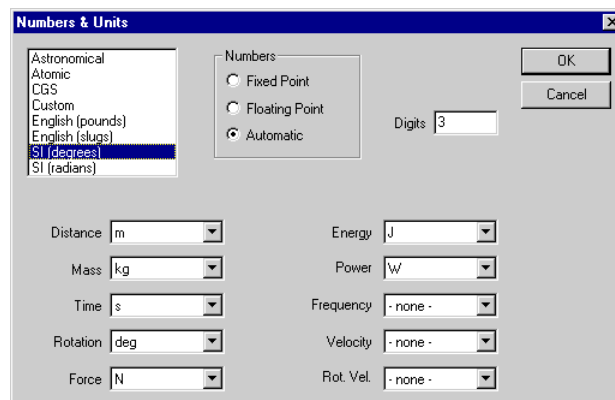
Setting Up the Units

To change the unit system:

1. Choose the Numbers & Units in the World menu.

The Numbers & Units dialog appears (Figure 3-1). By default, the SI (degrees) unit system is chosen.

Figure 3-1
*Numbers & Units Dialog
(expanded)*



2. Set the distance units to millimeters.
3. Click the OK button to close the dialog.

From there on, all distance measurements in this document will be in millimeters.

Setting Up the Edit Grid

You will modify the size of the Edit Grid and zoom rate of the document so that they are appropriate for the dimensions of the components you will create.

To change the Edit Grid size:

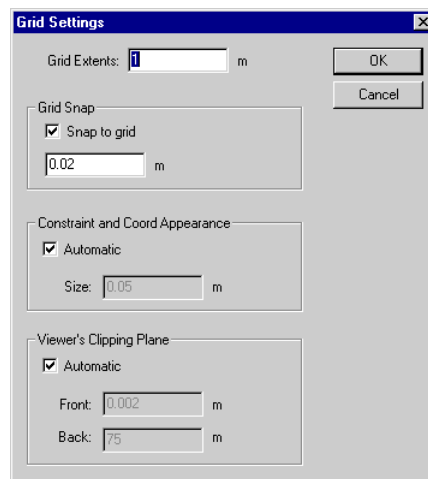
1. Choose **Show Grid** in the **Grid** menu.

The Edit Grid appears.

2. Choose **Grid Settings** in the **Grid** menu.

The Grid Settings dialog appears (Figure 3-2).

Figure 3-2
Grid Settings Dialog



3. Type 200 into Grid Extents and 10 into Grid Snap edit boxes.
4. Click OK to close the dialog.

Note that the units in the dialog are based on millimeters. Working Model 3D automatically scales the document window to accommodate the smaller Edit Grid.



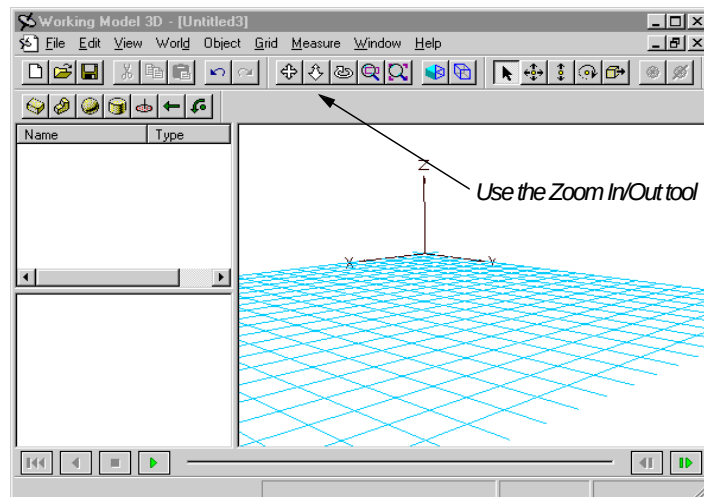
5. Click the **Zoom In/Out** tool on the **View** toolbar.

The Zoom tool allows you to zoom into the document with a particular focus. The first mouse click becomes your focus for zooming.

6. Click the global coordinate origin, hold down the mouse button, and drag the mouse downward to zoom in. Note that if you move the mouse upward, you can zoom out.
7. Release the mouse button when the Edit Grid fills a reasonable portion of your document window. You can repeat the previous step to zoom in further.

See Figure 3-3 for an example.

Figure 3-3
Document Zoomed In



If your Edit Grid does not appear to be quite centered, use the Pan tool to shift the view laterally.

3.2 Creating the Crankshaft

In this exercise, the crankshaft is simplified as a single disk, and you will later attach connecting rods from each piston to this disk so that the phase will be offset by 180°.



As in the previous gyroscope exercise, you will specify the geometry and position of the body using the Properties window.

1. Click the Cylinder tool on the Sketch toolbar.
2. Click once on the Edit Grid, then press the Enter key on your keyboard.

A small cylinder appears where you clicked.

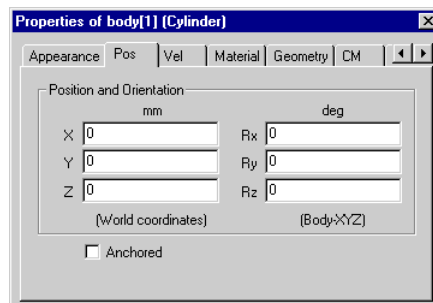
3. Double-click the cylinder in the document window or the Object Manager list.

The Properties window appears.

4. Click the Position (Pos) tab.

The Properties window displays the Position page (Figure 3-4).

Figure 3-4
Properties Window (Position Page)
for the Cylinder

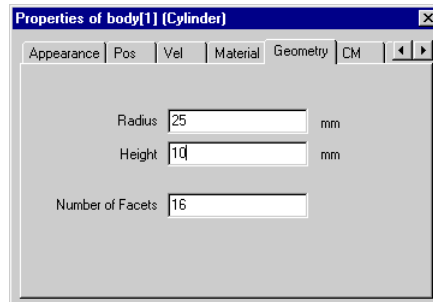


5. Enter the position as $(x, y, z) = (0, 0, 0)$,
6. Click the Geometry tab.

The Properties window displays the Geometry page (Figure 3-5).

Figure 3-5

Properties Window (Geometry Page) for the Cylinder



7. Enter radius = 25, height = 10 in the dialog.

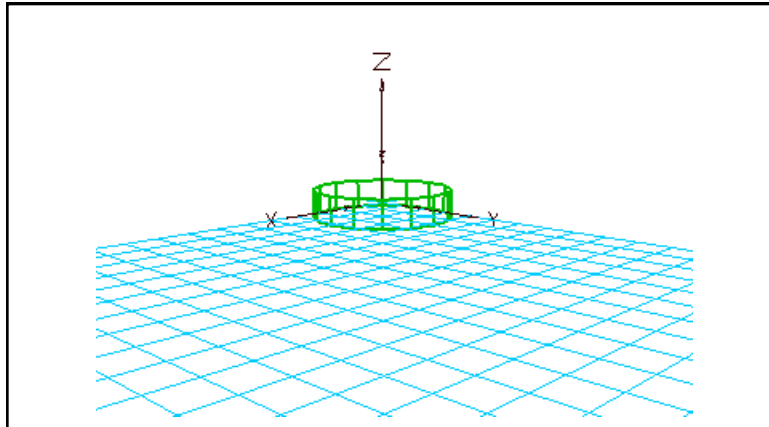
Note that all the numbers are interpreted in millimeters. You can use the tab key to move from one field to another.

8. Close the Properties window.

The cylinder is accordingly sized and positioned on the screen (see Figure 3-6).

Figure 3-6

Crankshaft Sized and Positioned



Currently, the crankshaft is oriented so that it lies flat on the xy plane. You will reorient the shaft so that it is slightly skewed.

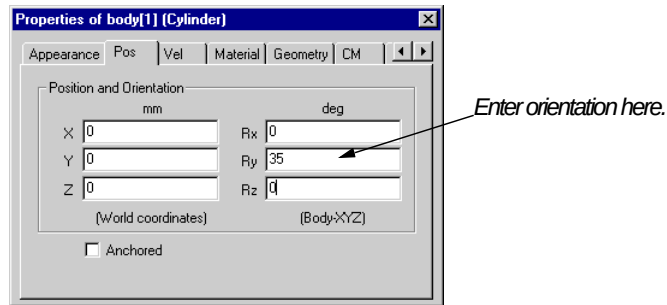
9. Double-click the crankshaft cylinder in the document window or the Object Manager list.

The Properties window reappears.

10. Click the Position (Pos) tab.

The Properties window displays the Position page (Figure 3-7).

Figure 3-7
*Properties Window (Position Page)
for the Cylinder*



11. Enter the orientation of the cylinder to (Rx, Ry, Rz) = (0, 35, 0).

*The crankshaft is reoriented as soon as you tab out of the Ry field.
You do not need to close the Properties window to have the changes
take effect.*

Assigning a Name to the Crankshaft

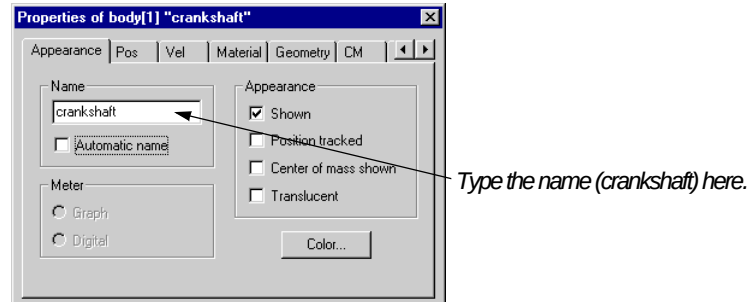
As you create bodies in a document, Working Model 3D automatically assigns a variable name (such as body[3] body[4]...) to each of them. Although not necessary for this exercise, you will name each body so that you can easily identify them later.

1. Click the Appearance tab at the top of the Properties window.

The Appearance panel appears as shown in Figure 3-8.

Figure 3-8

Properties Window (Appearance Panel)

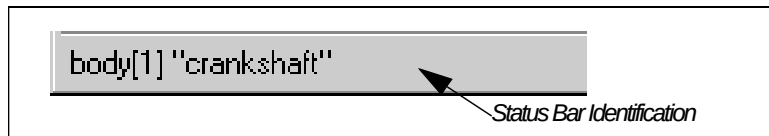


2. In the Name field of the window, type *crankshaft* in the edit box. Press Enter.

To see how the custom name helps you in identifying bodies, simply “hover” the mouse over the crankshaft cylinder. Note that the status bar at the bottom of the Working Model 3D window identifies the body with the variable name (body[n]) as well as the custom name you have just assigned. The custom name for this body also appears in the Object Manager list.

Figure 3-9

Status Bar Identifying Object



3.3 Attaching Motor to Crankshaft

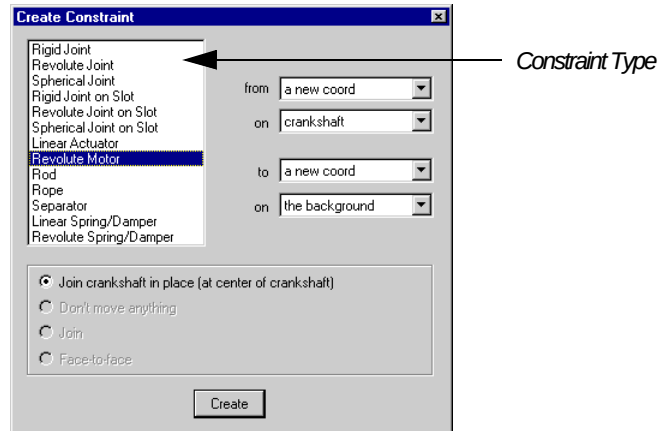
You will now fix the crankshaft to the background.

1. Select the crankshaft, if it is not already selected.
2. Click the Join/Create Constraint button on the Edit toolbar.



The Create Constraint window appears (Figure 3-10).

Figure 3-10
Create Constraint Window



3. Choose Revolute Motor in the list of constraint types.

The Create Constraint window is best understood when you read the first few lines of the window as a complete sentence. In our case, the dialog reads:

“Create a motor from a new coord on crankshaft to a new coord on the background.”

Therefore, the dialog is set up to create a motor between the crankshaft and the background.

Meanwhile, the bottom half of the window provides the only option that reads:

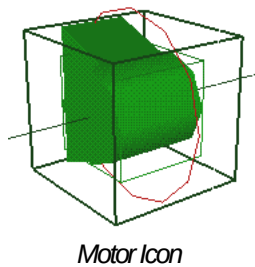
“Join crankshaft in place (at center of crankshaft)”

The option indicates that the motor will be attached to the crankshaft at its center and that the crankshaft will not move—the body will be joined “in place”.

Working Model 3D provides a range of options depending on the constraint type and objects you select before creating constraints. Please refer to the *Working Model 3D User’s Manual* for more information.

4. Click the Create button in the window.

The Create Constraint window closes.



Note that a motor icon appears at the center of the cylinder. Also, notice that:

- Motor icon has a green cube around it (when selected) with an axis that shows the plane in which the motor allows rotation.
- Working Model 3D created a red disk-shaped object called a *Coord*. The model actually has two Coords at the same global position, where one is attached to the crankshaft and the other to the background (Figure 3-11 displays only one of them, because two Coords for the motor are overlapped at this point).

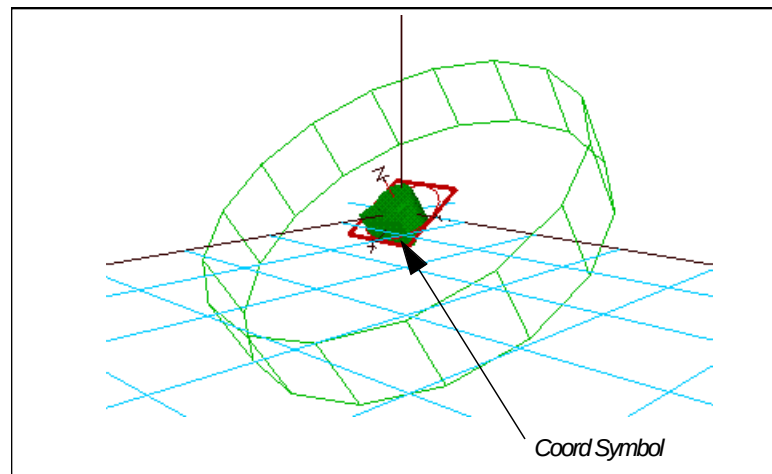
You may wish to zoom in further to gain a better view of these symbols.

What is a Coord?

Coords are basic objects to construct constraints in Working Model 3D. Physically, a Coord is a point element with a coordinate system attached. Therefore, a Coord has position *and* orientation.

To visualize a Coord, simply click the red circle around the motor icon. Notice that a set of coordinate axes appear and that the red circle is a representation of the xy-plane of the Coord. You may wish to use the Zoom tool to get a closer look.

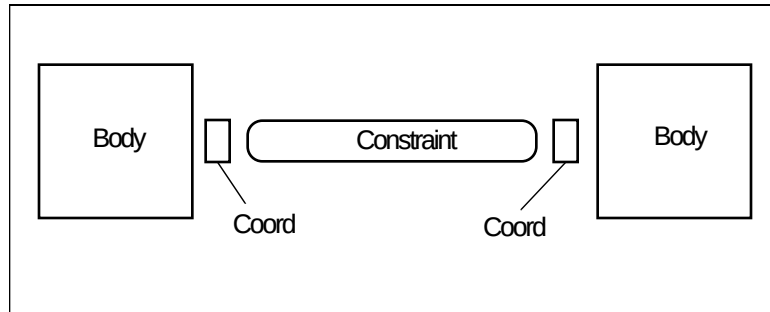
Figure 3-11
Coord and Motor



All constraints (except for external forces and torques) in Working Model 3D consist of a pair of Coords, each of which are attached to a body or the background. In essence, a Coord serves as a medium between a body and the constraint, as illustrated in Figure 3-12.

Figure 3-12

Concept: Construction of a Constraint



For constraints such as spring dampers and separators, Coords serve as endpoints to define those physical constraints. For joints, Working Model 3D uses the orientation of the pertinent Coords to determine the constraint conditions.

For example:

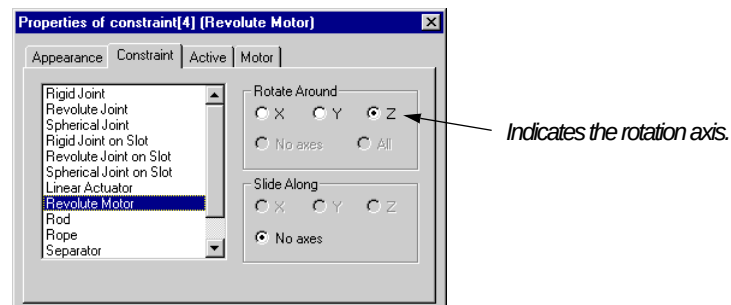
5. **Double-click the motor icon located at the center of the crankshaft or in the Object Manager list.**

The Properties window appears for the motor (Figure 3-13).

6. **Click the Constraint tab in the Properties Window.**

Figure 3-13

Properties Window (Constraint Page) for Motor



As Figure 3-13 illustrates, each constraint that limits any degree of freedom shows you the free axis in terms of rotation and/or translation. In the case of this motor, the Properties window shows that the motor allows rotation in the z-axis.

7. **Click the Motor tab in the Properties Window.**

Figure 3-14

*Properties Window (Motor Page)
for Motor*

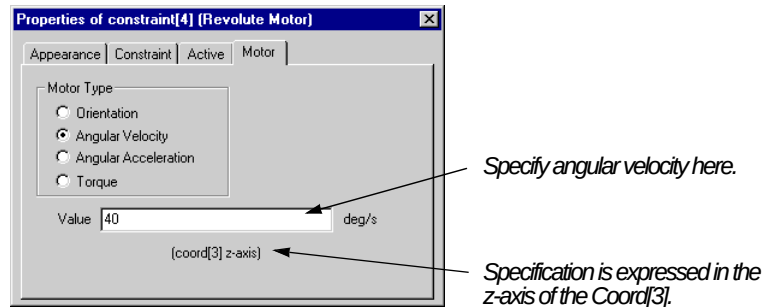


Figure 3-14 illustrates that the angular velocity specification is expressed in the z-axis of the Coord[3], one of the two (overlapping) Coords constituting the motor.

By modifying the position and the orientation of Coords, you can create a full range of constraints. For more information, please see the *Working Model 3D User's Manual*.

Specifying the Motor Function

You will specify the motor's performance characteristics through the Properties window.

1. **On the Motor page of the Properties window (Figure 3-14), select Angular Velocity, and type 40 in the value field.**

The specification calls for a velocity-controlled motor, maintained to rotate at 40 degrees per second. When you run the simulation, Working Model 3D will provide as much torque as necessary (in either direction) to maintain the specified angular velocity.



2. **Click the Run button on the Tape Player Control.**

Note that the crankshaft rotates counterclockwise (if seen from the positive z). Also, note that the frame indicator at the bottom shows the current frame being calculated.



3. **After several frames, click the Reset button.**

Before proceeding to construct the rest of the model, we will modify the motor function so that it only generates some torque at the beginning of the motion, just like a starter motor would.

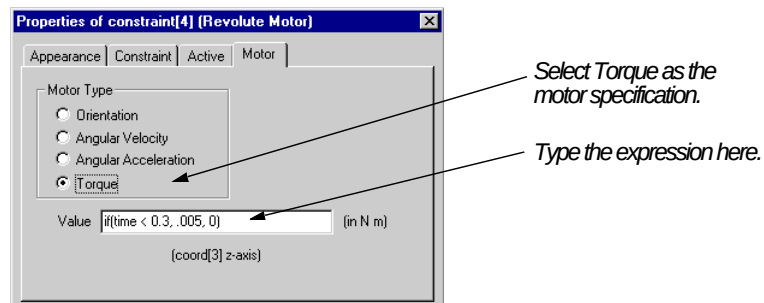
4. Double-click the motor icon.

The Properties window shows the motor characteristics (Figure 3-14).

5. Select Torque as Motor Type, and type the following expression in the Value field (Figure 3-15):

```
if(time < 0.3, .005, 0)
```

Figure 3-15
Specifying Motor Properties with Formula Language



The expression is an example of Working Model 3D Formula Language, and this expression is interpreted as follows:

If the elapsed time is less than 0.3 seconds, apply the torque of .005 N-m; otherwise, apply no torque.

Note that Working Model 3D interprets all the variables and numbers in formula expressions in SI units. Even though the distance unit in the current unit system is set to millimeters, you must enter the formula as if the distance unit is meters. In this example, you want the value of the torque to be 5 N-mm, so you must enter 0.005 N-m, the equivalent value in SI units.

For more information on the formula language, please refer to *Working Model 3D User's Manual*.

3.4 Creating the Connecting Rod

Next you will attach a connecting rod to the crankshaft. (Before doing so, you may wish to zoom out so that the bulk of the Edit Grid is visible in your document window.)

To sketch the connecting rod:



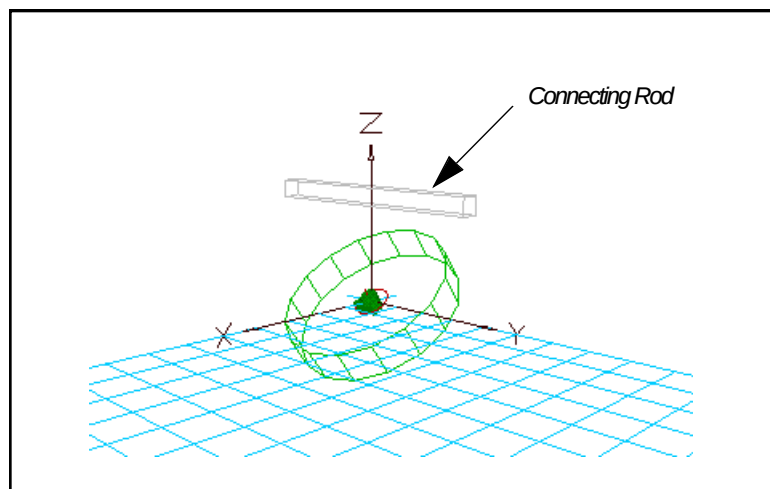
1. Click the **Box** tool in the **Sketch** toolbar.
2. Click once on the **Edit Grid**, then press the **Enter** key on the keyboard.
3. Open the **Properties** window by double-clicking the box.

The Properties window appears.

4. Click the **Position (Pos)** tab.
5. Type the position as $(x, y, z) = (0, 0, 30)$.
6. Click the **Geometry** tab.
7. Enter the dimensions as **(Width, Length, Height) = (5, 70, 5)**.

The connecting rod is sized and positioned accordingly (Figure 3-16).

Figure 3-16
Connecting Rod



As with the crankshaft, you will assign a custom name to this body as well.

8. Double-click the Connecting Rod.

The Properties window appears.

9. Click the Appearance tab.

The Properties window switches to the Appearance Page.

10. Select the text (body[5]) in the name field, and type the name `con_rod`. Press Enter.

Again, notice how the status bar identifies the connecting rod and the crankshaft as you move the mouse over them.

3.5 Attaching the Connecting Rod to Crankshaft

To attach the connecting rod to the crankshaft, you will specify the attachment point for the connecting rod on the crankshaft.

Creating Attachment Points

You will create one attachment point on the crankshaft, another attachment point on the connecting rod, and create a spherical joint based on these two attachment points.



1. Click the Coord tool on the Sketch toolbar.

2. Click anywhere on the top surface of the crankshaft; you will position the Coord precisely in the later steps.

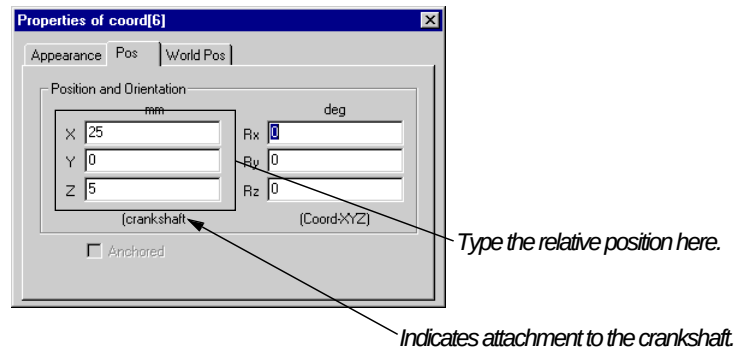
A Coord appears with a red circle and the coordinate axes.

3. Double-click the Coord.

The Properties window appears (Figure 3-17).

4. Click the Position (Pos) tab.

Figure 3-17
 Properties Window (Position Page)
 for Coord



5. In the field labeled **Position & Orientation with respect to crankshaft**, enter position at $(x, y, z) = (25, 0, 5)$.

This specification positions the Coord at the edge of the crankshaft.



6. Click the **Coord** tool on the Sketch toolbar.
7. Click anywhere on the connecting rod (you will position the Coord precisely in the later steps).
8. Double-click the Coord.

The Properties window shows the position of the Coord (similar to Figure 3-17).

9. In the field labeled **Position & Orientation with respect to con_rod**, enter position at $(x, y, z) = (0, 35, 0)$, and orientation as $(R_x, R_y, R_z) = (-90, 0, 0)$.

This specification (relative to the body) positions the Coord at the right end of the connecting rod. This time we are specifying the orientation as well as position because the Coord may have attached to the side surface of the rod and not have the correct orientation.

Creating Spherical Joint

You will attach the connecting rod to the crankshaft with a spherical joint.

1. Select the Coord on the connecting rod (if it is not already selected).
2. Holding down the shift key, and select the Coord located at the perimeter of the crankshaft.

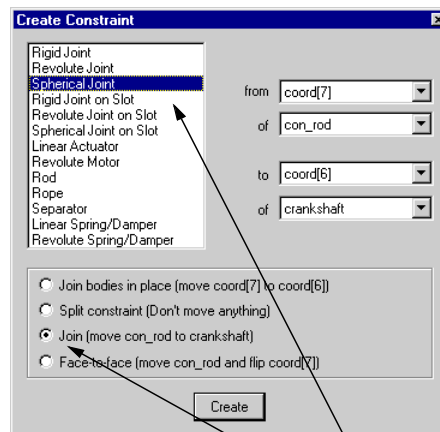
Both Coords are selected.



3. Click the Join/Create Constraint button on the Edit toolbar.

The Create Constraint window appears (Figure 3-18). Note that the body names appear as previously customized in the window, so that you can easily verify the bodies and Coords you are attaching are correct.

Figure 3-18
Create Constraint Window



Choose the spherical joint, with the Join option.

4. Choose Spherical Joint as the constraint type.

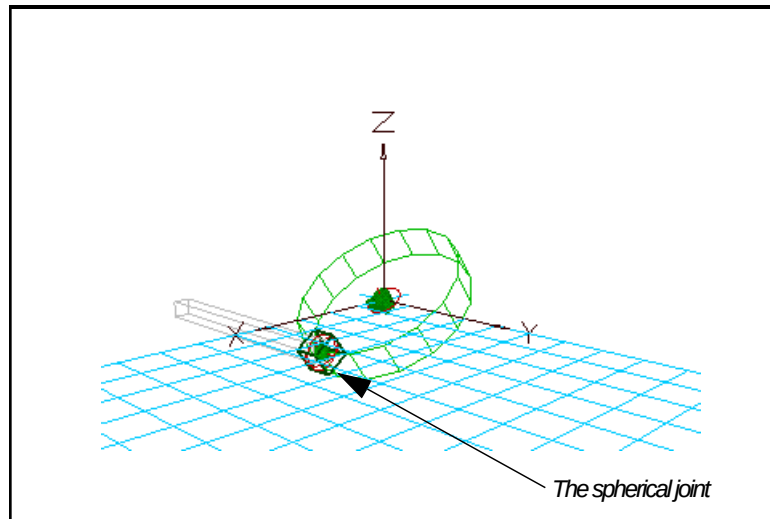
As discussed in “3.3 Attaching Motor to Crankshaft”, read the first few lines of the window as a complete sentence: “Create a spherical joint from coord[7] of con_rod to coord[6] of crankshaft.” Working Model 3D automatically selects the remaining options depending on the objects you selected.

5. Click the option “Join (move con_rod to crankshaft)” (see Figure 3-18).

6. Click the Create button in the window.

The Create Constraint window closes. The connecting rod is moved to the crankshaft, and a spherical joint icon appears at the attachment point (see Figure 3-19).

Figure 3-19
Bodies Connected with Spherical Joint



A green cube appears with three axes around the spherical joint when it is selected. For constraints that limit any rotational degree of freedom, these octagons appear to show the planes on which the rotations are allowed. Since a spherical joint allows all three rotational degrees of freedom, the joint symbol shows three axes.

3.6 Testing Your Model

When you create a simulation model, you should check from time to time to make sure that the model's components are in working order. If you are creating a complex model and put off testing it until the very end, finding a solution may be extremely difficult when your model yields erroneous simulation results.

So far, you have completed the following:

- Attached a crankshaft to the background with a motor in a skewed angle, and
- Attached a connecting rod to an edge of the crankshaft with a spherical joint.

To verify the result, you will run the simulation with a specific motor speed. You can easily expect that the connecting rod will swing out as the crankshaft rotates.

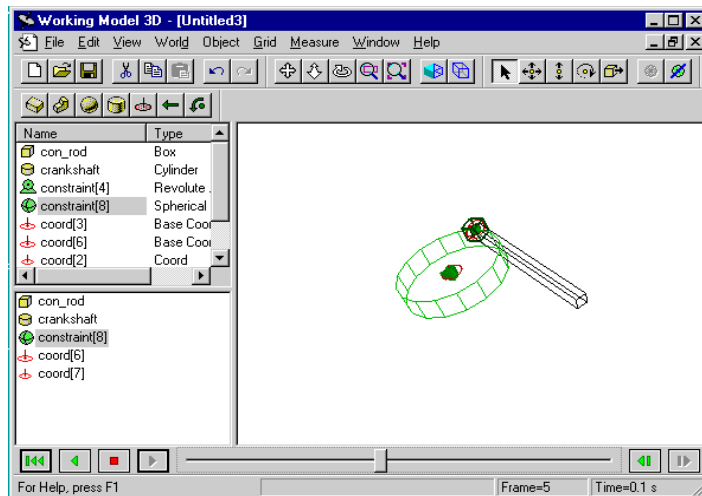
To start testing, simply run the simulation.



1. Click the Run button on the Tape Player Control.

The simulation starts, while the frame indicator shows you the number of frames calculated. Note how the connecting rod swings out as the crankshaft rotates.

Figure 3-20
*ConnectingRodSwungOutbythe
Crankshaft*



2. After running for a dozen frames or so, click the Reset button on the toolbar.

If the model did not behave as discussed, you should review your model. A few hints are:

- If the motor is not rotating, you may have forgotten to specify the motor's property. Review "3.3 Attaching Motor to Crankshaft".

- If the connecting rod is falling downward, you probably have failed to attach it to the crankshaft. Review “3.5 Attaching the Connecting Rod to Crankshaft”.

“Debugging” a large model is not always an easy task. As discussed in this section, you should take a moment every once in a while to verify your model assembly.

3.7 Creating the Piston

You will now create a piston and fix it in space with a slot joint.



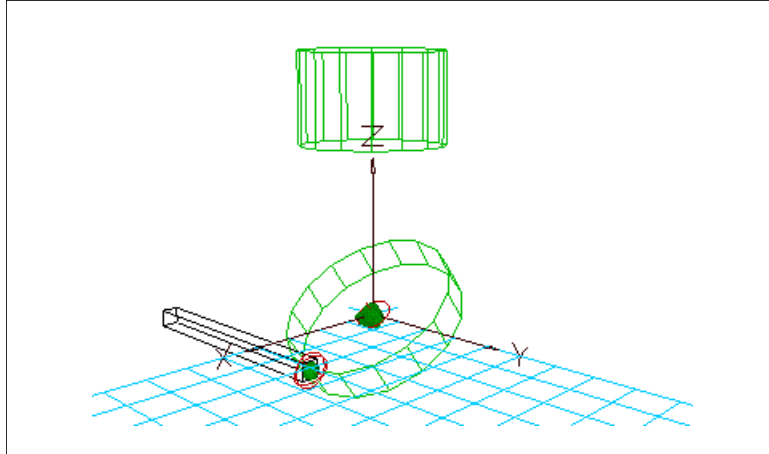
1. Click the cylinder tool on the Sketch toolbar.
2. Click once on the Edit Grid, then press the Enter key on the keyboard.
3. Double-click the cylinder in the document window or the Object Manager list.

The Properties window appears (Figure 3-5).

4. Click the Position (Pos) tab, then enter the position as (x, y, z) = (0, 0, 60).
5. Click the Geometry tab, then enter the geometry as (radius, height) = (20, 25).

The cylinder is accordingly sized and positioned on the screen (see Figure 3-21).

Figure 3-21
Piston Sized and Positioned



If the cylinder appears cut off at the top of your document window, click the Pan tool to shift the view.

To assign a custom name to the piston:

6. Double-click the piston to open the Properties window.
7. Click the Appearance tab.
8. Click once in the name field, and type *piston* in the edit box. Press Enter.

You will now proceed to attach the piston to the background with a slot joint.

Attaching Piston to the Background

The piston's motion is restricted to the vertical axis. You will attach the piston to the background using a rigid joint on slot.

1. Select the piston, if it is not already selected.
2. Click the Join/Create Constraint button on the Edit toolbar.

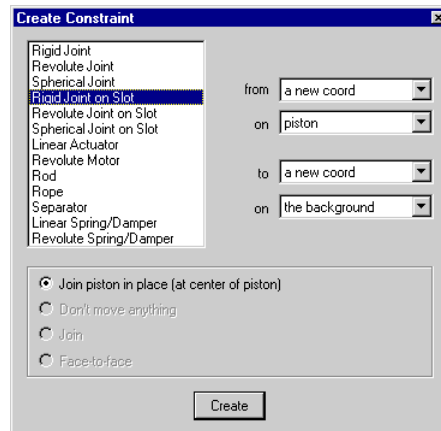
The Create Constraint dialog appears.

3. Choose "Rigid Joint on Slot" as the constraint type.



The window presents a single option: Join in place.

Figure 3-22
Create Constraint Window

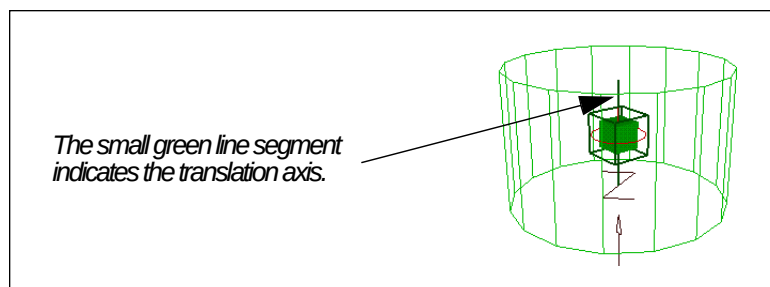


Since you selected a single body without Coords, Working Model 3D automatically presents the only reasonable option. The option means that Working Model 3D will create two Coords (one attached to the piston, the other attached to the background), and create a rigid joint on slot to connect the body to the background. One of the Coords will be attached to the body at its frame origin (coordinate origin).

4. Click the Create button in the window.

A rigid joint on slot icon appears at the center of the piston (Figure 3-23). You may want to zoom in with the zoom tool to obtain a better view.

Figure 3-23
Rigid Joint on Slot Created



Note that the small green line segment appears to indicate that this slot joint allows single degree of freedom in translation. Therefore, as it stands, the piston is allowed to slide “up-and-down” like we would like it to.

3.8 Attaching Connecting Rod to Piston

You will attach the connecting rod to the piston with a spherical joint, just as you attached the rod to the crankshaft.

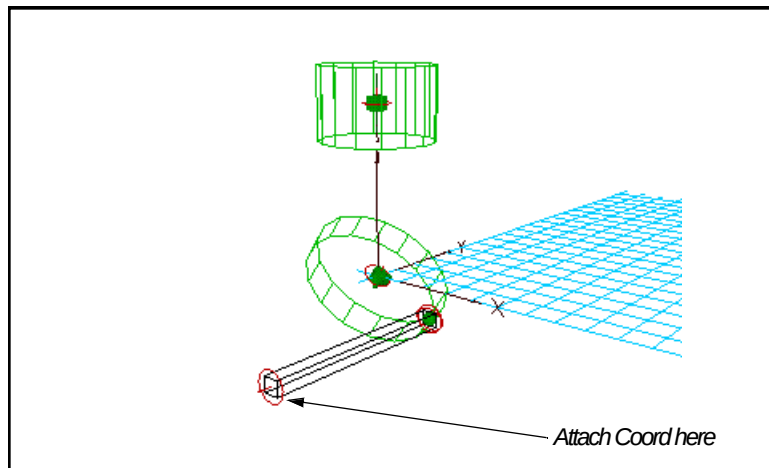
Creating Attachment Point on Connecting Rod

To create a pair of attachment points:



1. Use the Rotate Around tool so that you can see the free end of the connecting rod (see Figure 3-24).

Figure 3-24
*Attaching Coord
to the Connecting Rod*



2. Click the Coord tool on the Sketch toolbar.
3. Click at the free end of the connecting rod.

If you attached the Coord on a side surface of the connecting rod, do not be alarmed; you will correct it in the next step.

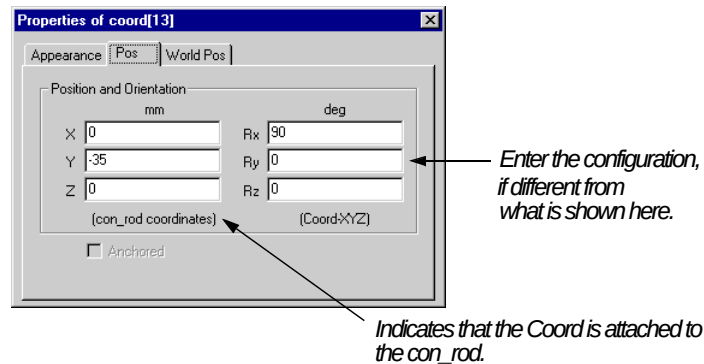
4. Double-click the Coord.

The Properties window appears.

5. Click the Position (Pos) tab, and verify that the Coord is attached to the connecting rod (see Figure 3-25), and the configuration is given as (X, Y, Z) = (0, -35, 0), and (Rx, Ry, Rz) = (90, 0, 0). If any values are different, correct them.

Figure 3-25

Properties for Coord Attached to Connecting Rod



Creating Attachment Point on Piston

To connect the connecting rod with the piston, you will also create an attachment point on the piston.



1. Use the Rotate Around tool so that the bottom surface of the piston is facing you.
2. Click the Coord tool on the Sketch toolbar.
3. Click at the bottom surface of the piston, near the center.

If you attached the Coord on a side surface of the piston, do not be alarmed; you will correct it in the next step.

4. Double-click the Coord.

The Properties window appears.

- Click the Position (Pos) tab, and verify that the Coord is attached to the piston, and the configuration is given as (X, Y, Z) = (0, 0, -12.5), and (Rx, Ry, Rz) = (0, 180, 0). If any values are different, correct them.

Creating Spherical Joint

To put together the two Coords, each attached to the connecting rod and the piston:

- Select the Coord attached to the connecting rod.
- Hold down the shift key, and select the Coord attached to the piston.

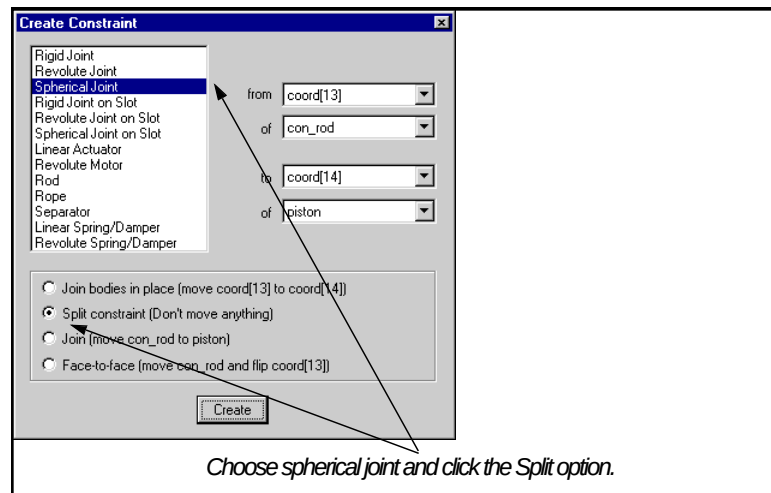
If you have trouble selecting these Coords in the document window, try selecting them in the Object Manager list.



- Click the Join/Create Constraint button on the Edit toolbar.

The Create Constraint dialog appears (Figure 3-26).

Figure 3-26
Create Constraint Window



- Select the “Spherical Joint” as the constraint type, and click the option labeled “Split constraint (Don’t move anything)”.

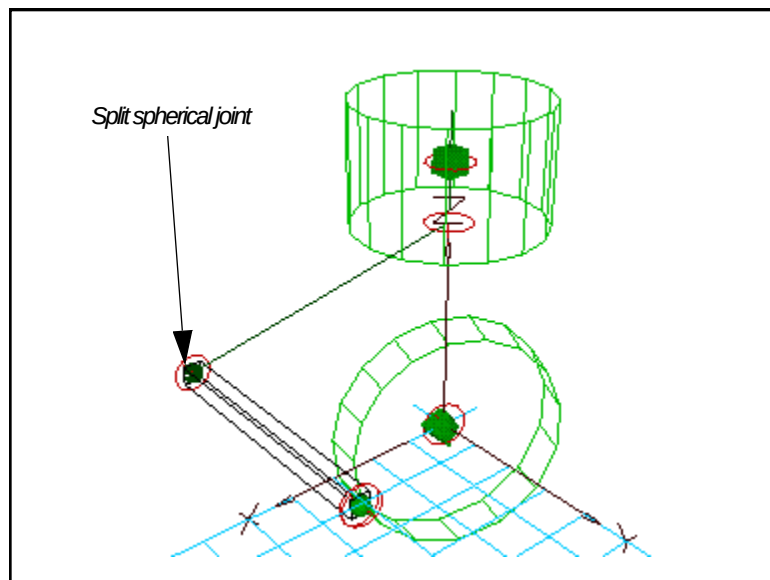
You can assign the constraint and leave the bodies intact this way. You will assemble the joint in a later step.

5. Click the Create button in the window.

A line appears between the two Coords, and a split spherical joint icon (Figure 3-27) appears in the Object Manager list. At this point, the joint is assigned, but the bodies are not assembled.



Figure 3-27
Split Spherical Joint



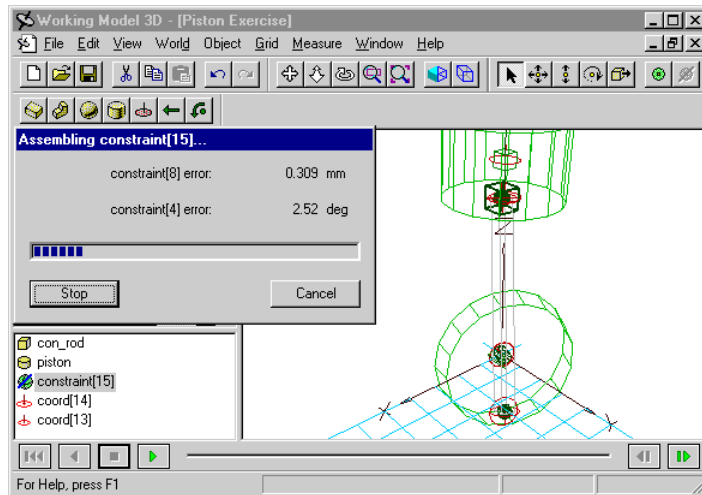
6. Click the Join/Create Constraint button on the Edit toolbar.

A progress dialog appears to show the assembly is in process (Figure 3-28). At the same time, the bodies in the model undergo motion to show you the assembly process.

NOTE: The progress dialog appears only during the assembly process for a closed-loop mechanism. If you do not see the progress dialog, you may have accidentally split another constraint by mistake, and the mechanism is still considered an open-loop. In that case, choose **Select All** in the **Edit** menu, and click the Join button on the toolbar. Working Model 3D will attempt to resolve all constraints simultaneously.

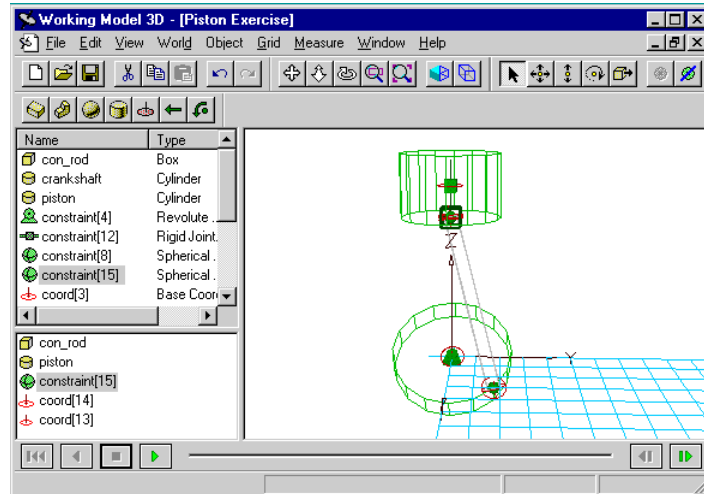
In the Join process, Working Model 3D is solving the configuration to find the converging solution to satisfy all the constraints. Bodies may appear to move away from their respective constraints at first, but Working Model 3D soon finds a converging solution that satisfies all the constraint conditions.

Figure 3-28
Joining Process



After a while, the progress bar reaches the completion, and the progress dialog disappears. At this point the model is fully assembled (Figure 3-29) and you are ready to run the simulation.

Figure 3-29
Fully Assembled Piston Engine



3.9 Running the Simulation

To run the simulation:



1. Click the Run button on the Tape Player Control.

The simulation starts, showing the motion of the piston engine mechanism. Note that the Tape Player Control located near the bottom of the application window shows the current frame number.



2. Let it run for 40 frames or so, and click the Reset button on the toolbar.

Use the Pan, Zoom, and Rotate Camera tools to obtain different view angles and repeat the simulation. Note that the simulation runs much faster the second time around, because Working Model 3D stores the simulation time history after the first calculations. The history remains in memory until you change the initial condition.

3.10 Taking Measurements

The simulation model you have just created serves as a powerful visualization tool to observe the general motion of your mechanism. Working Model 3D can take you further by allowing you to take numerical measurements on various aspects of your simulation.

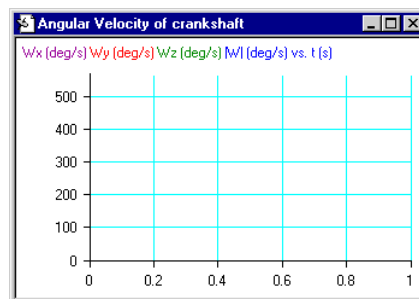
In this exercise, you will create a meter to measure the angular velocity of the crankshaft cylinder as well as the forces experienced by the joint between the piston and the connecting rod.

To create a meter to measure the angular velocity of the crankshaft:

1. **Select the crankshaft cylinder in the document window or the Object Manager list.**
2. **Choose Angular Velocity in the Measure menu.**

A meter with the title “Angular Velocity of Crankshaft” appears as a separate window (Figure 3-30).

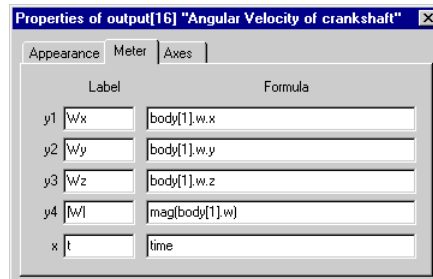
Figure 3-30
Angular Velocity Meter



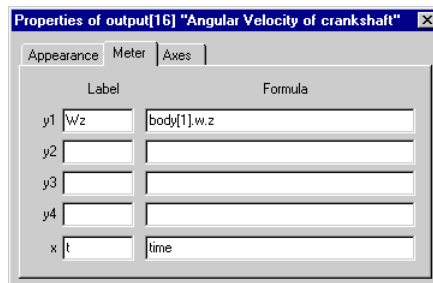
Note that when the meter window is active, the toolbar icons for tools which do not apply to meters are disabled (appear grayed out).

By default, the angular velocity meter shows plots for angular velocities in x-, y-, and z-axes as well as the magnitude. In this simulation, the crankshaft rotates with its z-axis fixed, so you are only interested in observing ω_z , or the z-component of the angular velocity.

3. **Double-click the meter to open the Properties window (see Figure 3-31).**
4. **Click the Meter tab if not already selected.**

Figure 3-31*Properties Window for Meter***5. Delete Wx, Wy, and |W| from the Label column**

As you remove items from the Label column, the formula is also deleted (Figure 3-32).

Figure 3-32*Properties Window for Meter after Unwanted Plots Removed*

You will now create another meter to measure the forces experienced by the spherical joint between the piston and the rod.

6. Select the spherical joint between the connecting rod and the piston by clicking the constraint icon.

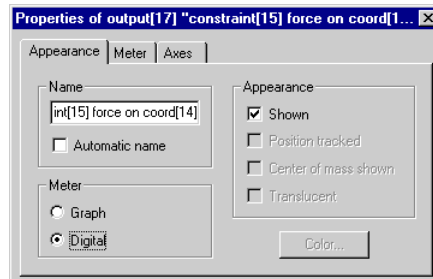
When a spherical joint is selected, it shows a green cube with short lines to show the allowable rotations.

7. Choose Constraint Force on Base Point in the Measure menu.

Another meter window appears.

8. Double-click the second meter window and then the Appearance tab in the Properties window.

Figure 3-33
Properties Window (Appearance Page) for Meter



9. Choose Digital.

The force meter switches to the digital format.

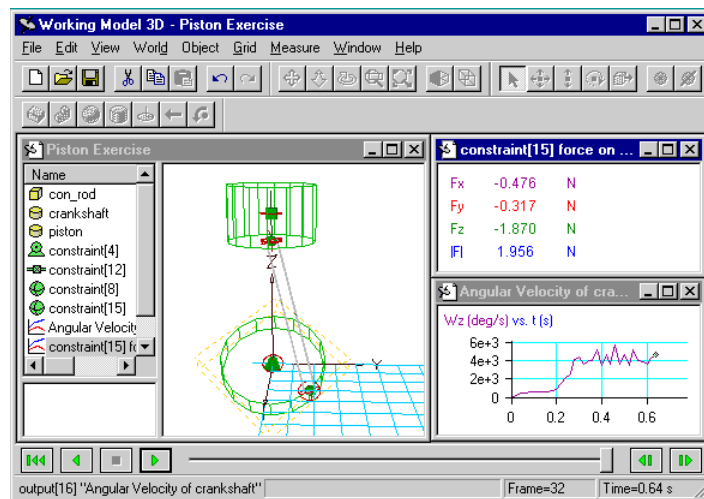
You are now ready to run the simulation to take measurements. To run the simulation:



10. Click the Run button in the Tape Player Control.

The simulation begins to show the model in motion with the meters showing the data. Note that the angular velocity meter rescales the graph axes automatically to growing magnitude of the data.

Figure 3-34
Simulation with Meters





11. Run the simulation until several rotation cycles are completed, and click the Reset button on the toolbar.

12. Repeat the simulation.

Note that the simulation runs faster during the playback; observe the “jerky” rotation of the crankshaft due to gravity.

3.11 Improving the Model

As the mechanism completes the cycle, note that the spherical joint shows a huge spike, especially when the crankshaft reaches the lowest point. In addition, the angular velocity of the crank motor (especially about the z-axis) varies rather dramatically.

To visualize the spike in the joint reaction force:

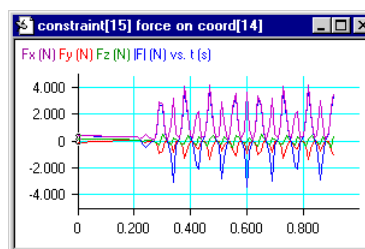
1. Click once inside the force meter.

The main menu changes to show the Meter menu.

2. Choose Graph in the Meter menu.

The digital meter changes to the graph format to show the history. Note the series of spikes seen in the force meter.

Figure 3-35
Force Meter and History



To avoid excessive wear and tear on the components, suppose you would like to put a simple control system on the crank motor to smooth the rotation. (Although this scenario appears rather impractical in real situations, this exercise is merely designed to provide examples in Working Model 3D.)

1. Double-click the motor icon located at the center of the crankshaft.

The Properties window appears to show the motor specification.

2. In the Value field, type the following expression. Expand the window if necessary, so that the entire expression is visible.

$0.2 * (360 - \text{body}[1].w.z)$

The expression implements a rudimentary proportional control system which monitors the angular velocity of the crankshaft (represented as `body[1]` in the formula). Note that, according to this expression, the motor is capable of applying torque in either direction.



3. Click the Run button in the toolbar.

Note that the force curve on the spherical joint is much smoother.

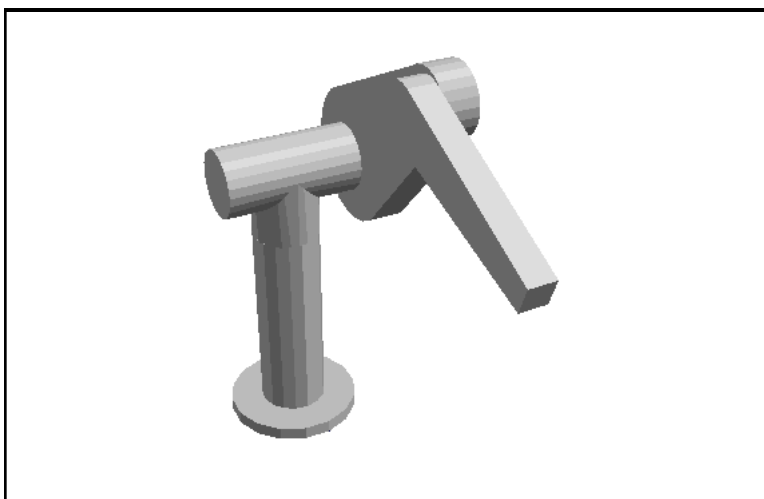


4. Run the simulation until several rotation cycles are completed, and click the Reset button on the toolbar.

You can export the meter data to a text file for further analysis by another application program. For more information, please see the *Working Model 3D User's Manual*.

EXERCISE 4

Robot Manipulator Arm



In this exercise, you will construct a robot manipulator arm by incorporating parts created in a CAD system. The exercise presents a scenario where you have already designed mechanical parts in a CAD program and have the data available as ACIS files. You will learn how to incorporate the existing CAD geometry through the ACIS conversion tool available with Working Model 3D.

The exercise allows you to construct the base unit and the arms of the manipulator arm robot. For simplicity, you will not implement the gripper in this model.

The files needed to complete this exercise are located in the “Program Files\Working Model 3D\Samples\Tutorials and Documentation\Robot Files” directory. The three files needed are “Shoulder.SAT”,

“Midarm.SAT”, and “Forearm.SAT”. If you do not find the directory or the files, use the Working Model 3D setup program to reinstall the sample files from the Working Model CD-ROM.

This exercise also requires the ACIS Converter, which was automatically installed on your hard disk by the Working Model 3D setup program.

4.1 Importing ACIS Files

You will start this exercise by importing CAD geometry in the form of SAT files (ACIS format). The three component parts of the robot—the shoulder, the midarm and the forearm—are stored as SAT files.

Converting ACIS Files to WM3 Format

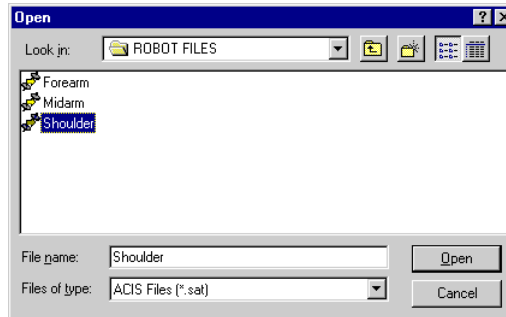
To incorporate the ACIS files to your Working Model 3D document, you will first convert the existing ACIS files (.SAT files) to the Working Model 3D format (.WM3 files) using the ACIS Converter.

NOTE: On Windows NT 3.5x, the program icon is located in the program group titled “Working Model 3D”. On Windows 95 and Windows NT 4.0, you can find the ACIS Converter in the Start menu. The executable ACIS2WM.EXE is located in the directory SAT2WIM.

1. Start the ACIS Converter.

As soon as ACIS Converter is launched, it presents the File Open dialog (Figure 4-1).

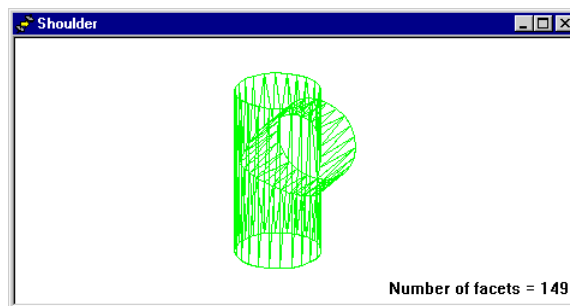
Figure 4-1
File Open Dialog of ACIS Converter



2. Find the file “Shoulder.SAT” in the “Program Files\Working Model 3D\Samples\Tutorials and Documentation\Robot Files” directory.
3. Click the Open button.

The geometry of the file appears in the ACIS Converter’s document window (Figure 4-2).

Figure 4-2
The File SHOULDER.SAT as it appears in the 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.

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 polygons. Figure 4-2 shows the result of the faceting process as it appears in Working Model 3D when the file is imported. Note that the window also shows how many facets are used to approximate the model.

NOTE: ACIS Converter only allows you to open, view, and export the ACIS files. To edit the geometry, you need to use CAD software.

4. Choose Export in the File menu.

The File dialog appears.

5. Save the file as “Shoulder.WM3” in the “Program Files\Working Model 3D\Samples\Tutorials and Documentation\Robot Files” directory.

You will later use this file to construct the robot arm mechanism.

6. To close the file “Shoulder.SAT”, choose Close in the File menu.

7. Repeat the above export steps for the files “Midarm.SAT” and “Forearm.SAT”.

When you are done, you will have the files “Shoulder.WM3”, “Midarm.WM3” and “Forearm.WM3” in the “Program Files\Working Model 3D\Samples\Tutorials and Documentation\Robot Files” directory.

8. To close the ACIS Converter, choose Exit in the File menu.

You have finished converting the ACIS files to Working Model 3D format. The remaining portion of the exercise will take place in the Working Model 3D application.

9. Launch Working Model 3D.

A blank document appears.

4.2 Setting Up the Workspace

Unit System and Importing Files

Working Model 3D files generated by ACIS Converter do not have unit information. Therefore, when you import these files into an existing Working Model 3D document, the current unit system is applied to the imported file. For example, if the original ACIS document had a mechanism that is designed to be 52.4 meters long, and if the current Working Model 3D document has the distance unit set to inches, the component becomes 52.4 inches long when imported into Working Model 3D.

On the other hand, WM3 files created by Working Model 3D contain complete unit information.

In this exercise, the original ACIS files are designed with millimeter units. Since the default unit system of a new document employs the SI unit system (where distance is measured in meters), you will change the distance unit to millimeters to accommodate the robot part files.

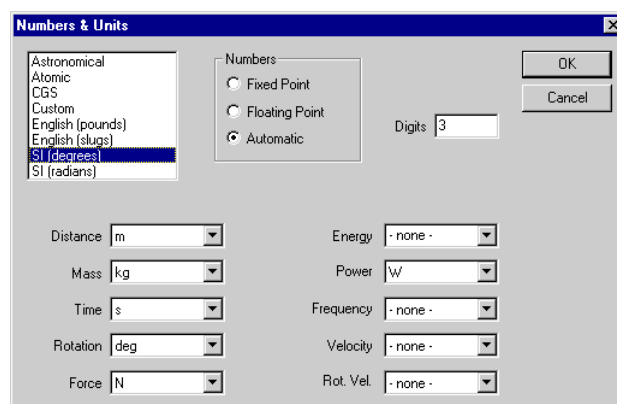
Setting Up the Units

To change the unit system:

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

The Numbers & Units dialog appears. By default, the SI (degrees) unit system is chosen (see Figure 4-3).

Figure 4-3
Numbers & Units Dialog



2. Set the distance units to millimeters (mm).
3. Click the OK button to close the dialog.

From now on, all distance measurements in this document will be in millimeters.

4.3 Importing WM3 Files

You will now import the files created by the ACIS Converter.

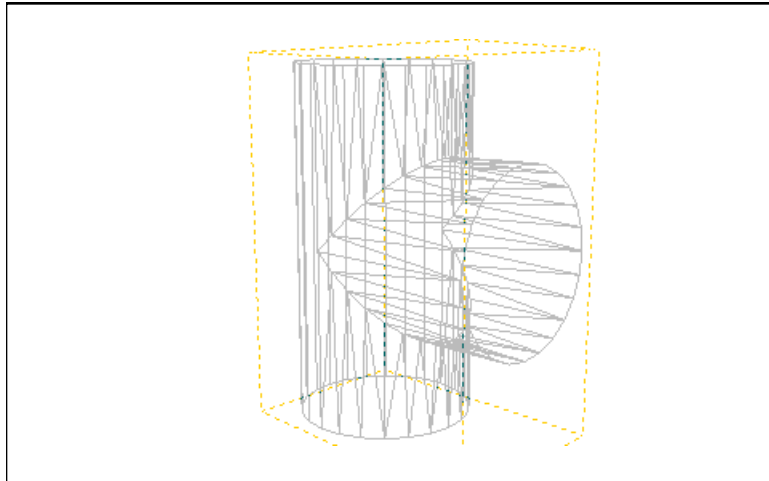
1. Choose Import in the File menu.

The Import dialog appears.

2. Find the file “Shoulder.WM3” in the “Program Files\Working Model 3D\Samples\Tutorials and Documentation\Robot Files” directory.
3. Click OK to import the file.

Working Model 3D automatically scales the view so that the imported part fits in the window (Figure 4-4).

Figure 4-4
SHOULDER.WM3 imported

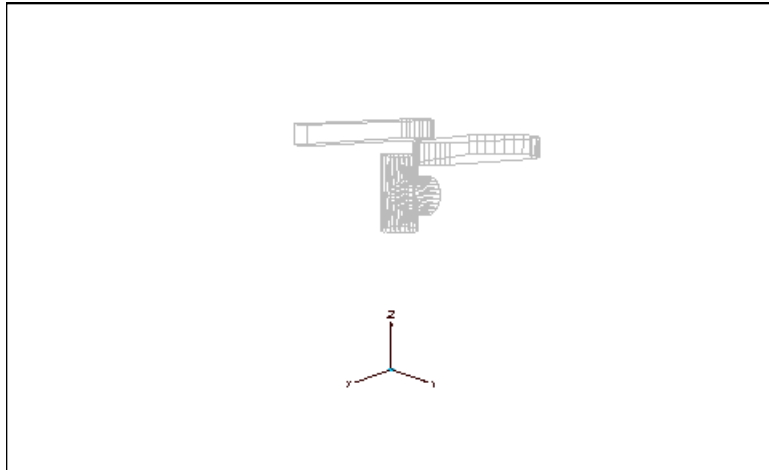


The shoulder part is positioned away from the coordinate origin.

4. Repeat the file import steps for the files “Midarm.WM3” and “Forearm.WM3”.

Note that Working Model 3D adaptively scales the window so that all parts are visible, while preserving the view angle (Figure 4-5).

Figure 4-5
All Parts Imported



Naming the Parts

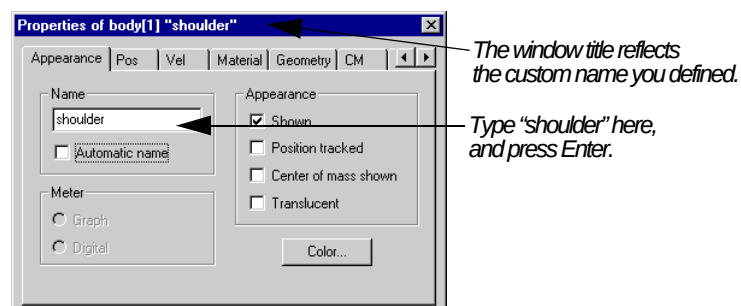
To facilitate the modeling effort, we will assign names to each part.

1. Double-click the shoulder part.

The Properties window appears.

2. If necessary, click the Appearance tab, as shown in Figure 4-6).

Figure 4-6
Properties Window (Appearance Panel)



3. In the Name field, overwrite the existing name (e.g., “body[1]”) and type “shoulder”. Press Enter.

Note that the window title (at the top of the Properties window) reflects the new name.

4. In the same fashion, name the midarm part as “midarm”.

Recall that the midarm part appears larger than the forearm.

5. Similarly, name the forearm part as “forearm.”

4.4 Arranging the View and Parts

As you construct the model, you will be using various tools to pan, zoom, and rotate the view presented in the window. While the tools provide tremendously flexible control, you could accidentally zoom too much or pan far away from the model, and you may find yourself “lost”.

To prevent you from losing sight of the model, Working Model 3D allows you to set a “favorite” view angle and lets you go back to it whenever you wish.

Setting the Home View

You will first specify the view angle that works best for you. Your goal is to obtain a view angle that resembles Figure 4-7.



1. Click the Rotate Around tool on the View toolbar.

Alternatively, you can use the F4 key to select the tool.

2. Click anywhere on the shoulder, hold the mouse button, and drag the mouse slightly downward.

Note that you can obtain a better view of the top surfaces of the parts.

3. Release the mouse button.

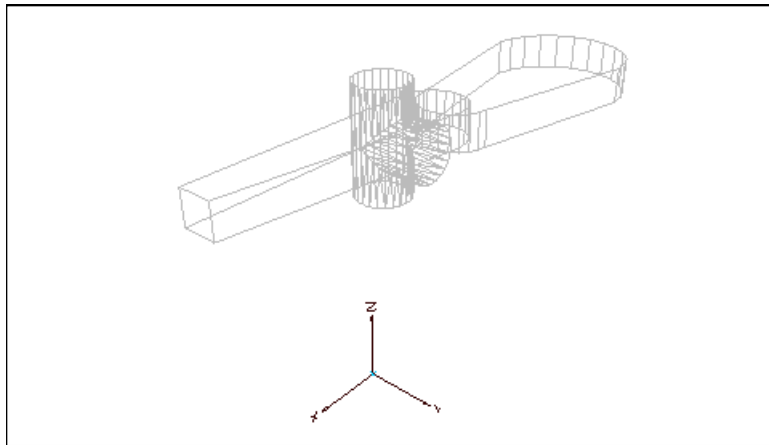


4. Click the Zoom In/Out tool on the View toolbar.

Alternatively, you can press the F3 key to select the tool.

5. Click anywhere on the shoulder, hold the mouse button, and drag the mouse slightly downward. When the document looks like Figure 4-7, release the mouse button.
6. If necessary, use the Rotate Camera tool and Zoom tool again. Repeat the steps until the document looks like Figure 4-7.

Figure 4-7
The Home View



7. Choose Set Home in the View menu.

The command tells Working Model 3D to “remember” the current view as your home view.

8. To go back to the home view, simply choose Go Home in the View menu, or press H on the keyboard.

To experiment with the Go Home feature, try changing the view angle using Zoom In/Out, Pan, and Rotate Around tools. You could pan so far that none of the model is visible in the document window, then press H. You will immediately retrieve the preset view angle.

Rotating and Positioning Parts

When ACIS Converter converted and exported all the part files, it preserved the original position and orientation information. For your model, the shoulder part had been designed and saved as it appears in

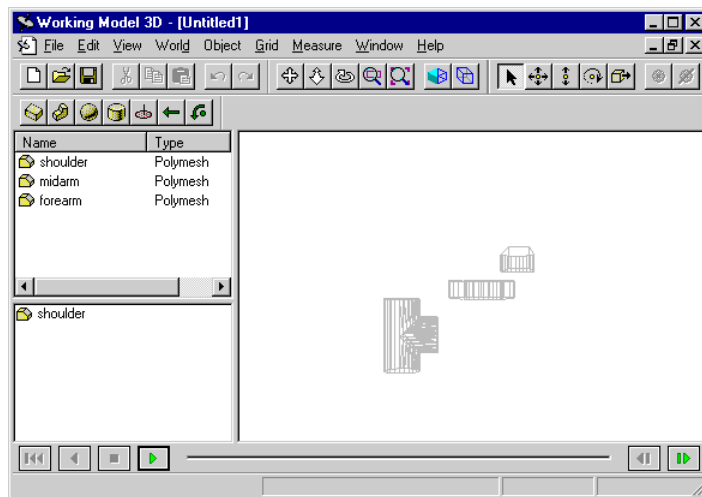
Working Model 3D, but we need to rotate it to construct the manipulator arm. You will use the Rotate tool in the Edit toolbar to correct the orientation.

First of all, you will change the viewing direction so that you can look at the shoulder from head on.

1. **Press F on your keyboard.**

The command provides the frontal view of the model (Figure 4-8).

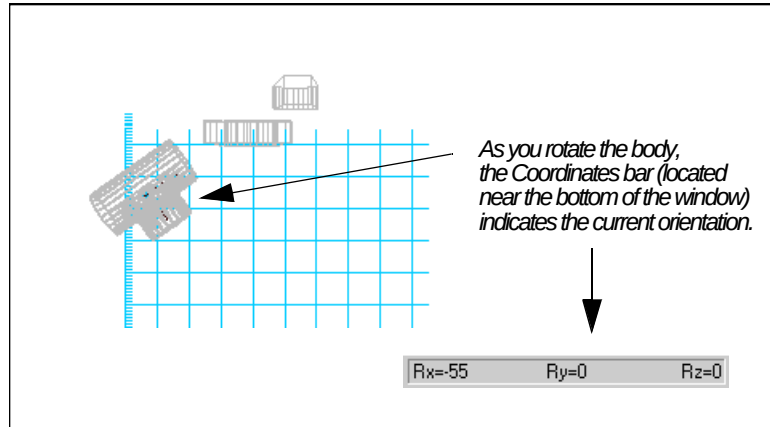
Figure 4-8
Frontal View



2. **Click the Rotate tool on the toolbar.**
3. **Click on the shoulder, and drag the mouse as if you are rotating the shoulder in the clockwise direction (Figure 4-9).**

As you rotate the body, the coordinates bar (located near the bottom of the application window) indicates the current orientation of the body. See Figure 4-9.

Figure 4-9
Rotating Shoulder



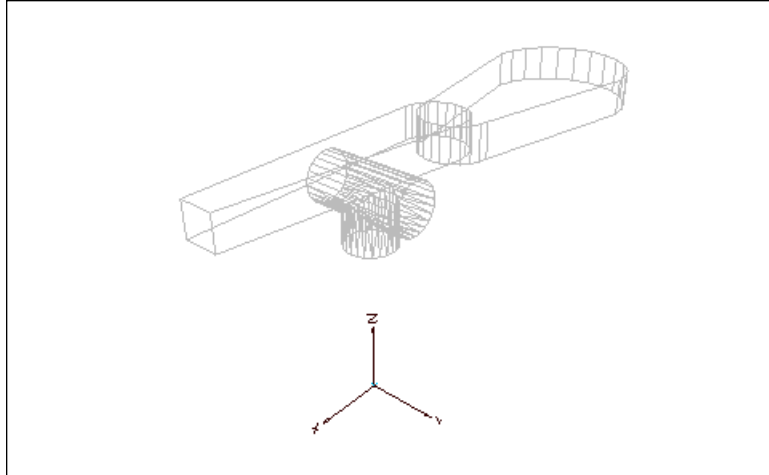
4. Rotate the shoulder so that Rx = -90, and release the mouse button.

5. Double-click on the shoulder.

The Properties window appears.

6. If necessary, click the Position (Pos) tab, then enter 0 as the X position, 0 as the Y position, and 1200 as the Z position.
7. Click any blank area of the document window (so that the next keystroke applies to the window), and press H on the keyboard.

The Home view is retrieved (Figure 4-10). Note that the shoulder part is rotated -90° compared to Figure 4-7.

Figure 4-10*The Home View Revisited*

4.5 Creating the Stand

The three parts imported so far are only the superstructure of the manipulator. You will build the platform to mount the arm assembly.

The stand is a simple cylindrical body.



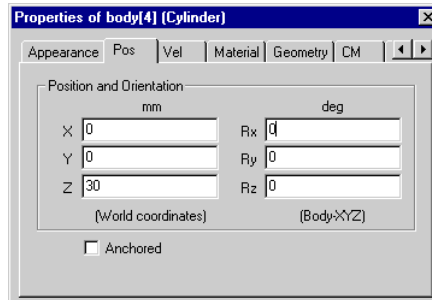
1. **Click the cylinder tool on the toolbar.**

The Edit Grid appears.

2. **Click anywhere on the Edit Grid, then press the Enter key on the keyboard. Open the Properties window by double-clicking the box.**

The Properties window displays the position of the cylinder (Figure 4-11).

Figure 4-11
*Properties Window (Position Page)
for the Cylinder*



3. Enter the position as $(x, y, z) = (0, 0, 30)$.
4. Click the Geometry tab, then enter the radius = 300 and height = 60. (Figure 4-12).

Note that all the numbers are interpreted in millimeters. The cylinder is accordingly sized and positioned on the screen (see Figure 4-13).

Figure 4-12
*Properties Window (Geometry
Page) for the Cylinder*

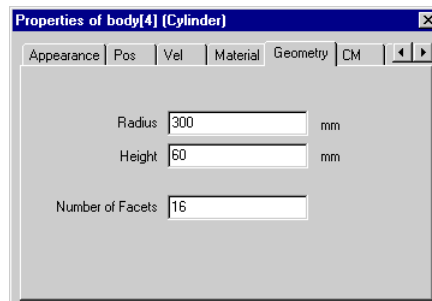
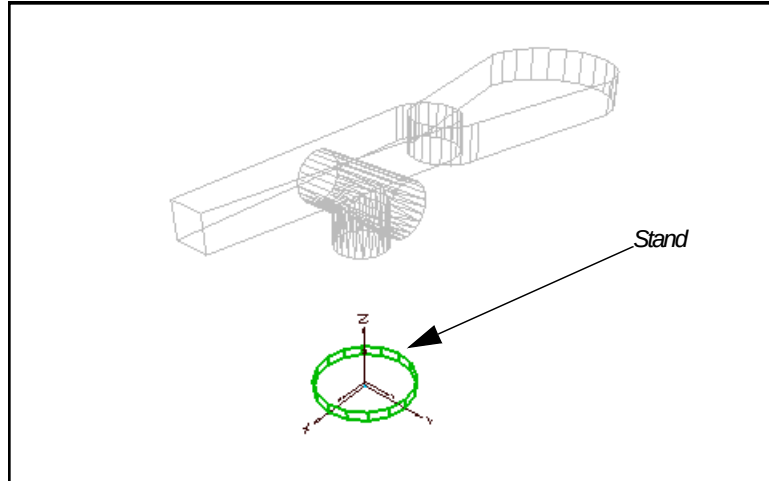


Figure 4-13
Stand Cylinder

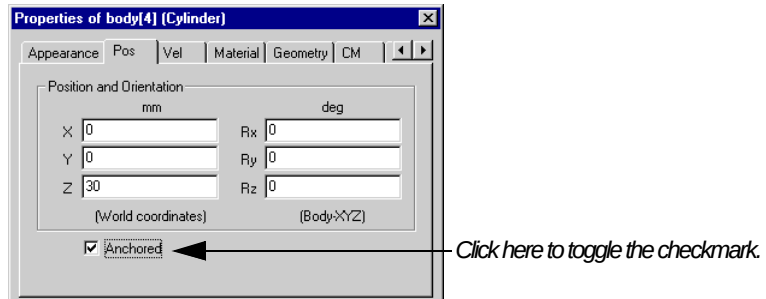


5. Click the **Position (Pos)** tab in the **Properties** window.

Since the stand is a base mount of the manipulator arm and should be immobile, you will designate this body as “anchored.” An anchored body remains stationary throughout the simulation regardless of collisions or gravity.

6. Check the box named **Anchored** (see Figure 4-14).

Figure 4-14
Anchored Checkbox



7. Click the **Appearance** tab, then Name the stand cylinder as “stand”.

See Figure 4-6 for an example.

4.6 Creating the Pedestal

You will create a cylinder to connect the shoulder part and the stand. Note that you have specified where the stand and the shoulder should be located, and you would like to create a cylinder that fits between the two bodies where they are. Working Model 3D provides a convenient feature to create a cylinder automatically sized to fit between two points specified.

Creating the Attachment Points

You will first create two Coords to indicate the endpoints of the cylinder.



1. Click the **Coord** tool on the toolbar.
2. Click at or near the center of the top surface of the stand.
3. Double-click on the **Coord**.

The Properties window appears.

4. If necessary, click the **Position (Pos)** tab in the Properties window, then verify that the **Coord** is located at $(x, y, z) = (0, 0, 30)$ and that **Rx** and **Ry** are 0.

*The value of R_z is rather irrelevant, but R_x and R_y need to be 0 for the **Coord** to “lay flat” on the stand.*

You will now proceed to attach a **Coord** at the bottom end of the shoulder.

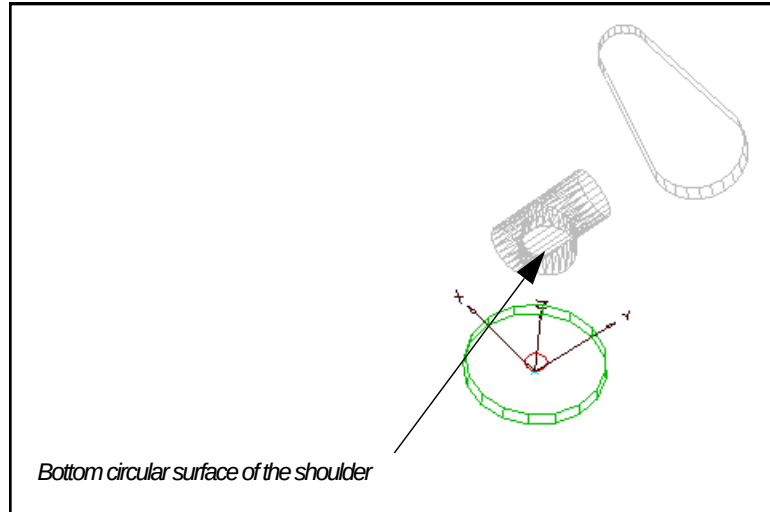


5. Click the **Rotate Around** tool on the **View** toolbar.
6. Click the shoulder (so that the shoulder's coordinate origin will be fixed in the **World** while you rotate the view) and hold the mouse button down.
7. Drag the mouse so that the circular bottom surface of the shoulder faces you (Figure 4-15).

You will attach a **Coord** on this surface. The surface must not be blocked or hidden by another object in order for the **Coord** to be attached (the **Edit Grid** does not interfere in this respect).

Figure 4-15

Attaching Coord to Bottom Face of the Shoulder



8. Click the Coord tool on the toolbar.

The Edit Grid appears.

9. Click the center of the shoulder's bottom circular face.

The Coord will attach to the surface, but not to the Edit Grid.

10. Double-click the Coord, and verify the position and orientation of the Coord are $(x, y, z) = (0, 252, 0)$, $(R_x, R_y, R_z) = (-90, 0, 0)$ on the Position page of the Properties window.

If any values are different, correct them in the Properties window.

Creating the Pedestal Cylinder

To create a cylinder between the two Coords located on the shoulder and the stand:

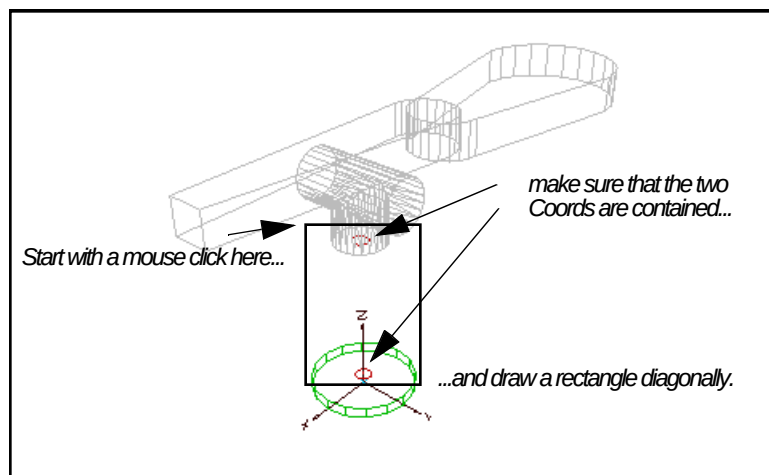
1. Click the blank area of the document, and press H on the keyboard to return to the home view.

You will use the “box-select” method to select two Coords.

2. Click the mouse away from the shoulder and hold the mouse button. Drag the mouse to draw a rectangle so that the two Coords (on the shoulder and the stand) are completely within the selection rectangle (see Figure 4-16).

Working Model 3D selects every object that is completely bounded within the selection rectangle. Any object that is only partially contained will not be selected. You need to make sure that other parts are not completely within the rectangle.

Figure 4-16
Box-selecting Two Coords



3. Release the mouse button.

The selection rectangle disappears, but the two Coords remain selected.

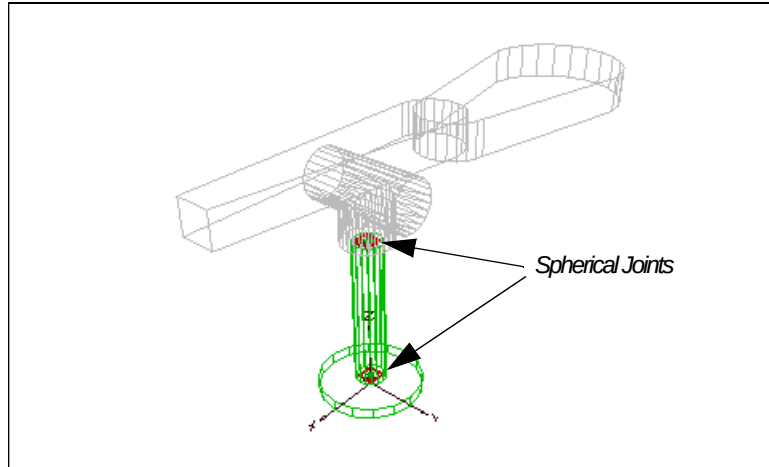
4. Choose **Create Connecting Body** in the **Object** menu.

NOTE: If the menu item **Create Connecting Body** is grayed out, you could have inadvertently selected another body or Coord besides the two specified. Simply click the blank area of the document (to de-select everything), and start the above steps again.

A cylinder appears to connect the two Coords (Figure 4-17). At this point, the cylinder is connected to each body via a spherical joint.

Figure 4-17

*Cylinder Connecting Two Coords
(via Spherical Joints)*



The default radius of the cylinder is set to one tenth of the distance between the two Coords (which becomes the height of the cylinder). The position and orientation of the two original Coords remain unchanged.

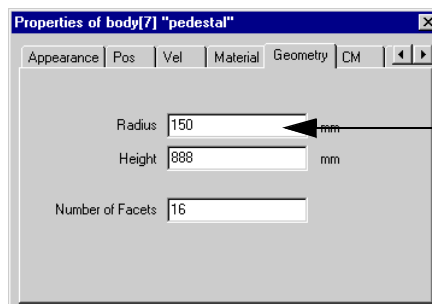
5. Double-click the cylinder, and name the cylinder as “pedestal” on the Appearance page of the Properties window.

See Figure 4-6 for an example.

6. Click the Geometry tab in the Properties window, then set the radius of the pedestal to 150 millimeters.

Figure 4-18

*Properties Window (Geometry
Page) for Pedestal*



Enter 150 as radius.

Modifying the Pedestal Joints

When Working Model 3D creates a cylinder between two Coords, the mating surfaces have spherical joints by default. (This design allows you to create a linkage mechanism very easily once you know the attachment points in space.)

In the manipulator model, the pedestal serves as a platform to which the shoulder is securely mounted. By using an intuitive control provided by Working Model 3D, you will modify the two spherical joints to a rigid joint and a motor.



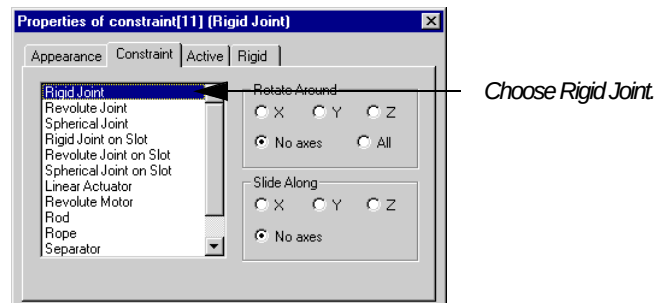
1. Double-click the spherical joint icon located between the stand and the pedestal.

The Properties window appears to show the constraint properties.

2. Choose Rigid Joint from the Type pop-up menu (Figure 4-19).

The constraint icon changes to represent a rigid joint. Note that the joint is currently split (temporarily disabled).

Figure 4-19
Constraint Properties Window



Working Model 3D always splits a joint constraint when its type is changed because the current body configurations may not always satisfy the constraint conditions.

In this case, the Coords are completely aligned, so you can enable the rigid joint constraint immediately as follows.



3. Click the Join/Create Constraint button on the Edit toolbar.
4. Double-click the spherical joint icon located between the pedestal and the shoulder.



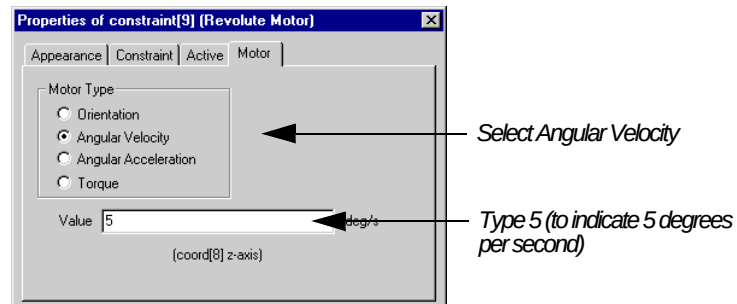
5. If necessary, click the **Constraint** tab in the **Properties** window, then choose **Revolute Motor** from the list of available joint types.

At this point, the motor constraint appears split. You will join the constraint later.

6. Click the **Motor** tab in the **Properties** window, then select **Angular Velocity**, and type **5** in the **Value** field (Figure 4-20).

The motor will rotate at 5 degrees per second.

Figure 4-20
Properties Window for Motor



7. Click the **Join/Create Constraint** button on the **Edit** toolbar.

The motor icon changes to show that it is no longer split.

4.7 Testing Your Model

As emphasized in Exercise 3, you should check from time to time to make sure that the model's components are in working order. If you are creating a complex model and put off testing it until the very end, finding a solution may be extremely difficult when your model yields erroneous simulation results.

So far, you have completed the following:

- You imported three components that were originally available as ACIS files.
- In addition to those three parts, you created the stand and the pedestal, and anchored the stand to set it immobile.

- You connected the shoulder to the pedestal with a motor, and the pedestal to the stand with a rigid joint.
- You specified the motor's angular velocity to be 5 degrees per second.

You can easily expect that the shoulder will rotate and the forearm and midarm will fall as they are subject to gravity.

To start testing, simply run the simulation.



1. **Click the Run button on the Tape Player Control.**

The simulation starts, while the frame indicator shows you the number of frames calculated. Note how the shoulder slowly rotates atop the pedestal as the midarm and forearm fall.



2. **After running for a few dozen frames or so, click the Reset button on the Tape Player Control.**

If the model did not behave as discussed, you should review your model. A few hints are:

- If the shoulder is not rotating, you may have forgotten to specify the motor's property. Review "Modifying the Pedestal Joints" on page 4-19.
- If the stand, pedestal, and the shoulder are falling down, you may have forgotten to anchor the stand. Review "4.5 Creating the Stand".

"Debugging" a large model is not always an easy task. You should take a break every once in a while to verify your model assembly.

4.8 Attaching the Midarm to the Shoulder

To proceed with the rest of the model, you will start out by attaching the midarm to the shoulder with a motor. Again, you need to create attachment points to construct the motor.

Creating Attachment Points

You will attach two Coords, one on the shoulder's side face and the other on the midarm.

1. To obtain the right view, click the blank area of the document and press R on the keyboard.

Note that the side face (the one facing the positive direction of the global Y-axis) of the shoulder is facing you (Figure 4-21).

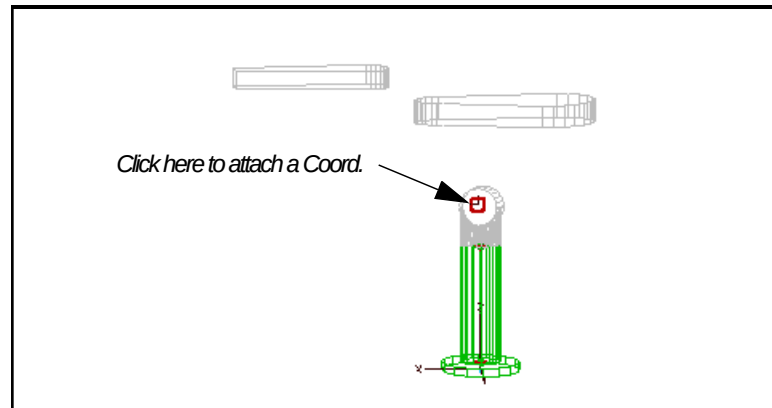


2. Click the Coord tool on the Sketch toolbar.

The Edit Grid appears.

3. Click the center of the shoulder's side face (see Figure 4-21).

Figure 4-21
Right View



4. In the Properties window for the Coord, verify that its position (with respect to the shoulder) is $(x, y, z) = (0, -40, 300)$, and $(R_x, R_y, R_z) = (0, 0, -90)$.

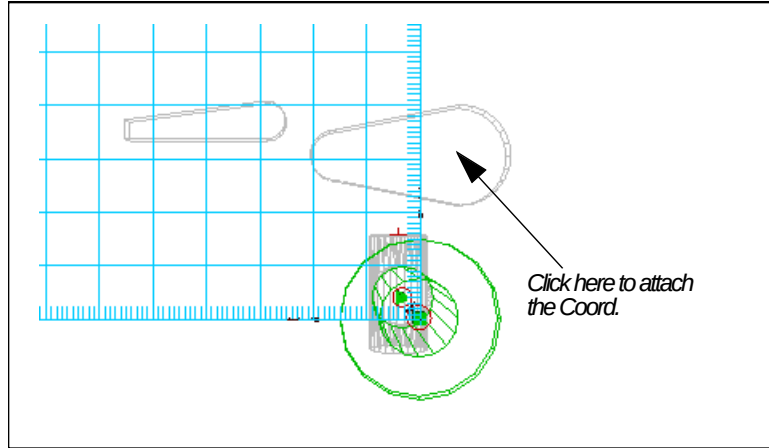
To attach the Coord on the midarm, you will look at the model from the “bottom”—or the negative z-direction—so that the bottom surface of the midarm faces you.

5. Click the blank area of the screen, and press M on the keyboard.

The model view quickly changes to show the view from the bottom (Figure 4-22).

Clicking the blank area of the screen activates the document window and ensures that your keystroke does not affect the edit fields in the Properties window.

Figure 4-22
The View from the Bottom



6. Click the Coord tool on the Sketch toolbar.
7. Click anywhere on the midarm's bottom face (Figure 4-22).
8. In the Properties window of the Coord, verify that the position and orientation of the Coord (with respect to the midarm) is $(x, y, z) = (-300, 0, -75)$, and $(R_x, R_y, R_z) = (-180, 0, 90)$.

Creating a Motor

You will attach the midarm to the shoulder with a motor.

1. Select the Coord on the midarm.

If you find it difficult to select the Coord, try selecting it (coord[13]) in the Object Manager list.

2. Press H to get the Home view.
3. Hold the shift key down, and select the Coord on the side face of the shoulder.

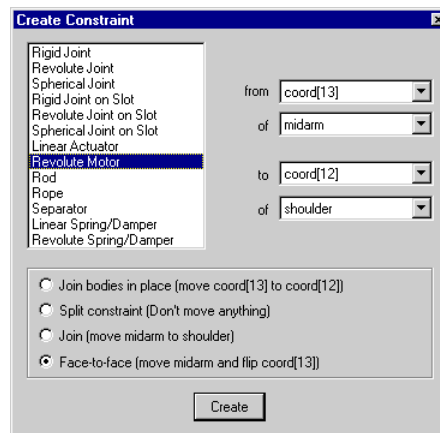
If you find it difficult to select the Coord, try holding down the Control key on your keyboard and selecting it (coord[12]) in the Object Manager list.



Figure 4-23
Create Constraint Window

4. Click the Join/Create Constraint button on the Edit toolbar.

The Create Constraint window appears (Figure 4-23).



5. Choose Revolute Motor as the constraint type.

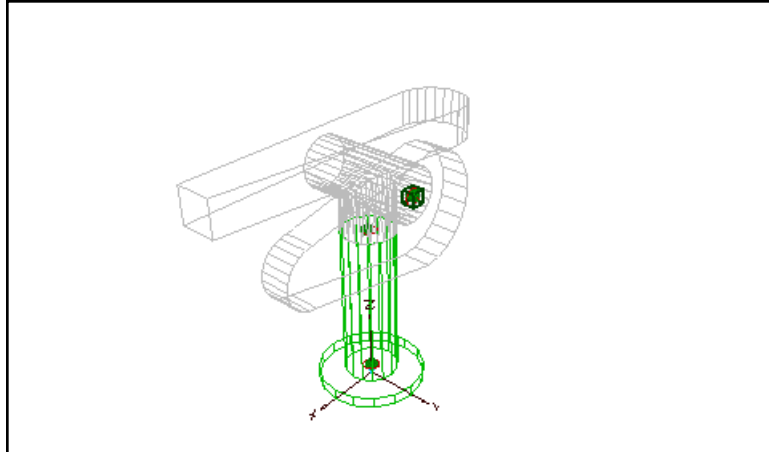
6. Click the option “Face-to-face”

This option will allow the two surfaces to mate “face-to-face”. See below for more information.

7. Click the Create button located in the window.

The midarm attaches to the shoulder. Note that the bottom surface of the midarm mates with the side face of the shoulder. The motor icon appears at the attachment point (see Figure 4-24).

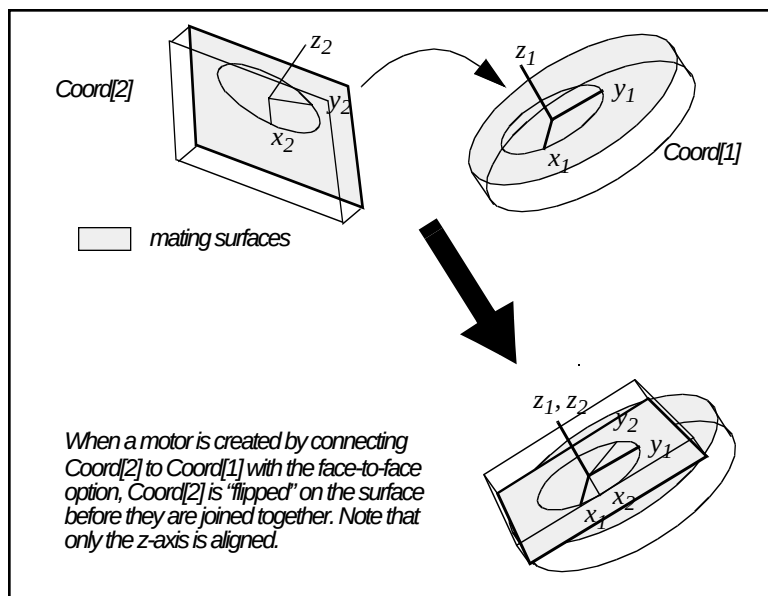
Figure 4-24
Midarm Attached to the Shoulder

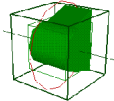


Why “Face-to-face” Option?

As shown in Figure 4-25, 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) 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 4-25
Aligning Coords for Motors and Revolute Joints





Revolute Motor Icon

Specifying the Motor's Angular Velocity

The motor driving the midarm will rotate at a constant angular velocity.

1. **Double-click the Revolute Motor icon on the midarm in the document window or constraint[14] in the Object Manager list.**

The Properties window appears to show the motor properties.

2. **On the Motor page of the Properties window, select Angular Velocity, and type -20 in the Value field.**

The motor will rotate at -20 degrees per second. The angular velocity is measured counterclockwise with respect to the Coords attached to the motor. In this model, therefore, the midarm will initially move up.



3. **To verify, click the Run button on the Tape Player Control.**

The simulation starts, and the midarm slowly moves upward. Note how the shoulder slowly rotates atop the pedestal as the forearm falls.



4. **After running for a few dozen frames, click the Reset button on the Tape Player Control.**

4.9 Attaching the Forearm to the Midarm

The forearm, the last remaining part, is to be connected to the midarm with another motor. This section will complete the model construction. As you have connected the midarm to the shoulder, you need to construct necessary attachment points before you put the parts together.

Creating Attachment Points

You will attach the forearm to the midarm's side surface that is mating with the shoulder.

1. **To obtain the left view, press L on the keyboard.**

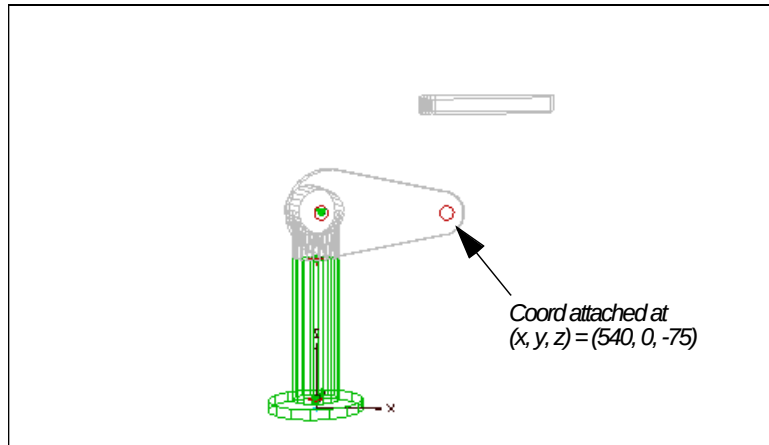
The keyboard shortcut provides a view of the midarm's side surface (see Figure 4-26).



2. Click the Coord tool on the Sketch toolbar.
3. Click on the side surface of the midarm.

See Figure 4-26 for an example.

Figure 4-26
The Left View



4. Using the Properties window, verify or correct so that the Coord's position and orientation are $(x, y, z) = (540, 0, -75)$ and $(R_x, R_y, R_z) = (-180, 0, 90)$ with respect to the midarm.

Let us proceed to attach a Coord on the forearm.

5. To obtain the view from the bottom, click the blank area of the screen (to move away from the Properties window's edit boxes), and press M on the keyboard.



6. Click the Coord tool on the Sketch toolbar.
7. Click on the bottom surface of the forearm.
8. Using the Properties window, verify or correct so that the Coord's position and orientation are $(x, y, z) = (-350, 30, -60)$ and $(R_x, R_y, R_z) = (-180, 0, 90)$.

Creating the Motor

You will connect the midarm and the forearm with a motor.

1. **Click the Coord on the forearm to select it.**
2. **Press H on the keyboard to get the Home View.**

Note that the Coord on the forearm remains selected.

3. **Hold the shift key down and select the Coord attached to the midarm.**

If you have trouble selecting this Coord, you can select it (coord [15]) by holding down the Control key on your keyboard as you click it in the Object Manager list.



4. **Click the Join/Create Constraint button on the Edit toolbar.**

The Create Constraint window appears.

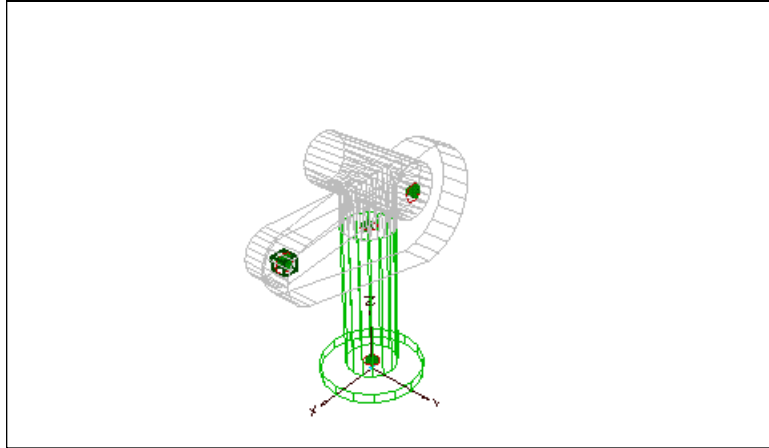
5. **Choose Revolute Motor as the constraint type.**
6. **Click the option “Face-to-face”.**

This option will allow the two surfaces to mate “face-to-face”, as previously done for the midarm.

7. **Click the Create button located in the window.**

The forearm attaches to the midarm. Note that the bottom surface of the midarm mates with the side face of the shoulder, but the forearm connects to the midarm in a completely retracted configuration (see Figure 4-27).

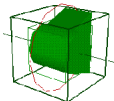
Figure 4-27
Forearm Connected (in Retracted Configuration)



Adjusting the Part Configuration

You will set the robot so that the forearm is in the fully stretched configuration. However, if you were to modify the orientation of the arm directly, the entire robot will change orientation, because the forearm is connected to the rest of the manipulator with the motor. (Although the stand is designated as “anchored” (see “4.5 Creating the Stand”), it is not explicitly constrained to the ground.)

To modify the configuration of the forearm independently, you need to *split* the motor constraint, reconfigure the forearm, and *join* the constraint again.



Revolute Motor Icon



1. Click the Revolute Motor icon once to select it.

When selected, the motor icon shows a green cube with short lines to show its degrees of freedom. The lines represent the axes on which the constraint allows rotation.

2. Click the Split button on the Edit toolbar.

The motor icon changes to indicate that the constraint is split (temporarily inactive).

3. Double-click the forearm.

If you find it difficult to select it, double-click it (“forearm”) in the Object Manager list.

4. Enter $(R_x, R_y, R_z) = (90, 0, 0)$ as the orientation of the forearm.

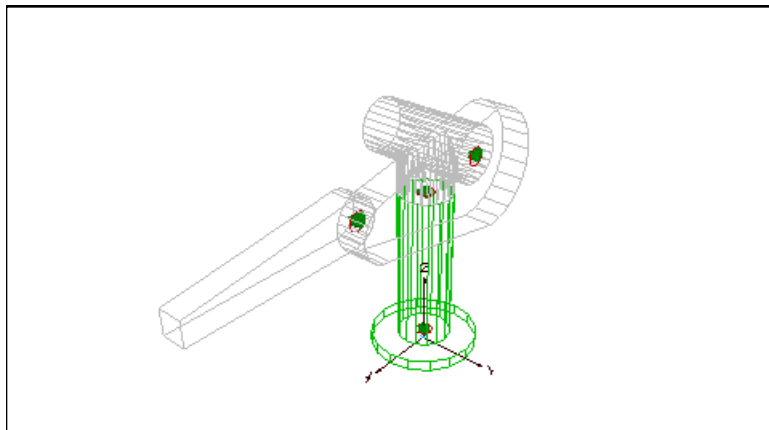
The forearm changes its orientation while its coordinate origin is fixed in the World. A green line appears along with the split motor icon to indicate that the two Coords are still related.

5. Click the split motor icon in the Object Manager list to select it.
6. Click the Join/Create Constraint button on the Edit toolbar.

The forearm is again connected to the midarm with the motor, this time in the fully stretched configuration (Figure 4-28).



Figure 4-28
Forearm Connected (in Fully Stretched Configuration)



Specifying the Motor Characteristics

Like the other two motors joining the robot arm pieces, you will specify this motor so that it rotates at a constant angular velocity.

1. Double-click the motor joining the midarm and the forearm in the document window or the Object Manager list.

The Properties window appears to show the motor properties.

2. Click the Motor tab in the Properties window, then select Angular Velocity, and type -30 in the value field.

The motor will rotate at -30 degrees per second. The angular velocity is measured counterclockwise with respect to the Coords attached to the motor. In this model, therefore, the forearm will initially move down.

4.10 Running the Simulation

To run the simulation:



1. Click the Run button on the Tape Player Control.

The simulation starts, and the arms start to fold as the shoulder rotates atop the pedestal.



2. After running for 100 frames or so, click the Reset button.

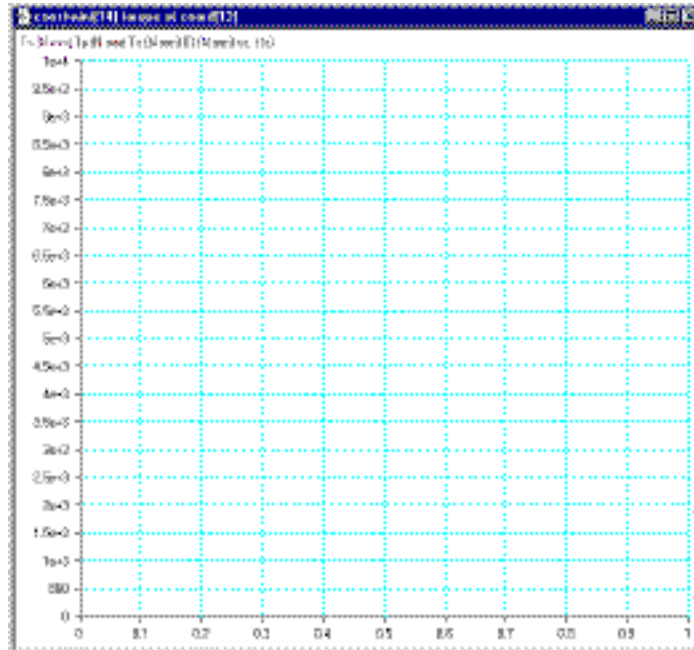
Taking Measurements

You will measure the torque transmitted by the shoulder motor (the motor attached to the shoulder driving the midarm). Working Model 3D allows you to design the mechanism by simply specifying the required angular velocity. Once the model behaves as you expected, you can use meters to measure the torque, so that you can rate the power required by the motor to maintain the velocity specification.

1. Click the motor connecting the midarm and the shoulder.
2. Choose Constraint Torque on Base Point in the Measure menu.

A meter appears to measure the x, y, and z components of the torque as well as the magnitude (Figure 4-29).

Figure 4-29
Meter Window

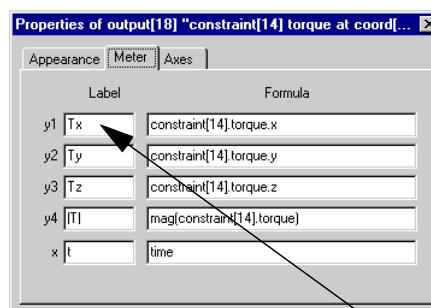


The only measurement of interest here is the torque in the z-axis. Therefore, you will eliminate the display of the other components.

3. Double-click the meter window.

The Properties window appears (Figure 4-30).

Figure 4-30
Properties Window (for a Default Torque Meter)



Select the text and delete it

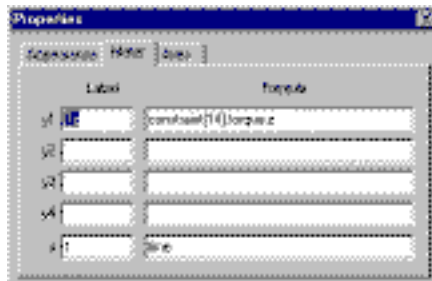
4. Click the Meter tab in the Properties window and then select the text Tx (in the y1 Label field, as shown in Figure 4-30). Press the delete key and hit Enter.

Note that the y1 field disappears altogether (along with the formula language expression) and all the other fields move up.

5. Repeat the process for the labels Ty and |T|, so that the only remaining fields are t (for time, in x field), and Tz (in y1 field).

See Figure 4-31 for an example.

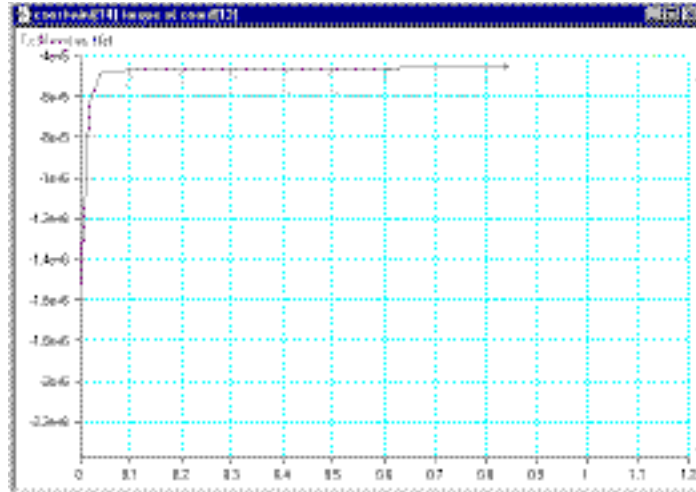
Figure 4-31
Properties Window for Meter
(Modified to Show Tz Only)



6. Close the Properties window.
7. Click the Run button on the Tape Player Control.

The simulation starts and the meter plots the torque transmitted by the motor (Figure 4-32).



Figure 4-32*Meter Plotting Torque with Jump*

Rescaling Meter Axis

Note that the meter has a large jump in the torque plot at the onset of the simulation. As a result, the meter's y-axis automatically scales itself to accommodate the large value, and the rest of the data becomes difficult to analyze since the plot looks flat.

This phenomenon reveals an interesting and convincing aspect of dynamics and how Working Model 3D simulates it. Observe the following predicates:

- the entire mechanism is initially at rest, and
- the motor is specified as an angular velocity motor; i.e., Working Model 3D will exert as much torque as necessary to attain the target angular velocity in as few frames as possible.

Consequently, the shoulder motor (and other motors as well) initially exerts a large torque to attain the angular velocity. Therefore, in practice, the torque reading in the first several frames may not show a strong correlation with the physical model being simulated.

Note that the same phenomenon applies to other constraints, such as a velocity actuator. A velocity actuator may exert a large force to attain the specified target velocity during the first several frames, if the initial relative velocity of the connected parts is significantly different from the specified velocity.

In order to analyze the realistic portion of the torque plot, you will limit the plotting range of the graph to extract the data you are interested in.

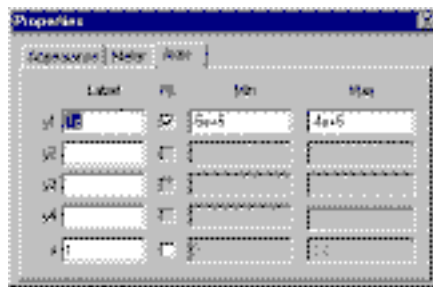
1. **Double-click the meter window.**

The Axes Properties window appears.

2. **To disable automatic scaling, click the Fit checkbox. Enter $-5e+5$ in the minimum field, and $-4e+5$ in the maximum field.**

The meter plots the torque range of -500,000 to -400,000 Newton-millimeters.

Figure 4-33
Specifying Axis Scaling



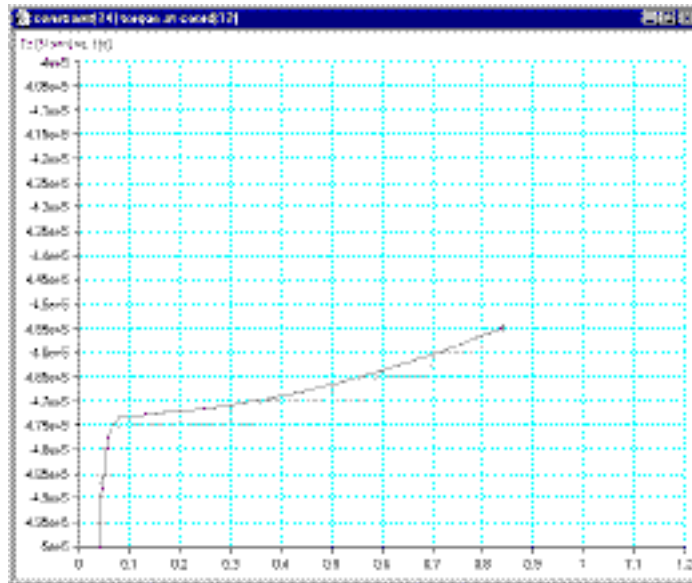
When you click the Fit checkbox, Working Model 3D applies the forced-scaling on the meter axis instead of automatic scaling. The meter will always plot torque only within the specified range. Note that the simulation history remains intact, as changing the scale has no effect on the initial condition of the model.



3. **Click the Run button on the toolbar and continue the simulation.**

The simulation starts and the meter plots the torque transmitted by the motor (Figure 4-34).

Figure 4-34
*Meter Plotting Torque in the
 Specified Range*



The meter's x-axis rescales in terms of time. Meanwhile, the y-axis does not rescale because of the fixed scaling.

NOTE: It is reassuring to note that even with the jump discontinuities in initial velocities and angular velocities caused by the angular velocity motors, the value of T_z at $t = 1.5$ seconds is within 0.5 percent of $-4.153e+5$ N-mm, which is the value obtained for T_z if there are no jump discontinuities in initial velocities and/or angular velocities.

E X E R C I S E 5

Working Model for Mechanical Desktop

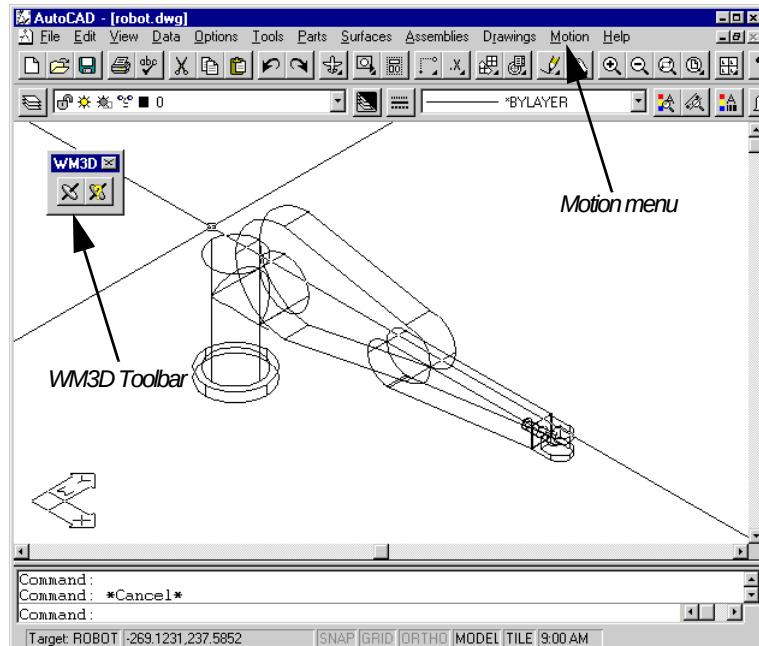
This tutorial illustrates how you can add motion to a model that has been created in Mechanical Desktop. It assumes that Working Model for Mechanical Desktop is already installed on your system. If you have not yet installed the program, please refer to the accompanying booklet, “Getting Started with Working Model 3D”, for installation instructions.

5.1 Importing a Mechanical Desktop Drawing

1. **From within Mechanical Desktop, open the file “Robot.DWG” located in your “Program Files\Working Model 3D\Mechanical Desktop\Samples\First Tutorial” directory.**

Your window should appear as shown in Figure 5-1. Notice that there is a new toolbar labeled WM3D as well as a new item named Motion in the menu bar.

Figure 5-1
Mechanical Desktop Window



2. Choose **Simulate Motion** from the **Motion** menu OR click the **Gyroscope** button on the **WM3D** toolbar (see Figure 5-2).

Either action activates Working Model for Mechanical Desktop, which maps the assembly components and constraints into real bodies and joints within the Working Model 3D window as shown in Figure 5-3

Figure 5-2
Activating Working Model for Mechanical Desktop

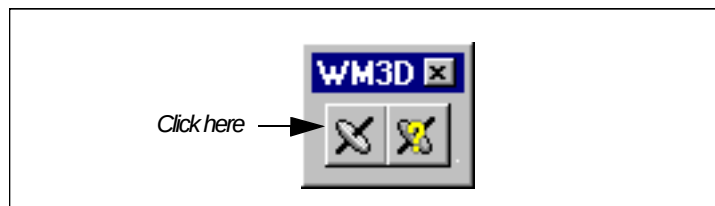
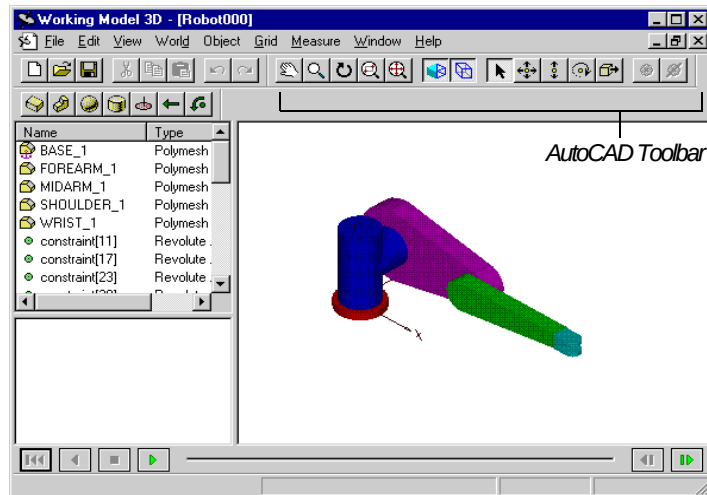


Figure 5-3
*Mechanical Desktop Model
 Opened in Working Model 3D*



5.2 Running the Model



1. Click the Run button in Working Model 3D's Tape Player Control.

Working Model 3D begins to simulate the motion of the model. Since this is the first time the simulation is being run, Working Model 3D is calculating the dynamics and storing the data.



2. Repeat the simulation by clicking the Stop button, then the Reset button, and then the Run button again.

The animation is faster because the history has already been calculated.

5.3 Improving the Model

When the model was imported, all of the joints mapped to revolute joints. Thus, the robot is acting as if it has lost all power to its motors. In this step you'll change those joints to motors to accurately simulate the motion.



1. If necessary, click the Toggle Wireframe button on the AutoCAD toolbar.

The shading is removed, and the model is displayed in wireframe mode.

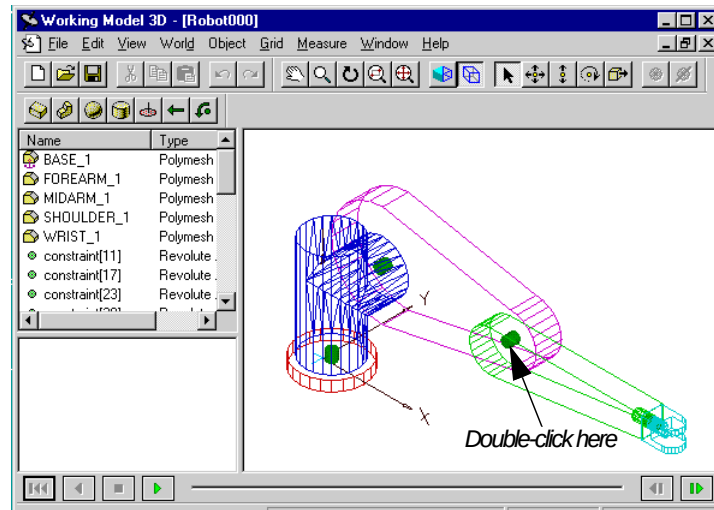


2. Click the Zoom tool on the AutoCAD toolbar, then position the mouse over the center of the drawing and drag down to move your view closer to the model.
3. Double-click the “elbow” joint that is the farthest to the right on the “MIDARM_1” in the document window (see Figure 5-4).

The Properties window for the joint appears.

Figure 5-4

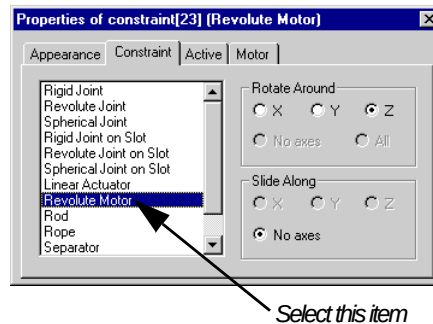
Double-click the “Elbow” Joint on the Robot Arm



4. If necessary, click the Constraint tab in the Properties window. Then, select Revolute Motor from the list of available constraint types (see Figure 5-5).

The revolute joint is changed into a revolute motor.

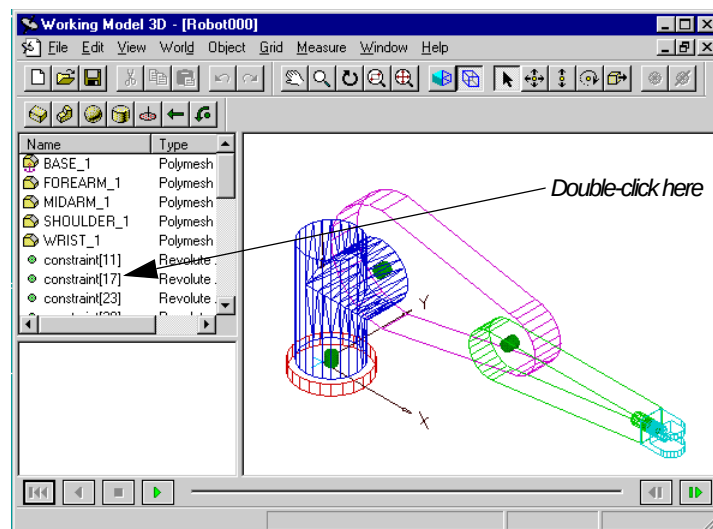
Figure 5-5
Properties Window (Constraint Page)



- Double-click the “shoulder” joint on the other end of the “MIDARM_1” (see Figure 5-6). It appears as constraint[17] in the Object Manager list.

You may have to move or close the Properties window to see the joint. Notice that the Properties window updates to display information for the newly selected joint.

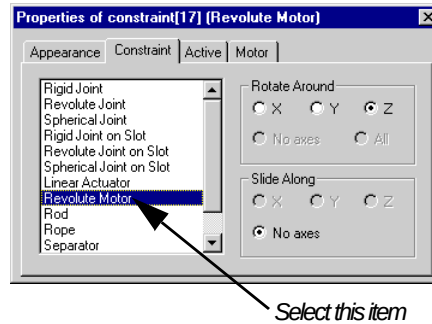
Figure 5-6
Double-click the “Shoulder” Joint on the Robot Arm



- Select Revolute Motor from list of available constraint types for this joint.

Figure 5-7

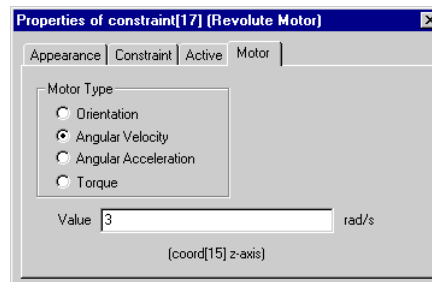
Properties Window (Constraint Page)



7. Click the Motor tab, then change the velocity of the motor to 3 as shown in Figure 5-8.

Figure 5-8

Properties Window (Motor Page)



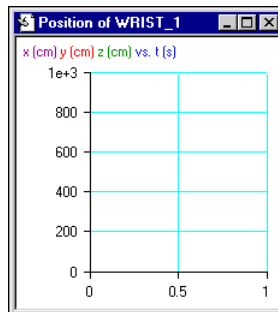
You can close the Properties window now.

5.4 Testing the Improvements

1. Select any body on the robot and choose **Position** from the **Measure** menu.

A meter window appears which displays the position of the body during the simulation.

Figure 5-9
Position Meter



2. Click the **Toggle Wireframe** button on the AutoCAD toolbar.

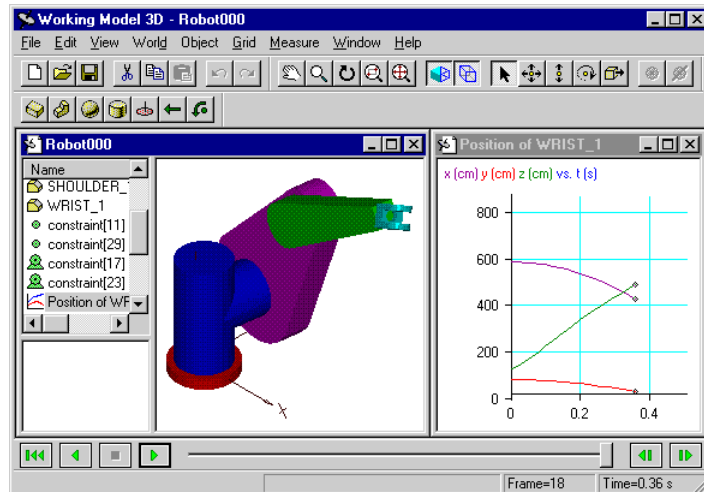
The shading returns and the model is displayed in shaded mode.



3. Click the **Run** button in the **Tape Player Control**.

The robot moves quite differently this time.

Figure 5-10
Simulation Running with Meter



4. Click the Stop button.

Congratulations! You have just completed the basic steps required to simulate the motion of an assembly.

You can also try measuring other quantities such as motor torques. Simply select a motor and choose Constraint Torque on Base Point from the Measure menu. Working Model 3D then displays the torque required to maintain the specified velocity. This is useful when trying to determine the appropriate motor power for a mechanism. Notice the force spike at the beginning of the simulation. Working Model 3D must do this to achieve the desired velocity from rest.

E X E R C I S E 6

Working Model for Solid Edge

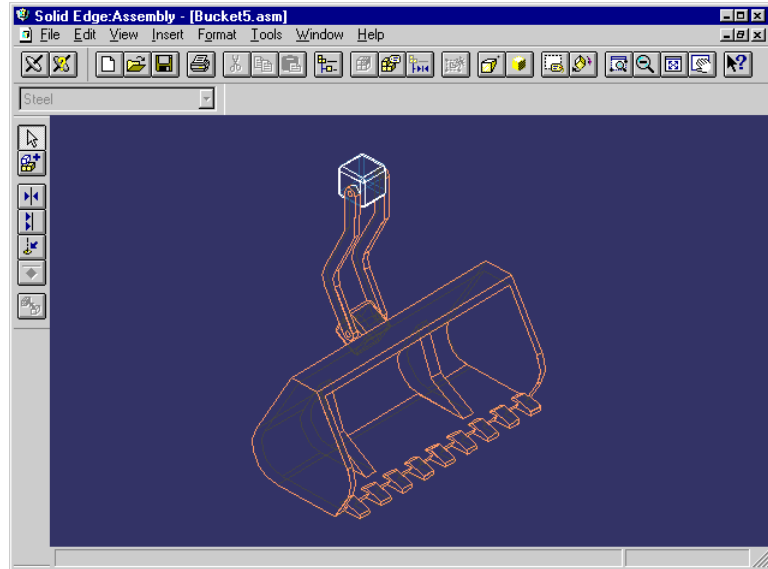
This tutorial illustrates how you can add motion to a model that has been created in Solid Edge. It assumes that Working Model for Solid Edge is already installed on your system. If you have not yet installed the program, please refer to the accompanying booklet, “Getting Started with Working Model 3D”, for installation instructions.

6.1 Importing a Solid Edge Drawing

1. From within Solid Edge, open the file “Bucket5.ASM” located in your “Program Files\Working Model 3D\Solid Edge\Samples\First Tutorial” directory.

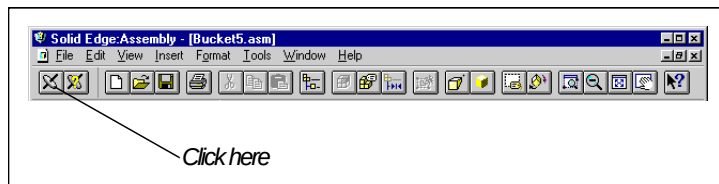
Your window should appear as shown in Figure 6-1. Notice that there are two new Working Model for Solid Edge tools at the upper left of the toolbar.

Figure 6-1
Solid Edge Window



2. Click the Gyroscope button on the toolbar (see Figure 6-2).

Figure 6-2
Activating Working Model for Solid Edge

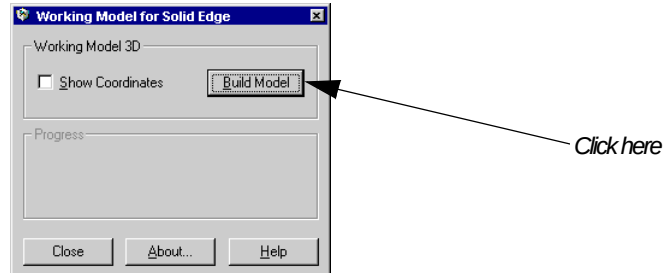


The Working Model for Solid Edge dialog appears.

NOTE: If this is your first time using Working Model for Solid Edge, you will see a welcome screen. Dismiss the screen and click the Gyroscope button again to proceed.

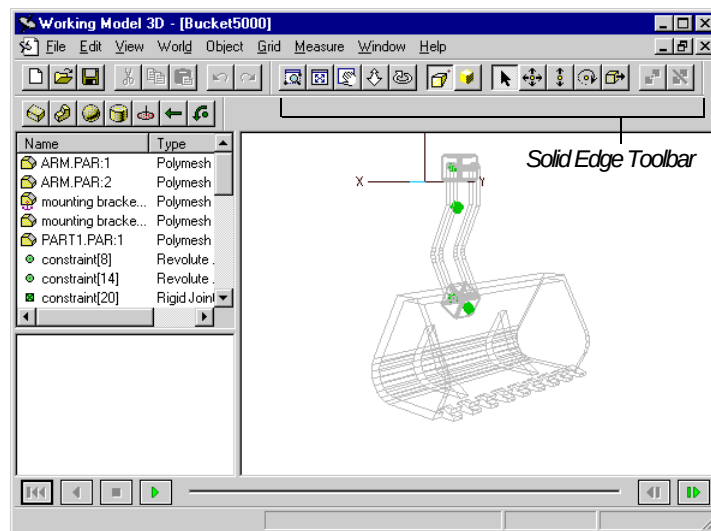
3. Click the Build Model button in the dialog (see Figure 6-3).

Figure 6-3
Working Model for Solid Edge
Dialog



Working Model for Solid Edge maps the assembly components and constraints into real bodies and joints within the Working Model 3D window as shown in Figure 6-4.

Figure 6-4
Solid Edge Model Opened in
Working Model 3D



6.2 Running the Model



1. Click the Run button in Working Model 3D's Tape Player Control.

Working Model 3D begins to simulate the motion of the model. Since this is the first time the simulation is being run, Working Model 3D is calculating the dynamics and storing the data.



2. Repeat the simulation by clicking the Stop button, then the Reset button, and then the Run button again.

The animation is faster because the history has already been calculated.

6.3 Improving the Model

When the model was imported, all of the joints mapped to revolute joints. Thus, the bucket is acting as if it has lost all power to its motors. In this step you'll change those joints to motors to accurately simulate the motion.

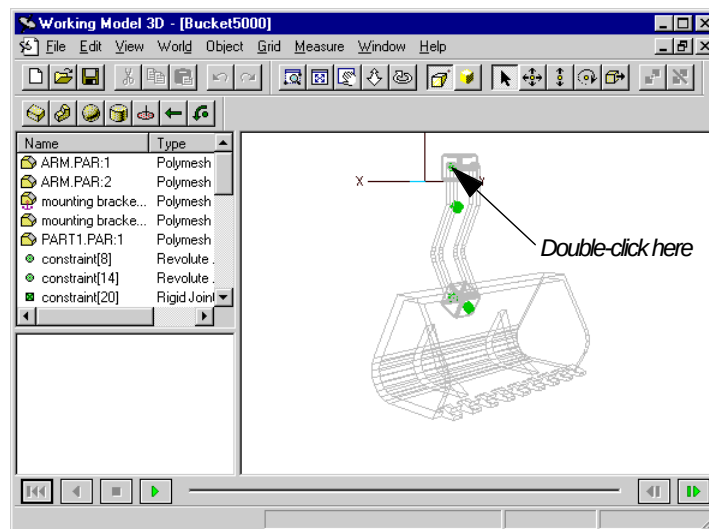


1. Click the Zoom In/Out tool on the Solid Edge toolbar, then position the mouse over the center of the drawing and drag down to move your view closer to the model.
2. Double-click the joint that is topmost in the document window (see Figure 6-5).

The Properties window for the joint appears.

Figure 6-5

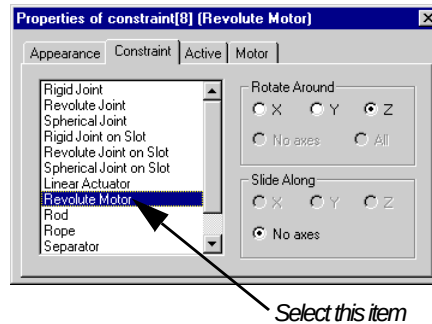
Double-click the Topmost Joint on the Bucket



3. If necessary, click the **Constraint** tab in the **Properties** window. Then, select **Revolute Motor** from the list of available constraint types (Figure 6-6).

The revolute joint is changed into a revolute motor.

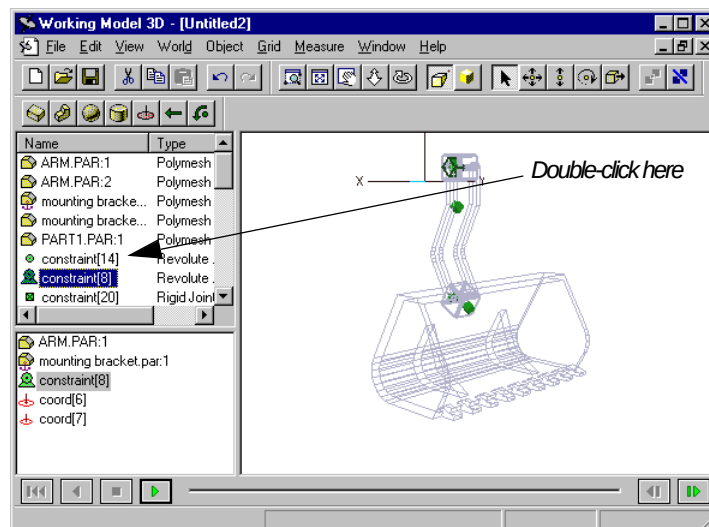
Figure 6-6
Properties Window (Constraint Page)



4. Double-click the remaining revolute joint (see Figure 6-7). It appears as constraint[14] in the Object Manager list.

You may have to move or close the Properties window to see the joint. Notice that the window updates to display information for the newly selected joint.

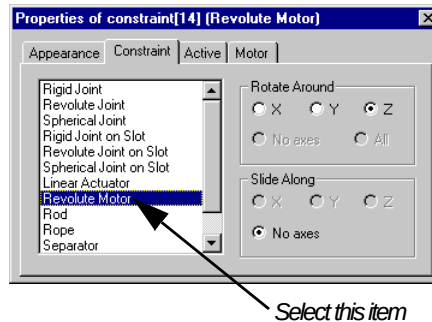
Figure 6-7
Double-click Other Revolute Joint on the Bucket



5. Select Revolute Motor from list of available constraint types for this joint.

Figure 6-8

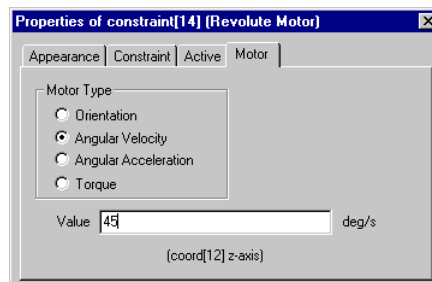
Properties Window (Constraint Page)



6. Click the Motor tab, then change the velocity of the motor to 45 as shown in Figure 6-9.

Figure 6-9

Properties Window (Motor Page)



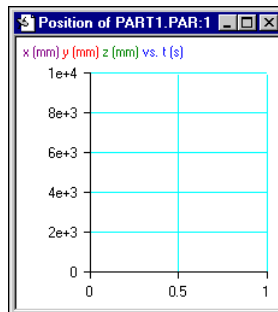
You can close the Properties window now.

6.4 Testing the Improvements

1. Select the scoop of the bucket assembly and choose Position from the Measure menu.

A meter appears which displays the position of the body during the simulation. The axis scaling may appear different depending on the window size.

Figure 6-10
Position Meter



2. Click the document window and then the Toggle Wireframe button on the Solid Edge toolbar.

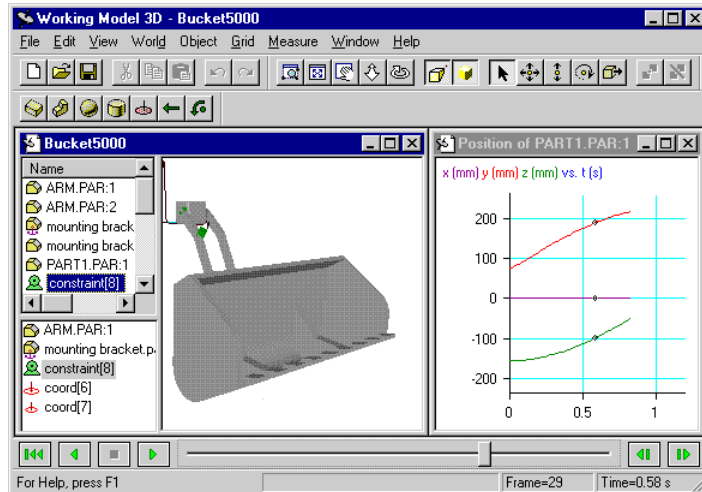
The model is displayed in shaded mode.



3. Click the Run button in the Tape Player Control.

The bucket moves quite differently this time.

Figure 6-11
Simulation Running with Meter



4. Click the Stop button.

Congratulations! You have just completed the basic steps required to simulate the motion of an assembly.

You can also try measuring other quantities such as motor torques. Simply select a motor and choose Constraint Torque on Base Point from the Measure menu. Working Model 3D then displays the torque required to maintain the specified velocity. This is useful when trying to determine the appropriate motor power for a mechanism. Notice the force spike at the beginning of the simulation. Working Model 3D must do this to achieve the desired velocity from rest.

EXERCISE 7

Working Model for SolidWorks

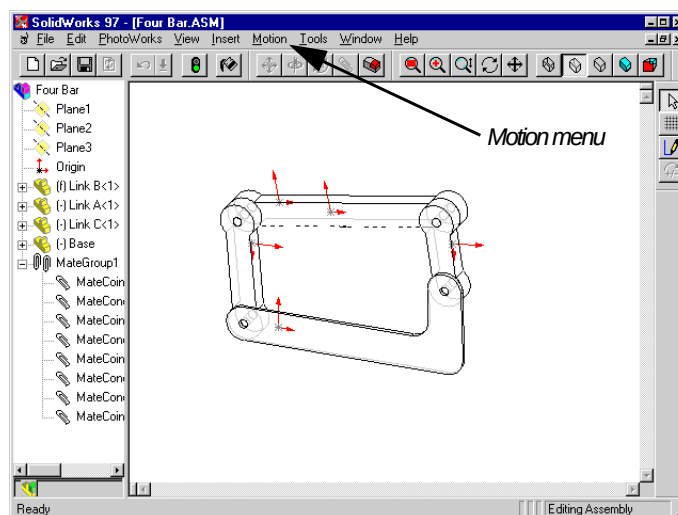
This tutorial illustrates how you can add motion to a model that has been created in SolidWorks. It assumes that Working Model for SolidWorks is already installed on your system. If you have not yet installed the program, please refer to the accompanying booklet, “Getting Started with Working Model 3D”, for installation instructions.

7.1 Importing a SolidWorks Drawing

1. From within SolidWorks 97, open the file “Four Bar.ASM” located in your “Program Files\Working Model 3D\SolidWorks\Samples\First Tutorial” directory.

Your window should appear as shown in Figure 7-1. Notice that there is a new item named Motion in the menu bar.

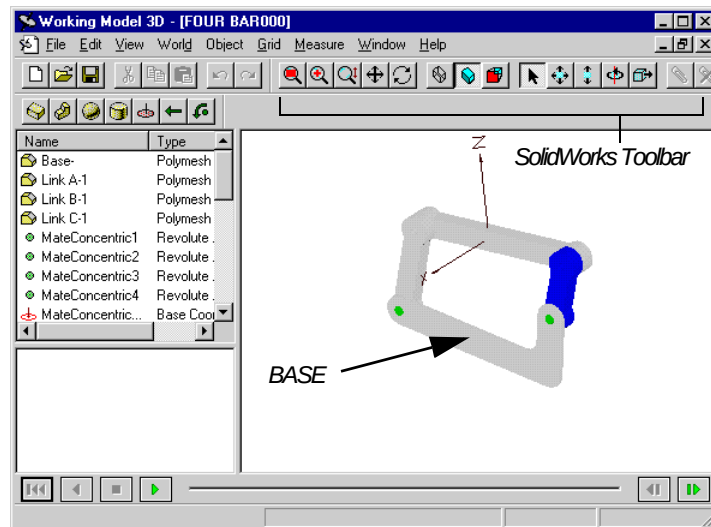
Figure 7-1
SolidWorks Window



2. Choose **Simulate Motion** from the **Motion** menu.

This activates Working Model for SolidWorks, which maps the assembly components and constraints into real bodies and joints within the Working Model 3D window as shown in Figure 7-2.

Figure 7-2
SolidWorks Model Opened in Working Model 3D



7.2 Running the Model

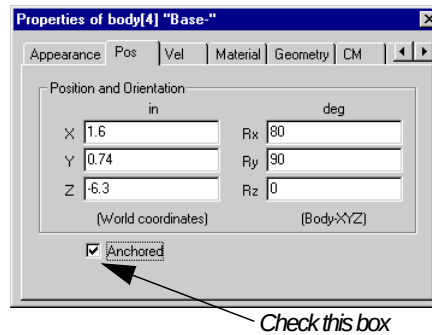
Currently, the entire mechanism is floating in space. We need to tell Working Model 3D which part to anchor to the background.

1. From within the **Working Model 3D** window, double-click the bottom L-bracket (labeled “**BASE**” in Figure 7-2).

The Properties window for the bracket appears with the Position page active.

2. If necessary, click the **Position (Pos)** tab.
3. Click the **Anchored** check box in the Properties window as shown in Figure 7-3.

Figure 7-3
Properties Window
(Position Page) for Base



4. **Click the Run button in Working Model 3D's Tape Player Control.**

Working Model 3D begins to simulate the motion of the model. Since this is the first time the simulation is being run, Working Model 3D is calculating the dynamics and storing the data.



5. **Repeat the simulation by clicking the Stop button, then the Reset button, and then the Run button again.**

The animation is faster because the history has already been calculated.

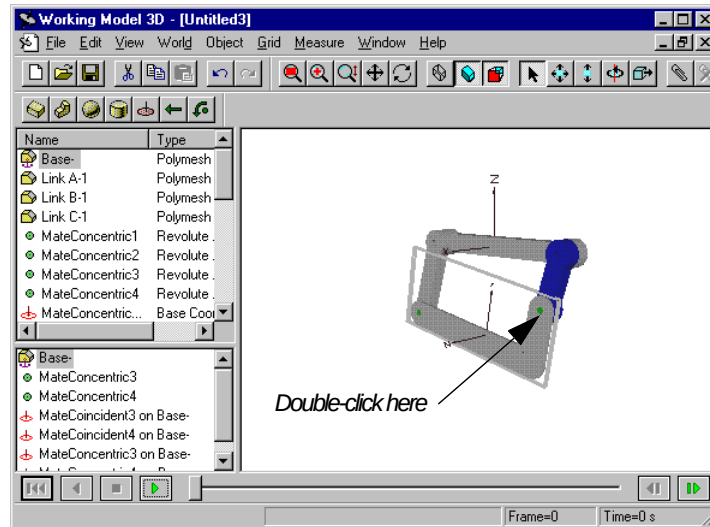
7.3 Improving the Model

When the model was imported, all of the joints mapped to revolute joints. Thus, the linkage is moving only in response to the effects of gravity. In this step you'll change those joints to motors to accurately simulate the motion.

1. **Double-click the joint that is on the right end of the bottom L-bracket (see Figure 7-4).**

Figure 7-4

Double-click the Joint
at the Right End of the Bracket



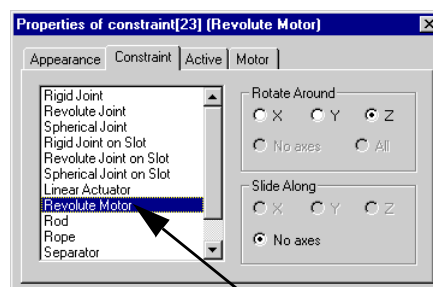
The Properties window for the joint appears.

2. If necessary, click the **Constraint** tab in the Properties window. Then, select **Revolute Motor** for the list of available joint types (see Figure 7-5).

The revolute joint is changed into a revolute motor. You can close the Properties window now.

Figure 7-5

Properties Window (Constraint
Page)

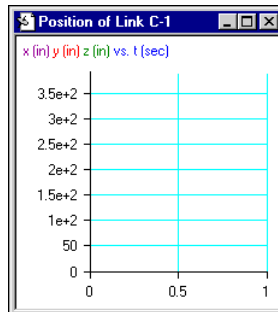


7.4 Testing the Improvements

1. Select any one of the unanchored bars in the linkage and choose Position from the Measure menu.

A meter window appears which displays the position of the body during the simulation. The axis scaling may appear different depending on the window size.

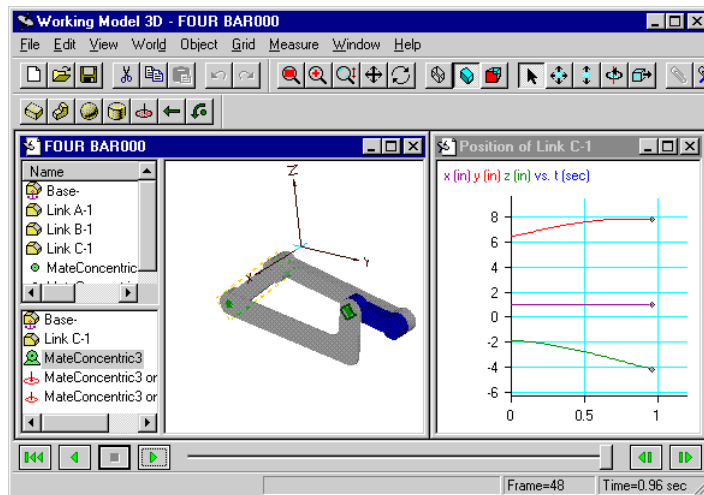
Figure 7-6
Position Meter



2. Click the Run button in the Tape Player Control.

The linkage moves quite differently this time.

Figure 7-7
Simulation Running with Meter





3. Click the Stop button.

Congratulations! You have just completed the basic steps required to simulate the motion of an assembly.

You can also try measuring other quantities such as motor torques. Simply select a motor and choose Constraint Torque on Base Point from the Measure menu. Working Model 3D then displays the torque required to maintain the specified velocity. This is useful when trying to determine the appropriate motor power for a mechanism. Notice the force spike at the beginning of the simulation. Working Model 3D must do this to achieve the desired velocity from rest.