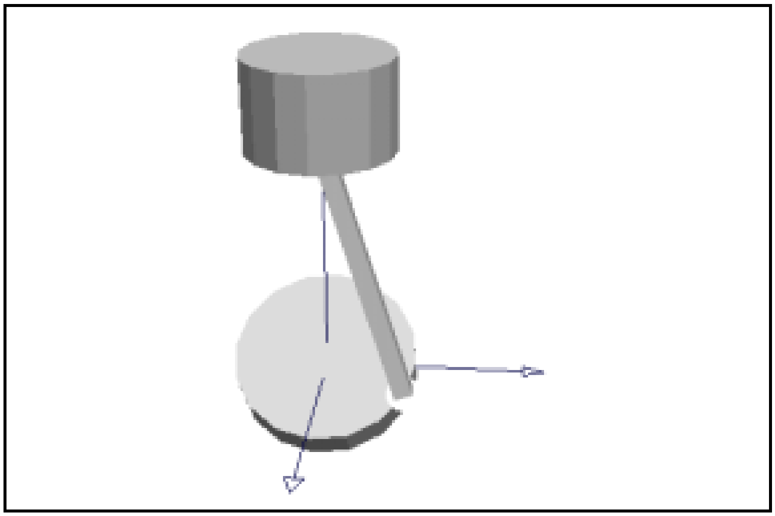
CHAPTER 7

Building a Model



Objectives

In this exercise you will learn to

- Build and assemble a model.
- Use meters to characterize the performance of the model.

Software

MSC.visualNastran 4D or MSC.visualNastran Motion

Support Files

None

This exercise shows you how to create constraints and assemble bodies in MSC.visualNastran Desktop and how to access data from the simulation through meters. The target model is a simple piston engine 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 be illustrated only in a three-dimensional simulator.

In a conventional internal combustion engine, a gas explosion in the cylinder pushes down the piston to drive the crankshaft. For simplicity, this tutorial exercise creates 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.

In the initial steps that follow, you will adjust the animation frame rate and the unit system in the MSC.visualNastran Desktop file. The unit system adjustment is 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.

Establish the Modeling Parameters

Set Up the Animation Frame Rate

1. Launch MSC.visualNastran Desktop.

2. Choose **Simulation Settings** in the **World** menu and click **Integration** in the **Simulation Settings** window.

3. Enter 200 in the **Rate** field.

4. Click **Close**.

Set Up the Units

1. Choose **Display Settings** in the **World** menu and click **Units** in the **Display Settings** menu.

The Units page of the **Settings** window appears. By default, the SI (degrees) unit system is chosen.

Figure 7-1
Display Settings Window (Units Page)

2. Set the distance units to **millimeters (mm)**.

3. Click the **X** in the top right corner to close the window.

All distance measurements in this document will now be in millimeters.

Set Up the Edit Grid

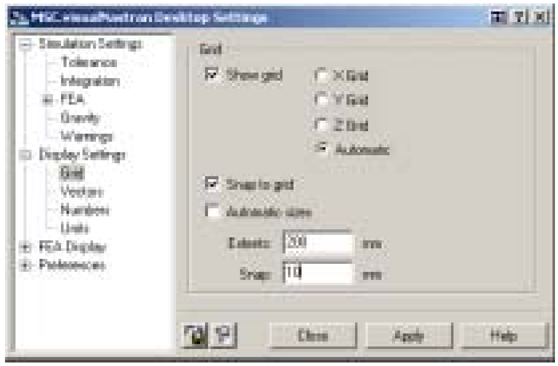
You will modify the size of the Edit Grid and zoom rate of the document so that they are appropriate for the dimensions of the components you will create.

1. Choose **Grid** in the **View** menu. Next, choose **Show Grid** in the **Grid** submenu. The Edit Grid appears in the modelling window.

2. Choose **Grid** in the **View** menu. Then choose **Grid Settings** in the **Grid** submenu.

3. Click **Grid** to view the **Grid** page of the **Display Settings** dialog.

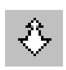
Figure 7-2
Display Settings Window (Grid Page)



4. Deselect **Automatic sizes**. Type 200 in the **Extents** field and 10 in **Snap** field.

5. Close the **Display Settings** dialog.

NOTE: The units in the dialog are based on millimeters. MSC.visualNastran Desktop automatically scales the document window to accommodate the smaller Edit Grid.



6. 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.

---

7. 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 in the other direction, you can zoom out.

8. 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.

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

---

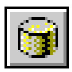
9. Choose **Wireframe** from the **View** menu.

Create a Piston Model

Create the Crankshaft

In this exercise, the crankshaft is modeled as a single disk. You will later attach a connecting rod from the piston to this disk so that the phase will be offset by 180°.

You will specify the geometry and position of the body using the Properties window.



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

You can select the toolbars that you wish to display in MSC.visualNastran Desktop. Choose **Toolbars** from the **View** menu. Then select the toolbar(s) to display and click **OK**.

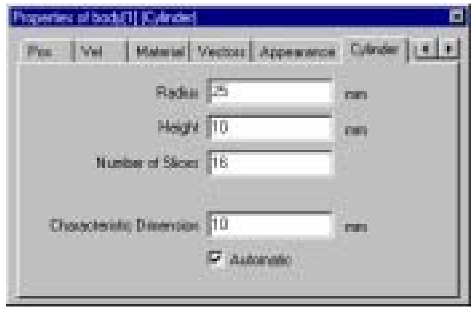
---

2. Click once anywhere 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 List** to display its **Properties** window.
4. Click the **Pos** tab to view the **Position** page.

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

5. Enter the position:(x, y, z) = (0, 0, 0).
6. Click the **Cylinder** tab.

Figure 7-4
Properties Window (Cylinder Page)

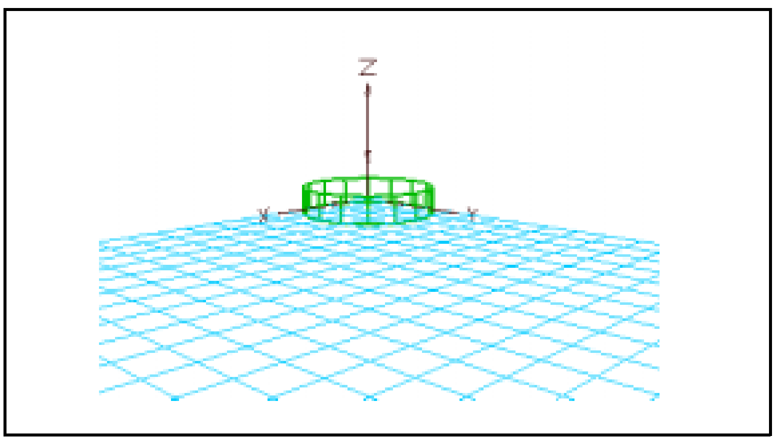


7. Enter Radius = 25, Height = 10. Note that all the numbers are interpreted in millimeters.

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 as shown in the following figure.

Figure 7-5
Crankshaft Sized and Positioned

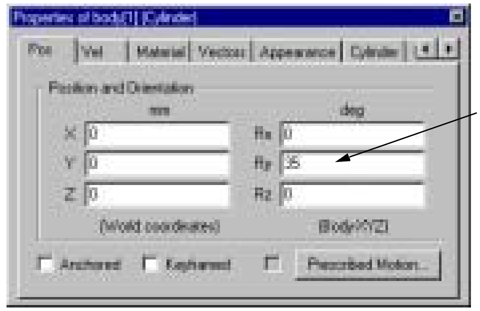


Currently, the crankshaft is oriented so that it lies flat on the xy plane. You will reorient the crankshaft so that it is slightly skewed.

9. Double-click the crankshaft cylinder in the document window or the **Object List**.

10. In the **Properties** window, click the Position (**Pos**) tab.

Figure 7-6
Properties Window (Position Page)



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

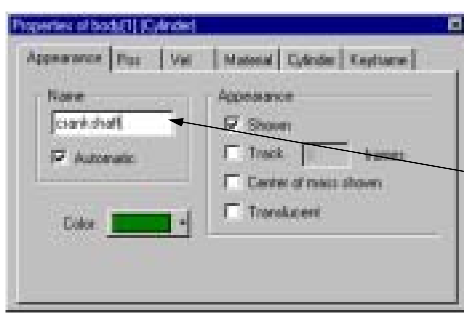
The crankshaft is reoriented as soon as you tab out of the Ry field. You do not need to close the **Properties** window to have the changes take effect.

Assign a Name to the Crankshaft

As you create bodies in a document, MSC.visualNastran Desktop 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.

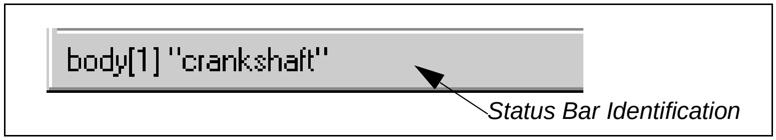
Figure 7-7
Properties Window (Appearance Page)



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

To see how the custom name helps you in identifying bodies, simply "hover" the mouse over the crankshaft cylinder. The body's variable name is displayed at the location of the mouse pointer. The status bar in the bottom left corner of the modeling window also identifies the body with the variable name (body[n]) as well as the custom name you have just assigned. See **Figure 7-8** for an example.

Figure 7-8
Status Bar Identifying Object

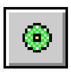


The custom name for this body also appears in the **Object List**.

Attach a Motor to the Crankshaft

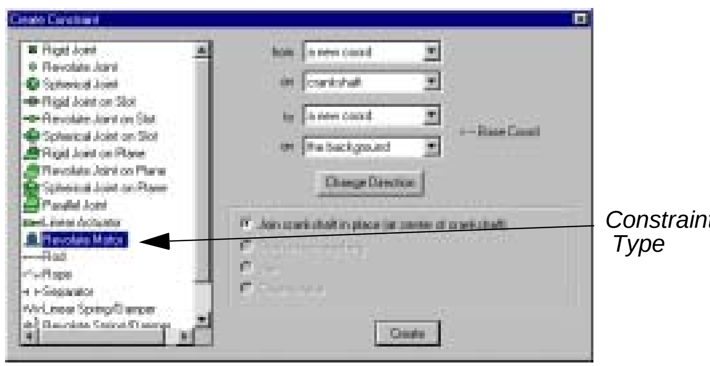
You will now join the crankshaft to the background.

1. Select the crankshaft, if it is not already selected.



2. Click the **Create Constraint** button in the **Edit** toolbar, or choose **Constraint** from the **Insert** menu.

Figure 7-9
Create Constraint Window



3. In the **Create Constraint** window, 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.

The bottom half of the window provides only one option, which 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.”

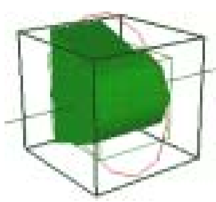
MSC.visualNastran Desktop provides a range of options depending on the constraint type and objects you select before creating constraints. Please refer to the **Online Help** for more information.

---

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:

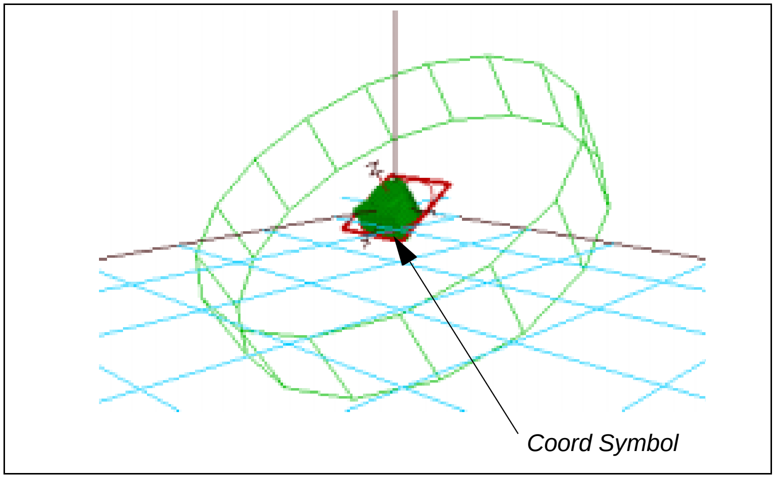


- The motor icon has a green cube around it (when it is selected) with an axis that shows the plane in which the motor allows rotation.
- MSC.visualNastran Desktop created a red disk-shaped object called a **Coord**. The model actually has two Coords at the same global position, where one is attached to the crankshaft and the other to the background. (Figure 7-10 displays only one of them, because the 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.

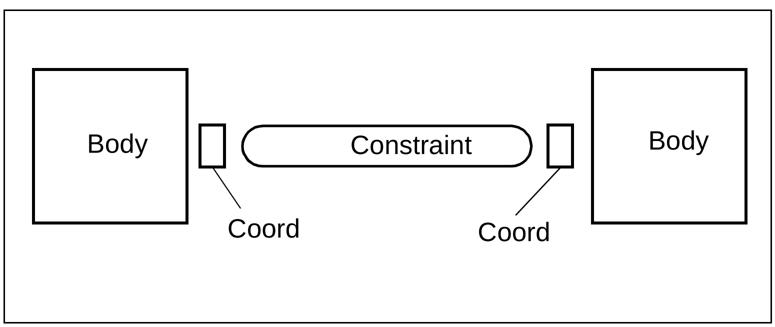
WHAT IS A COORD? Coords are basic objects to construct constraints in MSC.visualNastran Desktop. Coord is an abbreviation for Cartesian Coordinate system. A Coord consists of a point — the origin of the coordinate system and three coordinate axes (linear) x, y, and z — along which measurements are calculated. 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 7-10
Coord and Motor



All constraints (except for external forces and torques) in MSC.visualNastran Desktop consist of a pair of Coords, each of which are attached to a body or the background. In essence, a Coord serves as an intermediary between a body and the constraint, as illustrated in Figure 7-11.

Figure 7-11
Concept: Construction of a Constraint



For some constraints such as translational spring/dampers, ropes, and separators, the two Coords associated with the constraint are endpoints. For other kinds of constraints, such as a revolute motor, MSC.visualNastran Desktop uses both the position and the orientation of each Coord associated with the constraint.

5. Double-click the motor icon located at the center of the crankshaft or in the **Object List**. The **Properties** window appears.

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

Figure 7-12
Properties Window (Constraint Page)

As Figure 7-12 illustrates, each constraint shows you the free rotational or translational directions. In the case of this motor, the **Properties** window shows that the motor allows rotation about the z-axis.

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

Figure 7-13
Properties Window (Motor Page)

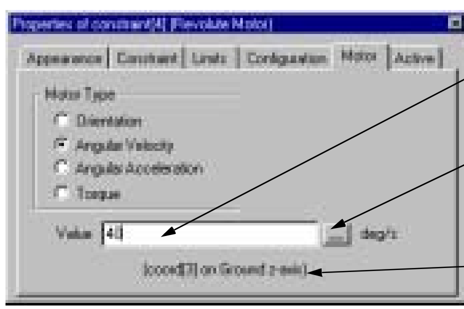


Figure 7-13 illustrates that the angular velocity specification uses the z-axis of the Coord on Ground, one of the two Coords associated with the motor.

NOTE: By modifying the position and the orientation of Coords, you can create a full range of constraints. For more information, please see the **Online Help**.

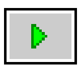
Specify the Motor Function

You will specify the motor's performance characteristics in its **Properties** window.

1. On the **Motor** page of the **Properties** window, select **Angular Velocity** and type 40 in the value field.

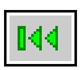
This specification results in a motor that rotates at 40 degrees per second. When you run the simulation, MSC.visualNastran Desktop will provide as much torque as necessary to maintain the specified angular velocity.

2. Close the **Properties** window.



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

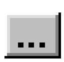
Note that the crankshaft rotates counterclockwise (if seen from the positive z). Also, note that the frame indicator at the bottom shows the current frame being calculated.



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

Before constructing 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.

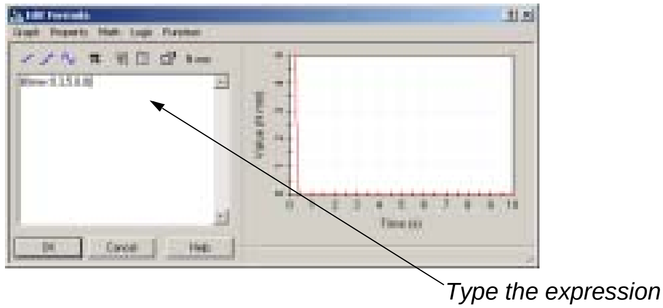
5. Double-click the motor icon. The **Properties** window shows the motor characteristics.



6. Under the **Motor** tab, select **Torque** as **Motor Type**, and click the **Formula** button.

The Formula Editor dialog appears as shown in Figure 7-14.

Figure 7-14
Entering an Expression in the Formula Editor Dialog



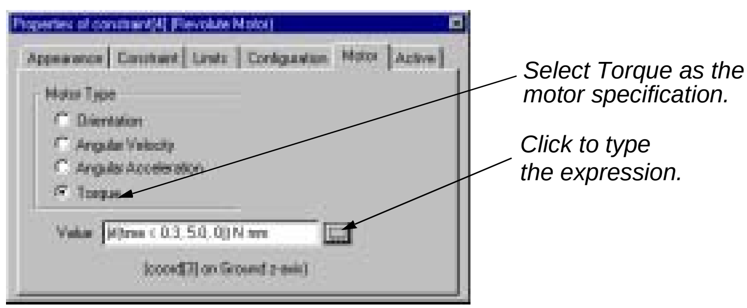
7. Type the following expression in the **Formula** field, as shown in Figure 7-14:

   if(time < 0.3, 5.0, 0)

MSC.visualNastran Desktop will automatically apply the unit system in which you are working.

8. Click **OK**. The expression with units appears in the **Value** text region as shown in Figure 7-15.

Figure 7-15
Specifying Motor Properties



The expression is an example of MSC.visualNastran Desktop **Formula Language**, and this expression is interpreted as follows:

If the elapsed time is less than 0.3 seconds, apply the torque of 5.0 N-mm; otherwise, apply no torque.

For more information on the formula language, please refer to the **Online Help**.

9. Close the **Properties** window.

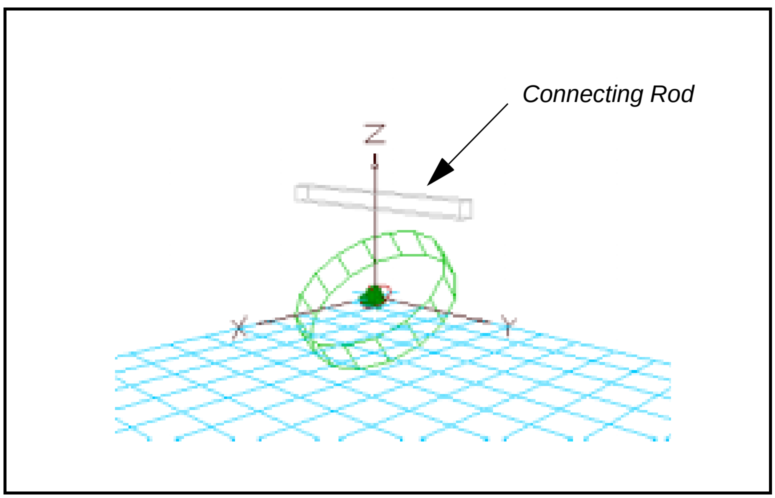
Create the Connecting Rod

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.)

1. Click the **Box** tool in the **Sketch** toolbar.
2. Click once anywhere on the Edit Grid and press the **Enter** key on the keyboard.
3. Open the **Properties** window by double-clicking the box.
4. Click the Position (**Pos**) tab.
5. Type the position as (x, y, z) = (0, 0, 30).
6. Click the **Box** tab.
7. Enter the dimensions: (Width, Length, Height) = (5, 70, 5).

The connecting rod is sized and positioned accordingly, as shown in Figure 7-16.

Figure 7-16
Connecting Rod



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

8. Click the **Appearance** tab of the **Properties** window.

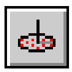
9. In the name field type the name con_rod and press **Enter**.

Again, notice how the hoover tool and status bar identify the connecting rod and the crankshaft as you move the mouse over the bodies.

Create Attachment Coords

To attach the connecting rod to the crankshaft, you need to create a Coord on the connecting rod, a Coord on the crankshaft, and then create a constraint between the two newly constructed Coords.

You will create one Coord on the crankshaft, another Coord on the connecting rod, and create a spherical joint based on these two Coords.



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

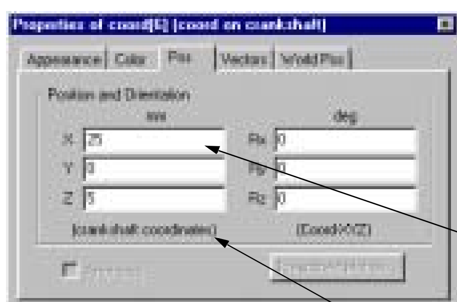
2. Click anywhere on the top surface of the crankshaft; you will position the Coord precisely in the later steps.

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

3. Double-click the Coord to view its **Properties** window.

4. Click the Position (**Pos**) tab.

Figure 7-17
Properties Window (Position Page)



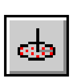
Type the relative position here.

Indicates Attachment to the crankshaft.

5. In the region labeled **Position and Orientation**, enter position at (x, y, z) = (25, 0, 5).

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

6. Close the **Properties** window.



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

8. Click anywhere on the connecting rod. (You will position the Coord precisely in the later steps.)

9. Double-click the Coord and click the **Pos** tab of the **Properties** window.

10. In the region labeled **Position and Orientation**, enter position (x, y, z) = (0, 35, 0), and orientation (Rx, Ry, Rz) = (-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 position as well as orientation because the Coord may have attached to the side surface of the connecting rod and not have the correct orientation.

11. Close the **Properties** window.

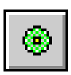
Create a Spherical Joint

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. While pressing the **Control** key, select the Coord located at the perimeter of the crankshaft.

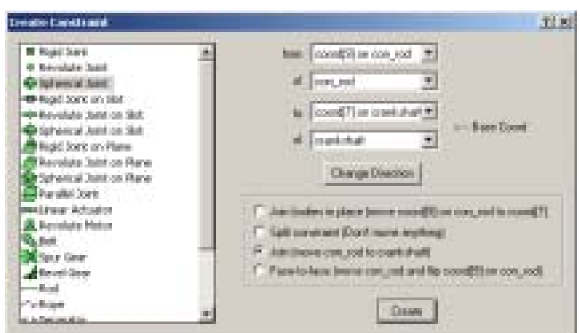
Both Coords are selected.



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

The **Create Constraint** window appears, as shown in Figure 7-18. 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 7-18
Create Constraint Window



4. Choose **Spherical Joint** as the constraint type.

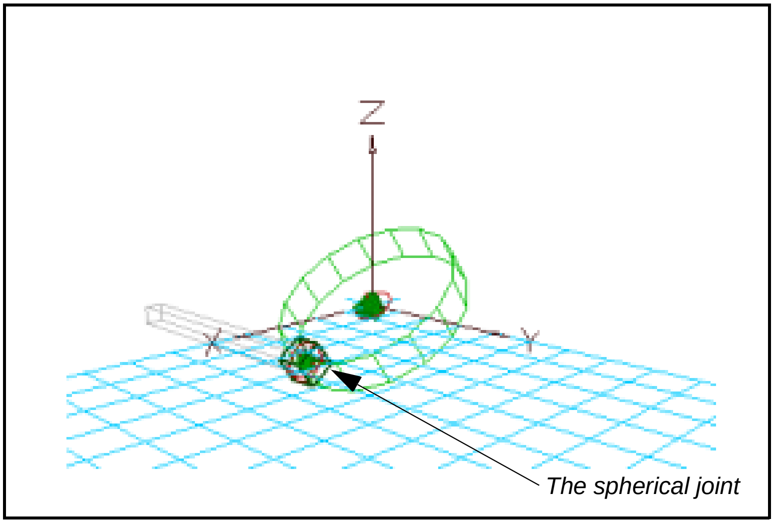
As discussed earlier, you should read the first few lines of the window as a complete sentence: “Create a spherical joint from coord[x] on con_rod to coord[x] on crankshaft.” MSC.visualNastran Desktop automatically selects the remaining options depending on the objects you selected.

5. Click the option **Join (move con_rod to crankshaft)**. See Figure 7-18.

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, as shown in Figure 7-19.

Figure 7-19
Bodies Connected with a Spherical Joint



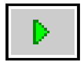
Test Your Model

As you create a simulation model, you should periodically check to ensure that the model’s components are working properly.

So far, you have completed the following:

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

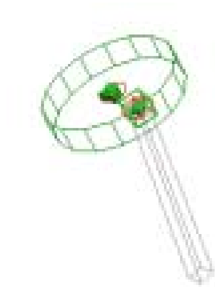
To test your model, simply run the simulation and visually verify the motion of the connecting rod.

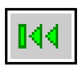


1. Click the **Run** button in the **Tape Player Control**.

After the simulation runs for a while, the connecting rod swings out as the crankshaft rotates, as shown in Figure 7-20.

Figure 7-20
Connecting Rod Swung Out by the Crankshaft





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

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 the section **Attach a Motor to the Crankshaft**.
- If the connecting rod is falling downward, you probably have failed to attach it to the crankshaft. Review **Create Attachment Coords**.

Validating a large model is not always an easy task. As discussed in this section, you should periodically verify your model's assembly before proceeding.

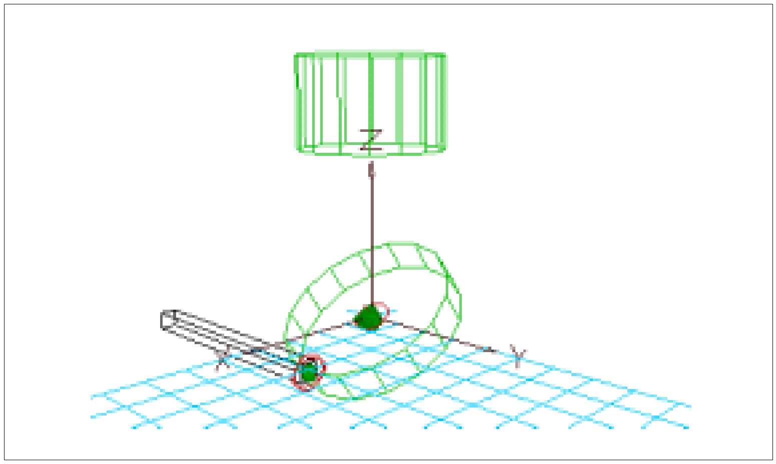
Create the Piston

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 List**.
4. Click the Position (**Pos**) tab. Enter the position (x, y, z) = (0, 0, 60).
5. Click the Cylinder tab. Enter the geometry (Radius, Height) = (20, 25).

The piston is sized accordingly and positioned on the screen (see Figure 7-21).

Figure 7-21
Piston Sized and Positioned



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

Next, you will assign a custom name to the piston.

6. Click the **Appearance** tab.

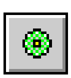
7. In the name field, type piston.

8. Close the **Properties** window.

Attach the 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.



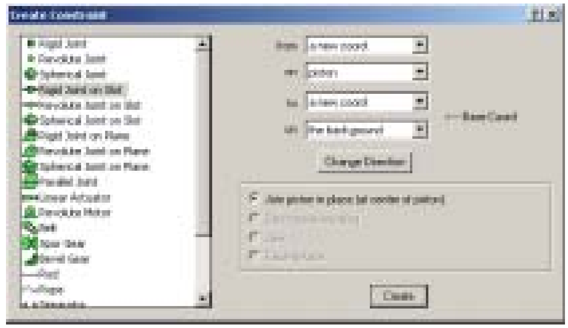
2. From the **Insert** menu, select **Constraint**.

The **Create Constraint** dialog appears, as shown in Figure 7-22.

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

The window presents a single option: **Join piston in place (at center of piston).**

Figure 7-22
Create Constraint Window

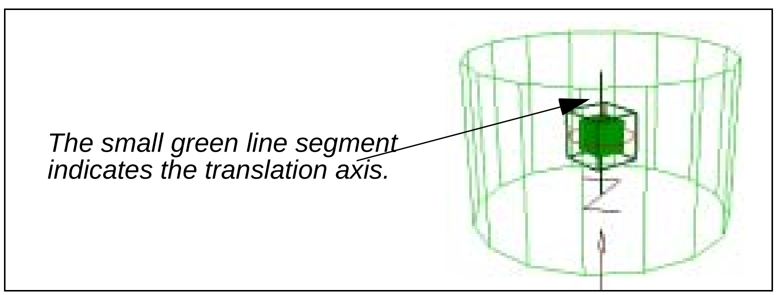


Since you selected a single body (you did not select Coords), MSC.visualNastran Desktop presents the only reasonable join option. MSC.visualNastran Desktop will create two Coords (one attached to the piston, the other attached to the background), and create a rigid joint on slot to connect the body to the background. One of the Coords will be attached to the body at its origin (coordinate origin).

4. Click the **Create** button in the window.

A rigid joint on slot icon appears at the center of the piston (see Figure 7-23). You may want to zoom in with the zoom tool to obtain a better view.

Figure 7-23
Rigid Joint on Slot Created

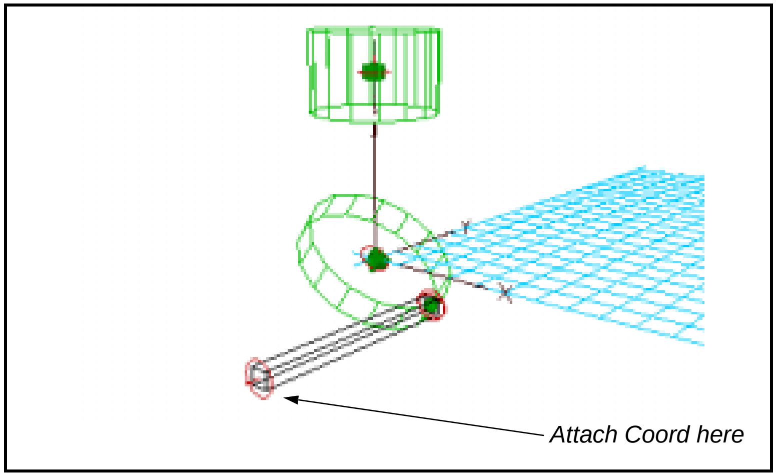


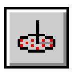
Note that the small green line segment indicates that this slot joint allows single degree of freedom (translation in the z direction). The piston can slide "up-and-down;" this is the desired behavior.

Create an Attachment Coord on the Connecting Rod

1. Use the **Rotate Around** tool so that you can see the free end of the connecting rod, as shown in Figure 7-24.

Figure 7-24
Attaching a Coord
to the Connecting Rod





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

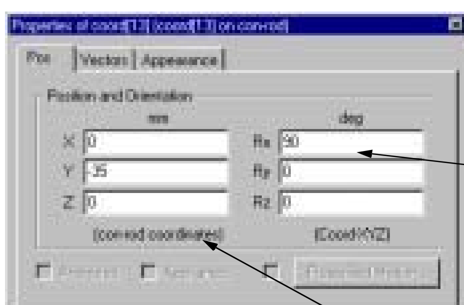
3. Click at the free end of the connecting rod.

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 to view its Properties window.

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

Figure 7-25
Properties for Coord Attached
to the Connecting Rod

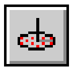


6. Close the **Properties** window.

Create an Attachment Coord on the Piston

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

1. Use the **Rotate Around** tool to position the piston so that its bottom surface is facing you.



2. Click the **Coord** tool on the **Sketch** toolbar.
3. Click at the bottom surface of the piston, near the center.

Do not worry if you attached the Coord on a side surface of the piston; you will correct it in the next step.

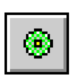
4. Double-click the Coord to display its Properties window.
5. Click the Position (**Pos**) tab, and verify that the Coord is attached to the piston, and the configuration is given as (X, Y, Z) = (0, 0, -12.5), and (Rx, Ry, Rz) = (0, 180, 0). If any values are different, correct them.
6. Close the **Properties** window.

Create a Spherical Joint

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

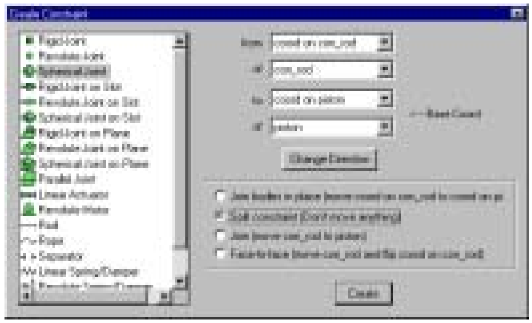
1. Select the Coord attached to the connecting rod.
2. Hold down the **Control** key, and select the Coord attached to the piston.

If you have trouble selecting these Coords in the modeling window, try selecting them in the **Object List**.



3. Click the **Join/Create Constraint** button on the Edit toolbar. The **Create Constraint** dialog appears (Figure 7-26).

Figure 7-26
Create Constraint Window

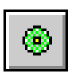


4. Select **Spherical Joint** as the constraint type, and click the option labeled **Split constraint (Don't move anything)**.

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.

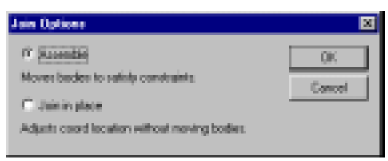
A line appears between the two Coords, and a split spherical joint icon appears in the **Object List**. At this point, the constraint is defined, but the bodies are not assembled.



6. With the new constraint is selected, right-click and select **Join**.

A dialog appears that offers you the choice of moving the bodies to satisfy the constraints or adjusting the coord location without moving the bodies, as shown in Figure 7-27.

Figure 7-27
Join Options Dialog



7. Click the **Assemble** radio button. Click **OK**.

MSC.visualNastran Desktop will attempt to assemble the bodies using its quick kinematic assembly algorithm, but at times, the relationships are too complex. When this occurs, MSC.visualNastran Desktop presents an **Assembly Error** dialog and you have the option of using a slower, dynamic assembly algorithm.

Kinematic Assembly Failure

When the Assembly Error dialog appears, you can choose which method you want to use to resolve the problem:

- Click the **Retry** button to attempt assembly using a dynamic assembly algorithm.
- Click **Cancel** to close the dialog, then use the **Move** tool to position the coords that you want to join closer together.

Use the Dynamic Assembly Algorithm

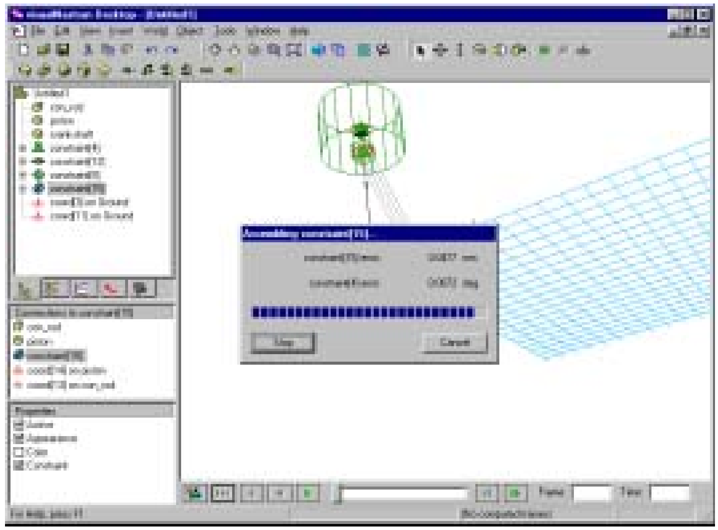
Click the **Retry** button in the **Assembly Error** dialog to have MSC.visualNastran Desktop assemble the bodies using a slower, iterative, dynamic assembly algorithm.

A progress dialog appears to show the assembly is in process, as shown in Figure 7-28. At the same time, the bodies in the model undergo motion to show you the assembly process.

NOTE: If you do not see the progress dialog, you may have accidentally split another constraint by mistake. In that case, choose **Select All** in the **Edit** menu, and click the **Join/Create Constraint** button in the **Edit** toolbar. MSC.visualNastran Desktop will attempt to resolve all constraints simultaneously.

In the Join process, MSC.visualNastran Desktop 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 MSC.visualNastran Desktop soon finds a converging solution that satisfies all the constraint conditions.

Figure 7-28
Joining Process

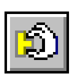


After a short while, the progress bar reaches the completion, and the progress dialog disappears. At this point the model is fully assembled, as shown in Figure 7-29, and you are ready to run the simulation.

Move the Coords Closer Together

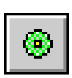
Another approach to dealing with assembly errors is to use the **Move** tool to position the Coords that you want to join closer together. Follow these steps to resolve the assembly problem using this method:

1. Click the **Undo** tool. This returns the spherical joint to its split state.



2. Select the **Move** tool on the **Edit** toolbar.

3. Drag the connecting rod to reposition it so that the Coord on its end is near the Coord on the piston.

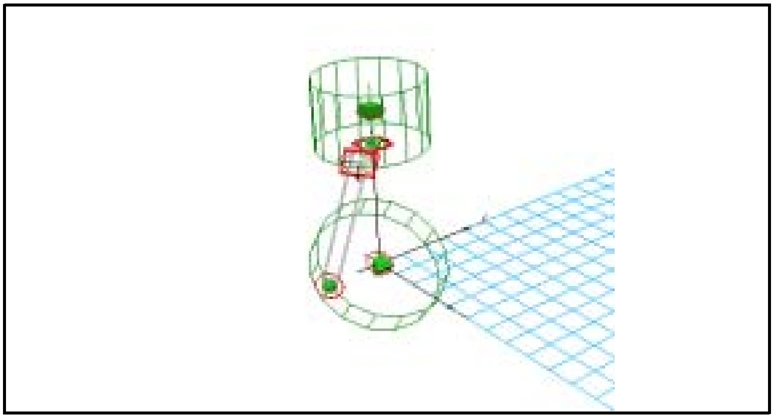


4. With the new constraint is selected, right-click and select **Join**.

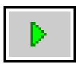
5. Click the **Assemble** radio button. Click **OK**.

MSC.visualNastran Desktop attempts to assemble the bodies again using its quick assembly algorithm. If the **Assembly Error** dialog appears again, click the **Retry** button to use the dynamic assembly algorithm.

Figure 7-29
Fully Assembled Piston Engine

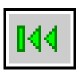


Run the Simulation



1. Click the **Run** button in 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 modeling 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** tools to obtain different view angles and repeat the simulation. Note that the simulation runs much faster the second time around, because MSC.visualNastran Desktop stores the simulation time history after the first calculations. The history remains in memory until you change the initial conditions.

Take Measurements

The simulation model you have just created serves as a powerful visualization tool to observe the general motion of your mechanism. MSC.visualNastran Desktop allows you to take numerical measurements on various aspects of your simulation.

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

Now, you will create a meter to measure the angular velocity of the crankshaft:

1. Select the crankshaft **cylinder** in the document window or the **Object List**.

2. Choose **Meter** in the **Insert** menu and then choose **Angular Velocity** in the submenu.

A meter with the title **Angular Velocity of crankshaft** appears as a separate window, as shown in Figure 7-30.

3. In the **Tiling Options** dialog, select **Tile Vertically** and then click **OK**.

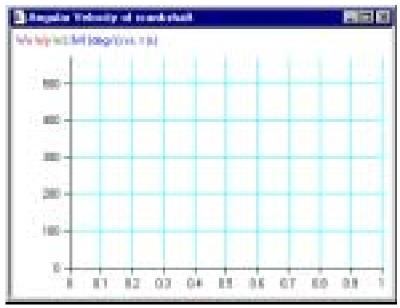


Figure 7-30
Angular Velocity Meter

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

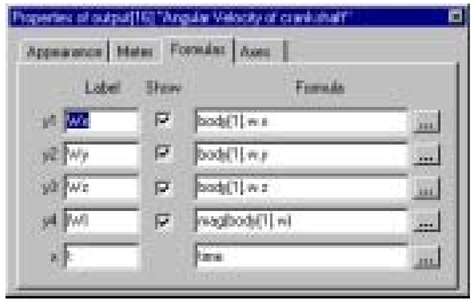
By default, the angular velocity meter shows plots for angular velocities in x-, y-, and z-axes as well as the magnitude.

In this simulation, the crankshaft rotates with its z-axis fixed, so you are only interested in observing w_z, or the z-component of the angular velocity.

4. Double-click the meter in the graph area. The **Properties** window for the meter is displayed, as shown in Figure 7-31.

5. Click the **Formulas** tab if not already selected.

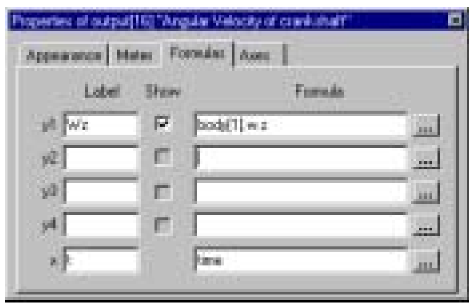
Figure 7-31
Properties Window for Meter



6. Delete Wx, Wy, and |W| from the **Label** column.

As you remove items from the Label column, the formula is also deleted, as shown in Figure 7-32.

Figure 7-32
Properties Window for Meter after Removal of Unwanted Plots



7. Close the **Properties** window.

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

8. Select the spherical joint between the connecting rod and the piston in the **Object List**.

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

9. Choose **Meter** in the **Insert** menu and then choose **Constraint Force...** in the submenu.

A dialog appears as shown in Figure 7-33.

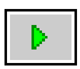
Figure 7-33
Meter Dialog for Constraint Force

10. Select the options as above and click **OK**.

11. In the **Tiling Options** dialog, select **Tile Vertically** and then click **OK**. Another meter window appears.

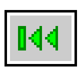
You are now ready to run the simulation to take measurements.

To run the simulation:



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

The simulation begins to show the model in motion and the meters display the data. Note that the meters rescale the graph axes automatically as the data grows in magnitude.



13. Run the simulation until several rotation cycles are completed, then click the **Reset** button on the toolbar.

14. Repeat the simulation.

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

Improve the Model

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

To visualize the spike in the joint reaction force:

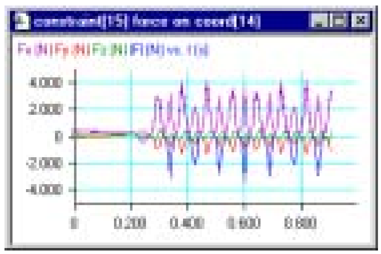
1. Select the force meter. Then choose **Properties** in the **Edit** menu (or double-click the force meter window).

The **Properties** window displays the meter's attributes.

2. Click the **Meter** tab, then choose the **Graph** radio button.

The digital meter changes to the graph format (if not already in graph format) to show the history. Note the series of spikes seen in the force meter, as shown in Figure 7-34.

Figure 7-34
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 crankshaft motor to smooth the rotation.

3. Double-click the motor icon located at the center of the crankshaft. The **Properties** window appears to show the motor specification.

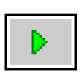
4. Click the **Motor** tab, if it is not already selected.

5. Choose the **Angular Velocity** radio button, then enter the following expression in the **Value** field.

   0.2*(3600 deg/s-body[1].w.z)

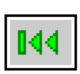
The expression implements a rudimentary proportional control system which monitors the angular velocity of the crankshaft (represented as body[1] in the formula). Note that, according to this expression, the motor is capable of applying torque in either direction.

6. Close the window.



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

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



8. Run the simulation until several rotation cycles are completed, then click the **Reset** button in the **Tape Player Control**.

You can export the meter data to a text file for further analysis by another application program. For more information, please see the **Online Help**.