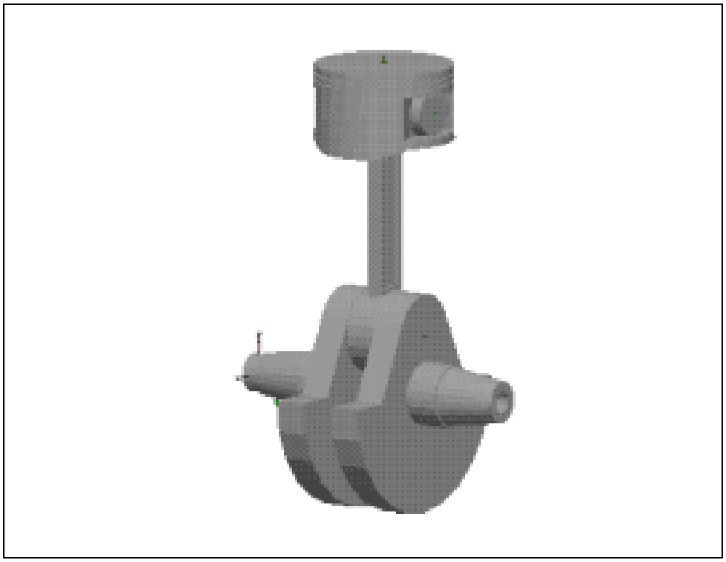
CHAPTER 5

Exploring CAD Integration and Associativity



The following exercises illustrate how you can add motion to a model that has been created in a CAD software package. You can create CAD assemblies in such a way that they map into MSC.visualNastran Desktop very cleanly. When you design assemblies with “motion in mind,” they can be imported easily and their motion simulated immediately.

In these exercises, you will add motion to an assembly that has been fitted together with no regard to motion. Although you will have to clean up a few constraints in the imported CAD model, MSC.visualNastran Desktop makes it easy to simulate the motion of the assembly.

To complete any of the exercises, you must have one of the following software installed on your computer in addition to MSC.visualNastran Desktop:

- Autodesk® Inventor
- Mechanical Desktop®
- Pro/ENGINEER®
- Solid Edge™
- SolidWorks®

You can follow the steps in this exercise using any of the supported CAD packages — Autodesk Inventor, Mechanical Desktop, Pro/ENGINEER, Solid Edge, or SolidWorks. Go to the section for your CAD package and follow the steps. Look for the icon that identifies your CAD package.

Exercise 5.1 Exploring Mechanical Desktop Integration and Associativity

Objectives

In this exercise you will learn to

- Export a CAD model from Mechanical Desktop.
- Verify and modify your simulation model using the Constraint Navigator.
- Change a constraint and add new constraints.
- Use CAD Associativity.
- Organize your design using subassemblies.

Support Files

- Tutorials\Chapter 05\Exercise 5.1-MDT\Piston.dwg

Exporting a CAD Model — Mechanical Desktop

1. Launch the Mechanical Desktop program.

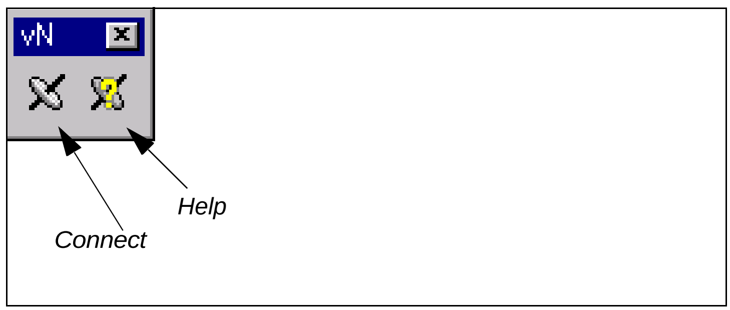
Note that a **visualNastran** menu appears in the Mechanical Desktop menu bar, as shown in Figure 5-1.

Figure 5-1
MSC.visualNastran Menu in Mechanical Desktop

The **Solid Representation** menu item displays a dialog that lets you choose faceted or ACIS geometry. See the online Help for more information on these choices, but you may ignore them for this exercise.

A **vN** toolbar also appears in the Mechanical Desktop window, as shown in Figure 5-2.

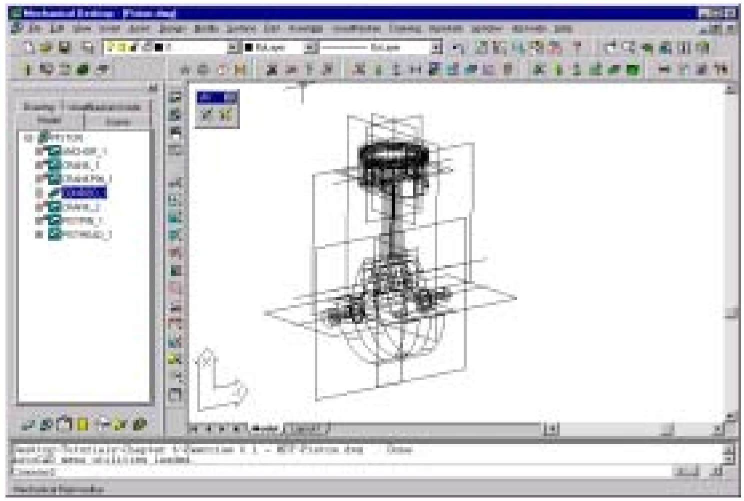
Figure 5-2
vN Toolbar in Mechanical Desktop



2. Open the file **Piston.dwg** located in your **Program Files\MSC.visualNastran Desktop\Tutorials\Chapter 05\Exercise 5.1-MDT** folder.

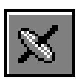
The CAD model of the piston assembly is displayed as shown in Figure 5-3.

Figure 5-3
Mechanical Desktop CAD Model of Piston Assembly



3. Choose **Units** from the MSC.visualNastran menu, and change units to **millimeters**.

This exports the model to scale in MSC.visualNastran Desktop.



4. Choose **Connect** in the MSC.visualNastran menu, or click the **Gyroscope** button in the **vN** toolbar.

MSC.visualNastran Desktop for Mechanical Desktop prompts you to view a tutorial overview. This happens only the very first time you Connect. When you dismiss the online tutorial and click again on the gyroscope button, MSC.visualNastran Desktop for Mechanical Desktop maps the assembly components and constraints into MSC.visualNastran Desktop bodies and joints, and creates a new, linked model named **Piston.wm3** in the directory where the CAD files are located. As MSC.visualNastran Desktop translates the geometry, the progress is displayed in the **Preparing Simulation** dialog.

When the export is complete, the MSC.visualNastran Desktop program opens and displays the **CAD Associativity** dialog, listing the MSC.visualNastran Desktop objects that are associated with objects in the Mechanical Desktop model as shown in Figure 5-4.

Figure 5-4
CAD Associativity Dialog

5. Click **OK** to close the **CAD Associativity** dialog.

MSC.visualNastran Desktop prompts you to run the Constraint Navigator.

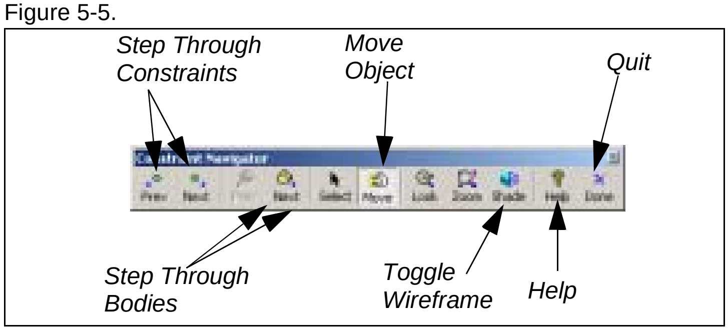
6. Click **Yes** to run the Constraint Navigator.

The linked model appears with the Constraint Navigator in the modeling window.

Using the Constraint Navigator — Mechanical Desktop

The Constraint Navigator allows you to examine relationships among bodies, subassemblies, and constraints so that you can verify and modify your simulation model. The exercise in this section shows you how to use the Constraint Navigator controls to check your exported model. Refer to Figure 5-5.

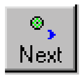
Figure 5-5
Constraint Navigator



When MSC.visualNastran Desktop exports the CAD model, it creates joints between parts of the assembly. The types of joints created depends on the constraints and geometry of the CAD model.

If the joints that MSC.visualNastran Desktop creates are not given the appropriate degrees of freedom, the parts cannot move appropriately when you run the simulation. To correct this problem, you should verify and adjust the joints between the parts of the assembly as necessary to correctly specify the motion desired.

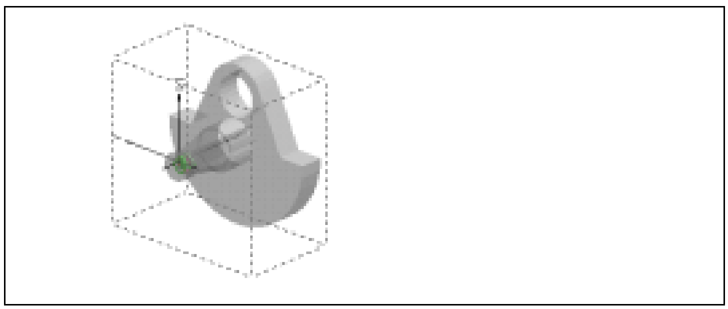
In the steps that follow, you will adjust the joints to give each the appropriate degrees of freedom.

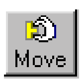


1. Click the **Next Constraint** button.

A revolute joint, **constraint[66]**, is isolated as shown in Figure 5-6.

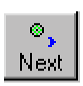
Figure 5-6
Isolated Constraint





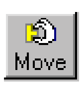
2. Click the **Move** button in the Constraint Navigator, click over the Crank, and drag the mouse.

This tests the kind of movement allowed by the constraint. The constraint is OK.



3. Click the **Next Constraint** button.

A revolute joint, **constraint[68]**, is isolated.



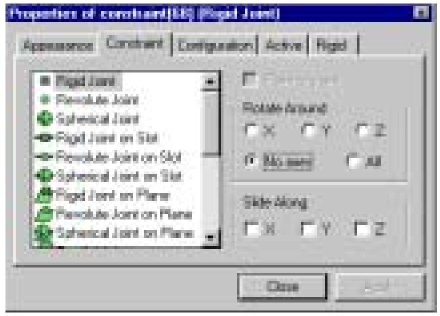
4. Use the Move tool to test the constraint by first moving the crank and then moving the crank pin.

The model requires a rigid connection between **CRANKPIN_1** and **CRANK_1**. That means changing this revolute joint into a rigid joint.

5. Double-click **constraint[68]** in the **Connections List**.

The **Properties** window appears as shown in Figure 5-7.

Figure 5-7
Properties Window (Constraint Page)

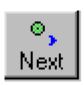


6. Click the **Rigid Joint** on the **Constraint** page of the **Properties** window.

7. Under the **Rigid** tab of the **Properties** window, accept the **Optimized** setting.

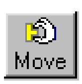
This combines the mass properties of the two connected bodies and computes the motion as if they were one body. Unless the constraint force is to be measured, optimized rigid joints should always be used because they improve simulation computational speeds.

8. Close the **Properties** window.



9. Click the **Next Constraint** button.

A rigid joint on slot, **constraint[70]**, is isolated.



10. Use the **Move** tool to test the constraint.

This model requires deleting this constraint and its two Coords.

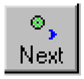
11. Hold down the Control key, and in the **Connections List** select the two Coords connected to constraint[70].

The rigid joint, **constraint[70]**, as well as its two Coords, should be selected when you are done.

12. Press the delete key.

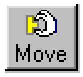
The **MSC.visualNastran Desktop** dialog appears to inform you that you are deleting an object linked to a CAD assembly.

13. Click **OK**.



14. Click the **Next Constraint** button.

A revolute joint on slot, **constraint[72]**, is isolated.



15. Use the **Move** tool to test the constraint.

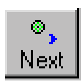
The model requires changing this constraint to a revolute joint.

16. Double-click **constraint[72]** in the **Connections List**.

The **Properties** window appears. Click the **Constraint** tab if the **Constraint** page is not already displayed.

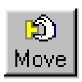
17. Select **Revolute Joint** in the **Constraint** page of the **Properties** window to change the constraint.

18. Close the **Properties** window.



19. Click the **Next Constraint** button.

A revolute joint, **constraint[74]**, is isolated.



20. Use the **Move** tool to test the constraint.

The model requires changing this constraint to an optimized rigid joint.

21. Double-click **constraint[74]** in the **Connections List**.

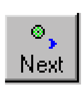
The **Properties** window appears. Click the **Constraint** tab if the **Constraint** page is not already displayed.

22. Select **Rigid Joint** in the **Constraint** page of the **Properties** window.

This changes the constraint.

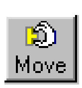
23. Under the **Rigid** tab of the **Properties** window, accept the **Optimized** setting.

24. Close the **Properties** window.



25. Click the **Next Constraint** button.

A revolute joint on plane, **constraint[76]**, is isolated.



26. Use the **Move** tool to test the constraint.

The model requires deleting this constraint and its two Coords.

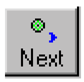
27. Hold down the Control key and select in the **Connections List** the two Coords connected to **constraint[76]**.

The revolute joint on plane, **constraint[76]**, as well as its two Coords should be selected when you are done.

28. Press the delete key.

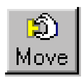
The **MSC.visualNastran Desktop** dialog appears to inform you that you are deleting an object linked to a CAD assembly.

29. Click **OK**.



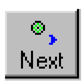
30. Click the **Next Constraint** button.

A revolute joint, **constraint[78]**, is isolated.



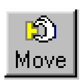
31. Use the **Move** tool to test the constraint.

This constraint is OK.



32. Click the **Next Constraint** button.

A revolute joint, **constraint[80]**, is isolated.



33. Use the **Move** tool to test the constraint.

The model requires changing this constraint to an optimized rigid joint.

34. Double-click **constraint[80]** in the **Connections List**.

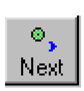
The **Properties** window appears. Click the **Constraint** tab if the **Constraint** page is not already displayed.

35. Select **Rigid Joint** in the **Constraint** page of the **Properties** window.

This changes the constraint.

36. Under the **Rigid tab** of the **Properties** window, accept the **Optimized** setting.

37. Close the **Properties** window.

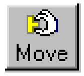


38. Click the **Next Constraint** button.

A revolute joint on plane, **constraint[82]**, is isolated.

39. Choose **Look At** from the **View** menu, then choose **Front View** from the **Look At** submenu.

This provides a better view of the constraint.



40. Use the **Move** tool to test the constraint.

The model requires deleting the constraint and its two Coords.

41. Hold down the Control key and select in the **Connections List** the two Coords connected to constraint[82].

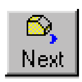
The revolute joint on plane, **constraint[82]**, as well as its two Coords should be selected when you are done.

42. Press the delete key.

The **MSC.visualNastran Desktop** dialog appears to inform you that you are deleting an object linked to a CAD assembly.

43. Click **OK**.

44. Choose **Previous View** in the **View** menu.



45. Next, you can step through each body by clicking on the **Next** and **Previous Body** buttons in the Constraint Navigator to verify body motion.

46. Click **Done** to close the Constraint Navigator.

Anchored Assemblies — Mechanical Desktop

Based on the constraints and geometry in the Mechanical Desktop CAD model, MSC.visualNastran Desktop may anchor some parts to the background when it creates the linked model. The first step in simulating the motion of the piston assembly is to make sure that it is properly anchored.

- If no bodies in the assembly are anchored, the piston is floating in space and gravity will cause it to fall when you run the simulation.
- If too many bodies are anchored, the piston won't be able to move appropriately when you run the simulation.

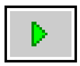
In this piston assembly, no bodies are anchored. To prevent the body from falling due to gravity, we must anchor, or fix, at least one body to the background.

1. In the **Object List**, click the **+** sign to the left of **ANCHOR_1** and select **ANCHOR**.

In the workspace, the anchor appears selected.

2. Right-click the anchor in the piston model and select **Fixed** from the pop-up menu.

The anchor is now fixed to the background.



3. Click the **Run** button in MSC.visualNastran Desktop's **Tape Player Control**.

There is no movement. This is because gravity is acting in the "wrong" direction. By default, gravity acts on the z axis in MSC.visualNastran Desktop, specifically in the "-z" direction. If the z-axis was not in an upward orientation when the CAD assembly was created, then the direction of gravity must be changed before MSC.visualNastran Desktop can properly simulate the motion.

4. Choose **Simulation Settings** in the **World** menu.

The **Simulation Settings** window appears.

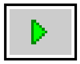
5. Click **Gravity** under **Simulation Settings** in the left pane.

The **Gravity** page appears as shown in Figure 5-8.

Figure 5-8
Settings Window (Gravity Page)

6. Click the "-X" direction.

7. Click **Close**.

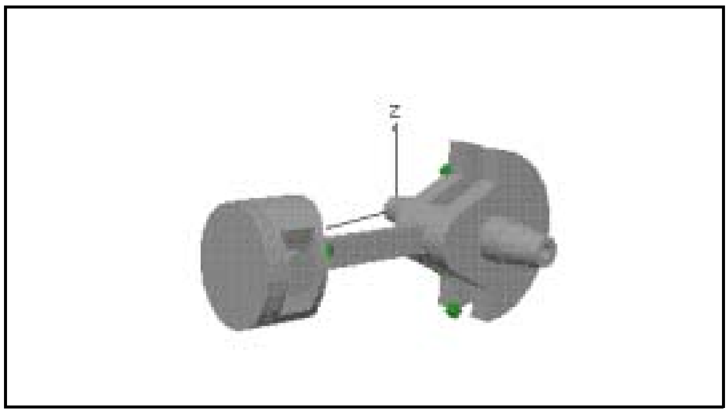


8. Click the **Run** button in MSC.visualNastran Desktop's Tape Player Control.

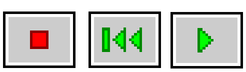
MSC.visualNastran Desktop begins to simulate the motion of the model. The piston assembly balances at first, then the piston head falls under the effect of gravity. Since the piston is almost in equilibrium, you will not see any movement at first. Allow the simulation to run 150 frames.

The geometry of the assembly causes the piston head to swing like a pendulum around the crank pin, which is held in place by the anchor, as shown in Figure 5-9.

Figure 5-9
Piston Head Swinging Around the Crank Pin Like a Pendulum



Since this is the first time the simulation is being run, MSC.visualNastran Desktop calculates the dynamics and stores the data.



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

Depending on the speed of your computer, the animation may be faster this time because the history has already been calculated.

Adding a Motor — Mechanical Desktop

Currently, the piston assembly is moving only in response to the effects of gravity. In this step, you will change the joint between the anchor and left side of the crank to a motor to simulate the force that drives the motion of the piston.

1. In the **Object List**, click the **+** sign to the left of **ANCHOR_1** and select **ANCHOR**.

The **Connections List** displays a list of the objects connected to this body.

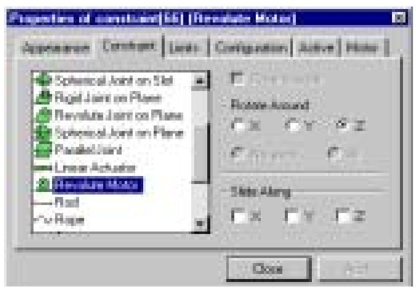
2. Double-click **constraint[66]**, the revolute joint that connects **ANCHOR** to **CRANK**.

The **Properties** window for **constraint [66]** appears.

3. If necessary, click the **Constraint** tab in the **Properties** window. Then, select **Revolute Motor** for the list of available joint types, as shown in Figure 5-10.

The revolute joint is changed into a revolute motor.

Figure 5-10
Properties Window (Constraint Page) for Revolute Motor



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

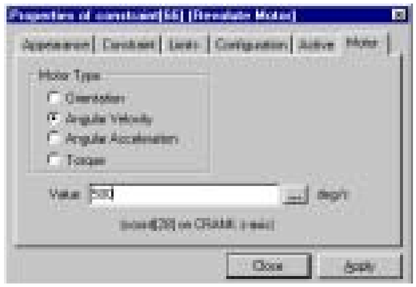
This brings up the motor characteristics page.

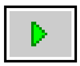
5. Accept **Angular Velocity** in the list of Motor Types.

6. Enter 500 as the value for angular velocity of this Revolute Motor, as shown in Figure 5-11.

You can close the **Properties** window now.

Figure 5-11
Properties WIndow (Motor Page)

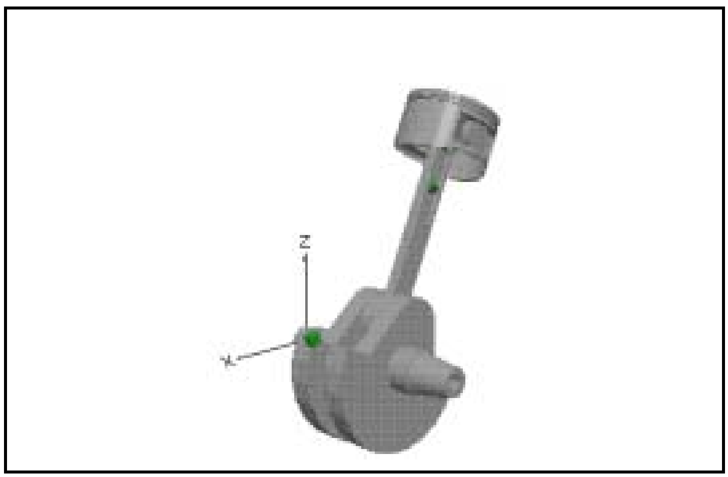


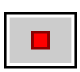


7. Click the **Run** button in MSC.visualNastran Desktop's **Tape Player Control**.

The piston assembly rotates all the way around the crank pin, as shown in Figure 5-12. It no longer swings like a pendulum, because the motor provides the driving force to keep the piston rotating.

Figure 5-12
Piston Head Rotating All the Way Around the Crank Pin

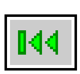




8. Click the **Stop** button.

Adjusting the Joints — Mechanical Desktop

In this step, you will add a new constraint that limits the motion of the piston head, so that it can only move vertically.



1. Reset the simulation by clicking the **Reset** button.

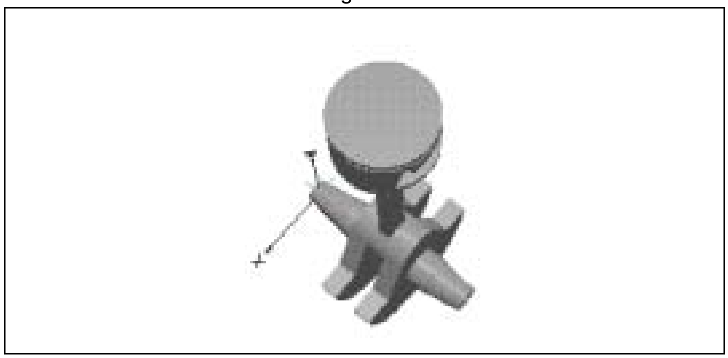
The piston assembly returns to its starting position.

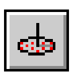
2. Press “X” on the keyboard, or choose **Look At** in the **View** menu and choose **Front View** in the submenu.

3. Click the **Rotate Around** tool in the **View** toolbar and rotate the model so that you can see the top of the piston head and the rest of the drawing.

Your view should be similar to Figure 5-13.

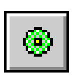
Figure 5-13
Top of the Piston Head





4. Click the **Coord** tool in the **Edit** toolbar, then place a Coord on the top surface of the piston head, near the center.

Blue lines appear around the piston head and cranks. This feature recognition helps guide placement of the Coord. When you place the Coord on the blue circular outline on top of the piston head, it automatically snaps to the center of the piston head. The z-axis of the Coord should point upward and its red outline surrounds the top center of the piston head.



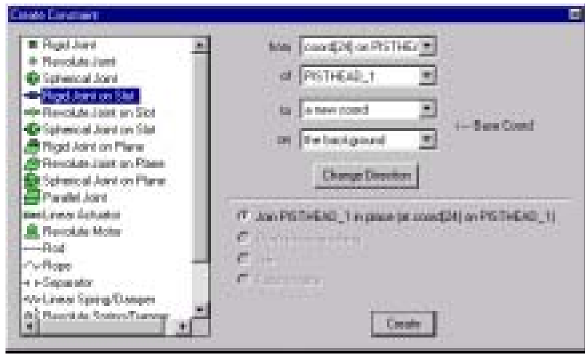
5. Make sure that the Coord that you just created is selected. Then click the **Join/Create Constraint** button in the **Edit** toolbar.

The **Create Constraint** window appears.

6. Select **Rigid Joint on Slot** from the list of available constraint types, as shown in Figure 5-14. Then click **Create**.

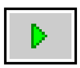
A rigid joint on slot appears on top of the piston head. This constraint connects the piston head to the background and allows the piston head to move only in the vertical direction.

Figure 5-14
Create Constraint Window



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

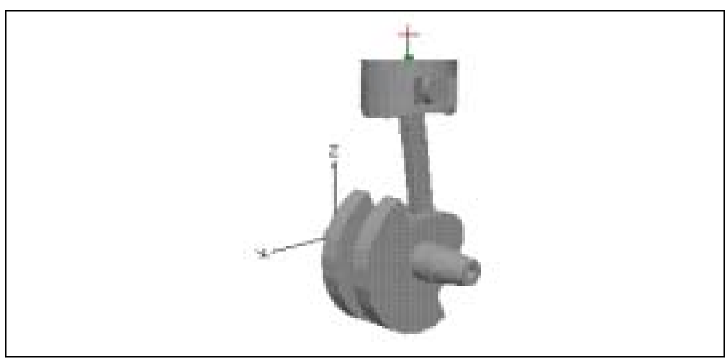
Your original view of the piston assembly (which provides a better view of the motion) is restored.

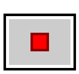


8. Click the **Run** button in MSC.visualNastran Desktop's **Tape Player Control**.

The piston head is pushed up and pulled down by the connecting rod as the crank rotates about the crank pin, as shown in Figure 5-15.

Figure 5-15
Piston Head Pushed Up and Pulled Down by the Connecting Rod





9. Click the **Stop** button.

Your model of the motion of the piston assembly is now complete!

Using CAD Associativity — Mechanical Desktop

You've put a lot of work into adapting the CAD model to accurately simulate the motion of the piston assembly. What happens if the CAD model changes?

When you re-export the CAD model, any changes made to it in Mechanical Desktop are automatically incorporated into your simulation of the motion. All of the changes and enhancements that you have already made in the simulation are left unchanged.

Change the Connecting Rod Length and Re-export the CAD Model

In this step, you will make changes to the CAD model in Mechanical Desktop and re-export the model to MSC.visualNastran Desktop. When you return to MSC.visualNastran Desktop, the changes you made will automatically be incorporated into your simulation of the motion.

1. Reopen the Piston model in your CAD software.

2. Change the length of the **Connecting Rod** part from 150 mm to 250 mm.

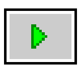
If needed, refer to your CAD software User's Manual or online help for instructions on how to change the length of the connecting rod part.

3. Choose **Connect** in the **MSC.visualNastran** menu.

MSC.visualNastran Desktop for Mechanical Desktop re-exports the piston assembly components and constraints. When the process is done, the MSC.visualNastran Desktop program reopens the linked model, as shown in Figure 5-16.

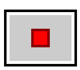
Figure 5-16
Changes in the Mechanical Desktop CAD Model Reflected in MSC.visualNastran Desktop

Your changes to the CAD model are reflected in MSC.visualNastran Desktop. The length of the connecting rod has been changed.



4. Click the **Run** button in MSC.visualNastran Desktop's **Tape Player Control**.

Your simulation of the motion of the piston assembly runs with no problems.



5. Click the **Stop** button.

MSC.visualNastran Desktop preserves and protects your enhancements to the simulation model, while giving you the flexibility to change the underlying geometry of the CAD model as needed.

Using Sub-Assemblies — Mechanical Desktop

You can use MSC.visualNastran Desktop's sub-assembly feature to help organize your design.

1. Choose **Sub-assembly** in the **Insert** menu.

The **Create Sub-assembly** dialog appears.

2. Enter Crank Parts in the text region of the **Sub-assembly** dialog and click **OK**.

The Crank Parts sub-assembly is added to the **Object List**.

3. In the **Object List**, drag and drop the Anchor, both Cranks, and the Crank Pin into the sub-assembly.

The drawing is unchanged by the re-organized **Object List**.

Exercise 5.2 Exploring Pro/ENGINEER Integration and Associativity

Objectives

In this exercise you will learn to

- Export a CAD model from Pro/Engineer.
- Verify and modify your simulation model using the Constraint Navigator.
- Change a constraint to a motor and add new constraints.
- Use CAD Associativity.
- Organize your design using subassemblies.

Support Files

- Tutorials\Chapter 5\Exercise 5.2-ProE\Piston.asm

Exporting a CAD Model — Pro/ENGINEER

1. Launch Pro/ENGINEER.

For Pro/ENGINEER versions 20 and above, a **visualNastran** menu item will appear in the part or assembly menus as appropriate; for earlier versions, the item will also appear in the **Main** menu as shown in Figure 5-17.

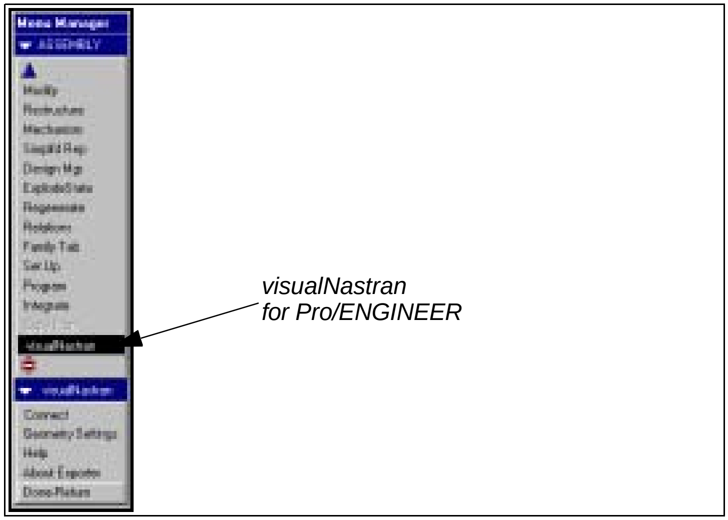


Figure 5-17
MSC.visualNastran menu in Pro/ENGINEER

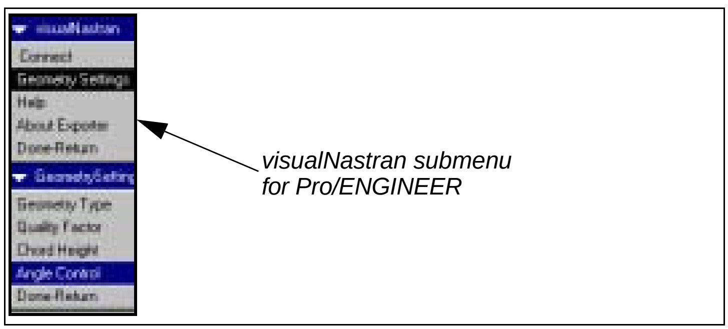


Figure 5-18
MSC.visualNastran Submenu in Pro/ENGINEER

For more information on Geometry and Quality settings, see the online Help. You may ignore these choices for this exercise.

2. Open the file **piston.asm** located in your **Program Files\visualNastran Desktop\Tutorials\Chapter 05\Exercise 5.2-ProE** folder.

The CAD model of the piston assembly is displayed as shown in Figure 5-19.

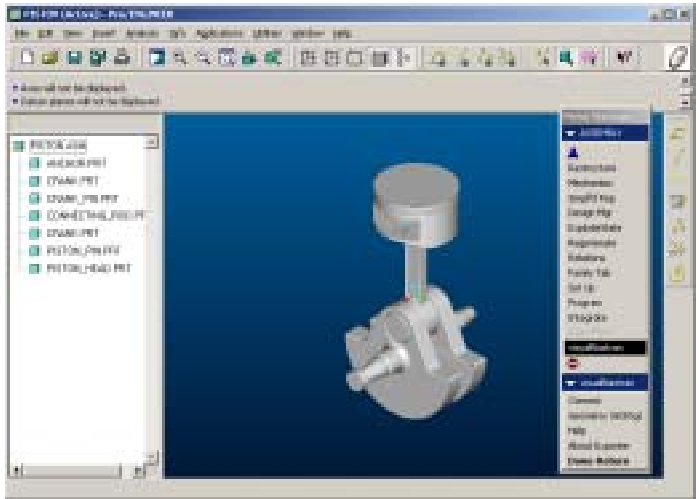


Figure 5-19
Pro/ENGINEER CAD Model of Piston Assembly

3. Choose **Connect** in the**visualNastran** submenu.

MSC.visualNastran Desktop for Pro/ENGINEER may prompt you to view a tutorial overview. This happens only the very first time you Connect. When you close the online tutorial and choose Connect again, MSC.visualNastran Desktop for Pro/ENGINEER maps the assembly components and constraints into MSC.visualNastran Desktop bodies and joints, and creates a new, linked model named **Piston.wm3** in the directory where the CAD files are located.

When the export is complete, the MSC.visualNastran Desktop program opens and displays the CAD Associativity dialog, listing the MSC.visualNastran Desktop objects that are associated with objects in the Pro/ENGINEER model as shown in Figure 5-20.

Figure 5-20
CAD Associativity Dialog

4. Click **OK** to close the **CAD Associativity** dialog.

MSC.visualNastran Desktop prompts you to run the Constraint Navigator.

5. Click **Yes** to run the Constraint Navigator.

The linked model and the Constraint Navigator appear in the modeling window.

6. Click the **Rotate** button and rotate the piston to a vertical orientation.

Using the Constraint Navigator — Pro/ENGINEER

The Constraint Navigator allows you to examine relationships among bodies, subassemblies, and constraints so that you can verify and modify your simulation model. The exercise in this section shows you how to use the Constraint Navigator controls to check your exported model. Refer to Figure 5-21.

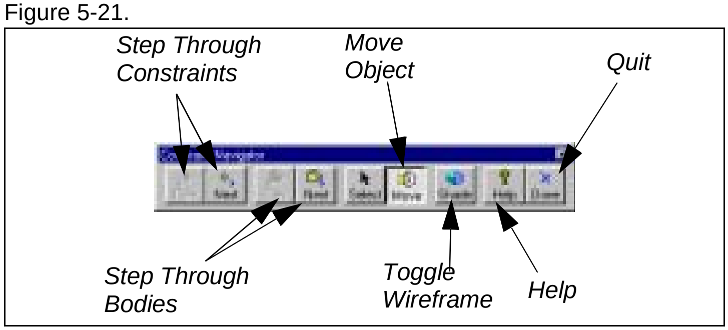
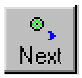


Figure 5-21
Constraint Navigator

When MSC.visualNastran Desktop exports the CAD model, it creates joints between parts of the assembly. The types of joints created depends on the constraints and geometry of the CAD model.

If the joints that MSC.visualNastran Desktop creates are not given the appropriate degrees of freedom, the parts cannot move appropriately when you run the simulation. To correct this problem, you should verify and adjust the joints between the parts of the assembly as necessary to correctly specify the motion desired.

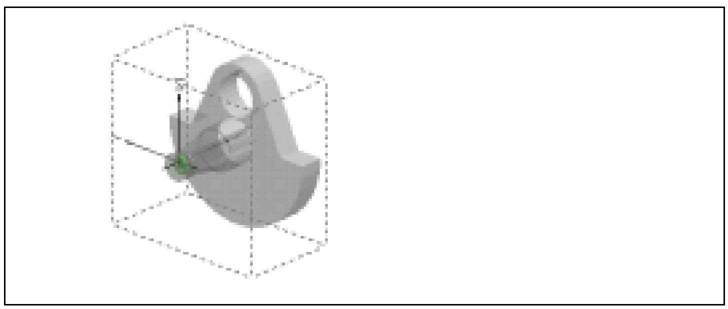
In the steps that follow, you will adjust the joints to give each the appropriate degrees of freedom.

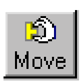


1. Click the **Next Constraint** button.

A revolute joint, **constraint[46]**, is isolated as shown in Figure 5-22.

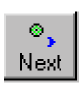
Figure 5-22
Isolated Constraint





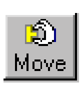
2. Click the **Move** button in the Constraint Navigator, click over the Crank, and drag the mouse.

This tests the kind of movement allowed by the constraint. This constraint is OK.



3. Click the **Next Constraint** button.

A revolute joint, **constraint[50]**, is isolated.



4. Use the **Move** tool to test the constraint by moving the crank and then moving the crank pin.

The model requires a rigid connection between **CRANK_PIN** and **CRANK**. That means changing this revolute joint into a rigid joint.

5. Double-click **constraint[50]** in the **Object List**.

The **Properties** window appears as shown in Figure 5-23. Click the **Constraint** tab if the **Constraint** page is not already displayed.

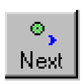
Figure 5-23
Properties Window (Constraint Page)

6. Click the **Rigid Joint** on the Constraint page of the **Properties** window.

7. Under the **Rigid** tab of the **Properties** window, accept the **Optimized** setting.

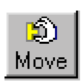
This combines the mass properties of the two connected bodies and computes the motion as if they were one body. Unless the constraint force is to be measured, optimized rigid joints should always be used because they improve simulation compute speeds.

8. Close the **Properties** window.



9. Click the **Next Constraint** button.

A rigid joint on slot, **constraint[52]**, is isolated.



10. Use the **Move** tool to test the constraint.

The model requires deleting this constraint and the two Coords attached to it.

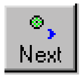
11. Hold down the Control key and select in the Connections List **coord[24] on CONNECTING_ROD** and **coord[26] on CRANK**, the two Coords connected to **constraint[52]**.

The rigid joint on slot, **constraint[52]**, as well as its two Coords should be selected when you are done.

12. Press the delete key.

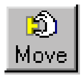
The **MSC.visualNastran Desktop** dialog appears to inform you that you are deleting an object linked to a CAD assembly.

13. Click **OK**.



14. Click the **Next Constraint** button.

A revolute joint on slot, **constraint[54]**, is isolated.



15. Use the **Move** tool to test the constraint.

The model requires changing this constraint to a revolute joint.

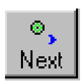
16. Double-click **constraint[54]** in the **Connections List**.

The **Properties** window appears. Click the **Constraint** tab if the **Constraint** page is not already displayed.

17. Select **Revolute Joint** in the **Constraint** page of the **Properties** window.

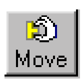
This changes the constraint.

18. Close the **Properties** window.



19. Click the **Next Constraint** button.

A revolute joint, **constraint[56]**, is isolated.



20. Use the **Move** tool to test the constraint.

The model calls for an optimized rigid joint here.

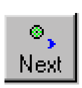
21. Double-click **constraint[56]**.

The **Properties** window appears.

22. Click **Rigid Joint** on the **Constraint** page of the **Properties** window.

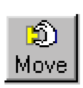
23. Under the **Rigid** tab of the **Properties** window, accept the **Optimized** setting.

24. Close the **Properties** window.



25. Click the **Next Constraint** button.

A revolute joint on plane, **constraint[58]**, is isolated.



26. Use the **Move** tool to test the constraint.

The model requires deleting this constraint and the two Coords attached to it.

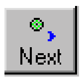
27. Hold down the Control key and select in the Connections List **coord[39] on Crank_2** and **coord[40] on Crank**, the two Coords connected to **constraint[58]**.

The revolute joint on plane, **constraint[58]**, as well as its Coords, should be selected when you are done.

28. Press the delete key.

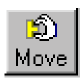
The **MSC.visualNastran Desktop** dialog appears to inform you that you are deleting an object linked to a CAD assembly.

29. Click **OK**.



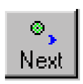
30. Click the **Next Constraint** button.

A revolute joint, **constraint[60]**, is isolated.



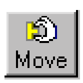
31. Use the **Move** tool to test the constraint.

This joint is OK.



32. Click the **Next Constraint** button.

A revolute joint, **constraint[62]**, is isolated.



33. Use the **Move** tool to test the constraint.

The model requires changing this to an optimized rigid joint.

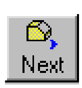
34. Double-click **constraint[62]**.

The **Properties** window appears.

35. Click **Rigid Joint** on the **Constraint** page of the **Properties** window.

36. Under the **Rigid** tab of the **Properties** window, accept the **Optimized** setting.

37. Close the **Properties** window.



38. Next, you can step through each body by clicking on the **Next** and **Previous Body** buttons in the Constraint Navigator to verify body motion.

39. Close the Constraint Navigator.

Anchored Assemblies — Pro/ENGINEER

Based on the constraints and geometry in the Pro/ENGINEER CAD model, MSC.visualNastran Desktop may anchor some parts to the background when it creates the linked model. The first step in simulating the motion of the piston assembly is to make sure that it is properly anchored.

- If no bodies in the assembly are anchored, the piston is floating in space and gravity will cause it to fall when you run the simulation.
- If too many bodies are anchored, the piston won't be able to move appropriately when you run the simulation.

When MSC.visualNastran Desktop imports the piston assembly from Pro/ENGINEER, it doesn't anchor any bodies.

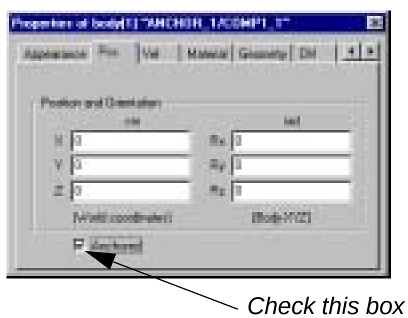
In order for the piston to move properly, you must anchor the body named **ANCHOR** to the background.

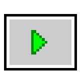
1. Double-click **ANCHOR** in the **Object List**.

The **Properties** window for **ANCHOR** appears.

2. If necessary, click the **Pos** tab in the **Properties** window, then click the **Fixed (Anchored)** box to put a check mark in it, as shown in Figure 5-24.

Figure 5-24
Properties Window (Position Page) for the Anchor



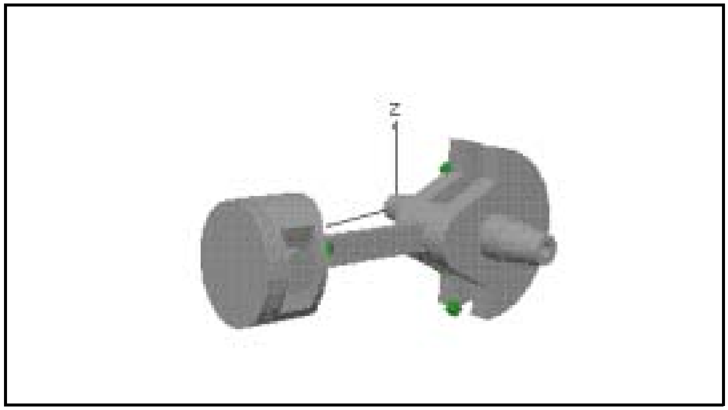


3. Click the **Run** button in MSC.visualNastran Desktop's **Tape Player Control**, and let the simulation run for about 200 frames.

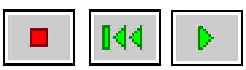
MSC.visualNastran Desktop begins to simulate the motion of the model. The piston assembly balances at first, then the piston head falls under the effect of gravity. Since the piston is almost in equilibrium, you will not see any movement at first.

The geometry of the assembly causes the piston head to swing like a pendulum around the crank pin, which is held in place by the anchor, as shown in Figure 5-25.

Figure 5-25
Piston Head Swinging Around the Crank Pin Like a Pendulum



Since this is the first time the simulation is being run, MSC.visualNastran Desktop calculates the dynamics and stores the data.



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

Depending on the speed of your computer, the animation may be faster this time because the history has already been calculated.

Adding a Motor — Pro/ENGINEER

Currently, the piston assembly is moving only in response to the effects of gravity. In this step, you will change the joint between the anchor and left side of the crank to a motor to simulate the force that drives the motion of the piston.

1. Select **ANCHOR** in the **Object** List.

The **Connections List** displays a list of the objects connected to this body.

2. Double-click **constraint[46]**, the revolute joint that connects **ANCHOR** to **CRANK**.

The Properties window for **constraint[46]** appears.

3. If necessary, click the **Constraint** tab in the **Properties** window. Then, select **Revolute Motor** from the list of available joint types, as shown in Figure 5-26.

The revolute joint is changed into a revolute motor. The **Motor** page of the **Properties** window appears.

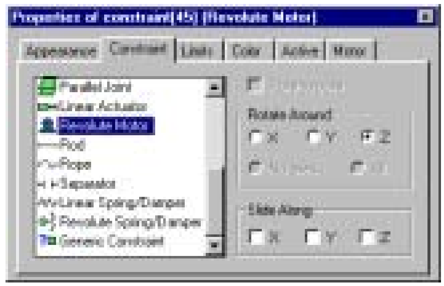


Figure 5-26
Properties Window (Constraint Page) for Revolute Motor

4. Click the **Motor** tab.

This displays the **Motor** page of the **Properties** window.

5. Accept **Angular Velocity** in the list of Motor Types.

6. Enter 500 as the value for angular velocity of this Revolute Motor, as shown in Figure 5-27.

You can close the **Properties** window now.

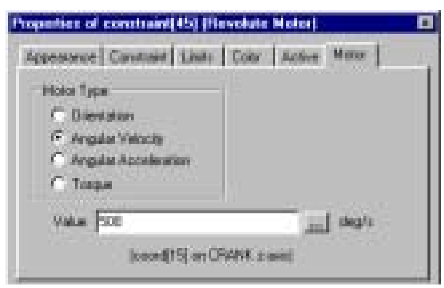
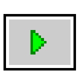


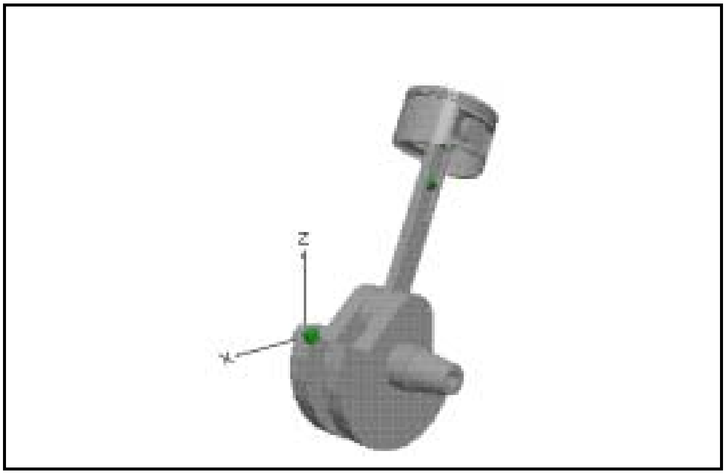
Figure 5-27
Properties WIndow (Motor Page)

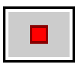


7. Click the **Run** button in MSC.visualNastran Desktop’s **Tape Player Control**.

The piston assembly rotates all the way around the crank pin, as shown in Figure 5-28. It no longer swings like a pendulum, because the motor provides the driving force to keep the piston rotating.

Figure 5-28
Piston Head Rotating All the Way Around the Crank Pin

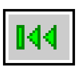




8. Click the **Stop** button.

Adjusting the Joints — Pro/ENGINEER

In this step, you will add a new constraint that limits the motion of the piston head, so that it can only move vertically.



1. Reset the simulation by clicking the **Reset** button.

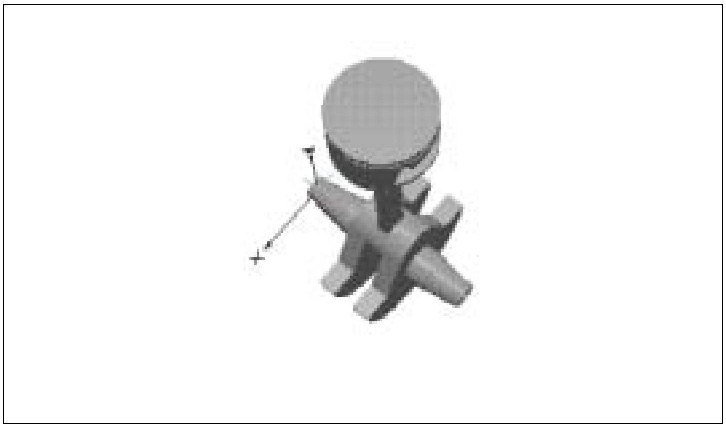
The piston assembly returns to its starting position.

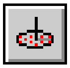
2. Press “Z” on the keyboard, or choose **Look At** in the **View** menu and choose **Top View** in the submenu.

3. Click the **Rotate Around** tool in the **View** toolbar or press “T” on the keyboard so that you can see the top of the piston head and the rest of the drawing.

Your view should be similar to Figure 5-29.

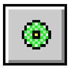
Figure 5-29
Top of the Piston Head





4. Click the **Coord** tool in the **Sketch** toolbar, then place a Coord on the top surface of the piston head, near the center.

Blue lines appear around the piston head and cranks. This feature recognition helps guide placement of the Coord. When you place the Coord, it automatically snaps to the center of the piston head. The z-axis of the Coord should point upward and its red outline surrounds the top center of the piston head.



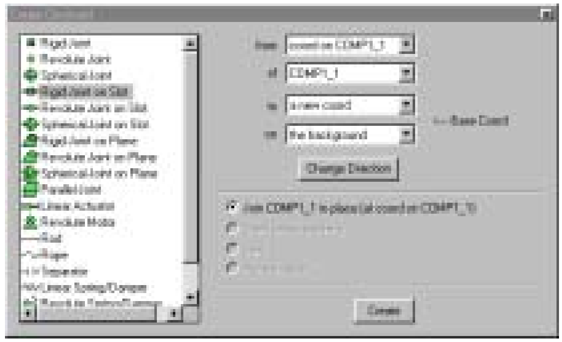
5. Make sure the Coord that you just created is selected. Then click the **Join/Create Constraint** button in the **Edit** toolbar.

The **Create Constraint** window appears.

6. Select **Rigid Joint on Slot** from the list of available constraint types, as shown in Figure 5-30, and click **Create**.

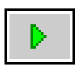
A rigid joint on slot appears on top of the piston head. This constraint connects the piston head to the background and allows the piston head to move only in the vertical (z-axis) direction.

Figure 5-30
Create Constraint Window



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

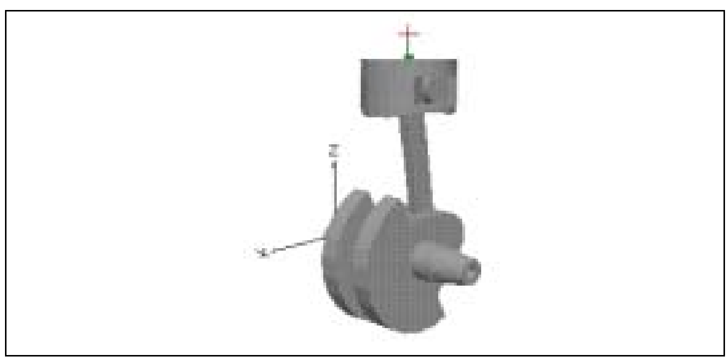
Your original view of the piston assembly (which provides a better view of the motion) is restored.

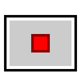


8. Click the **Run** button in MSC.visualNastran Desktop's **Tape Player Control**.

The piston head is pushed up and pulled down by the connecting rod as the crank rotates about the crank pin, as shown in Figure 5-31.

Figure 5-31
Piston Head Pushed Up and Pulled Down by the Connecting Rod





9. Click the **Stop** button.

Your model of the motion of the piston assembly is now complete!

Using CAD Associativity — Pro/ENGINEER

You've put a lot of work into adapting the CAD model to accurately simulate the motion of the piston assembly. What happens if the CAD model changes?

When you re-export the CAD model, any changes to it are automatically incorporated into your simulation of the motion. All of the changes and enhancements that you have already made in the simulation are left unchanged.

Change the Connecting Rod Length and Re-export the CAD Model

In this step, you will re-export the model to MSC.visualNastran Desktop. When you return to MSC.visualNastran Desktop, the changes you made will automatically be incorporated into your simulation of the motion.

1. Reopen the Piston model in your CAD software.

2. Change the length of the **Connecting Rod** part from 150 mm to 250 mm.

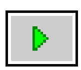
If needed, refer to your CAD software User's Manual or online help for instructions on how to change the length of the connecting rod part.

3. Choose **Connect** in the **MSC.visualNastran** submenu.

MSC.visualNastran Desktop for Pro/ENGINEER re-exports the piston assembly components and constraints. When the process is done, the MSC.visualNastran Desktop program reopens the linked model, as shown in Figure 5-32.

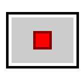
Figure 5-32
Changes in the Pro/ENGINEER CAD Model Reflected in MSC.visualNastran Desktop

Your changes to the CAD model are reflected in MSC.visualNastran Desktop. The length of the connecting rod has been changed.



4. Click the **Run** button in MSC.visualNastran Desktop's **Tape Player Control**.

The piston assembly simulation runs with no problems.



5. Click the **Stop** button.

MSC.visualNastran Desktop preserves and protects your enhancements to the simulation model, while giving you the flexibility to change the underlying geometry of the CAD model as needed.

Using Sub-Assemblies — Pro/ENGINEER

You can use MSC.visualNastran Desktop's sub-assembly feature to help organize your design.

1. Choose **Sub-assembly** in the **Insert** menu.

The **Create Sub-assembly** dialog appears.

2. Enter **Crank Parts** in the text region of the **Sub-assembly** dialog.

The Crank Parts sub-assembly is added to the **Object List**.

3. In the **Object List**, drag and drop the Anchor, both Cranks, and the Crank Pin into the sub-assembly.

The drawing is unchanged by the re-organized **Object List**.

Exercise 5.3

Exploring Solid Edge Integration and Associativity

Objectives

In this exercise you will learn to

- Export a CAD model from Solid Edge.
- Verify and modify your simulation model using the Constraint Navigator.
- Change a constraint to a motor and add new constraints.
- Use CAD Associativity.
- Organize your design using subassemblies.

Support Files

- Tutorials\Chapter 05\Exercise 5.3-Solid Edge\Piston.asm

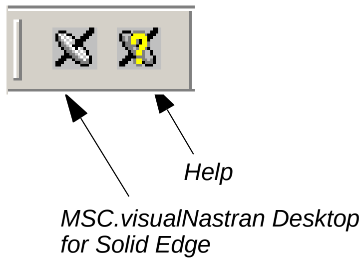
Exporting a CAD Model — Solid Edge

NOTE: To activate the MSC.visualNastran Desktop toolbar in Solid Edge, you must install MSC.visualNastran Desktop for Solid Edge and then launch it from the Start menu as follows: **Start->Programs->visualNastran Desktop->Solid Edge Integration**. You need to perform this procedure only once.

1. Launch the Solid Edge Assembly program.

Note that the MSC.visualNastran Desktop toolbar appears at the left edge of the Solid Edge window, as shown in Figure 5-33.

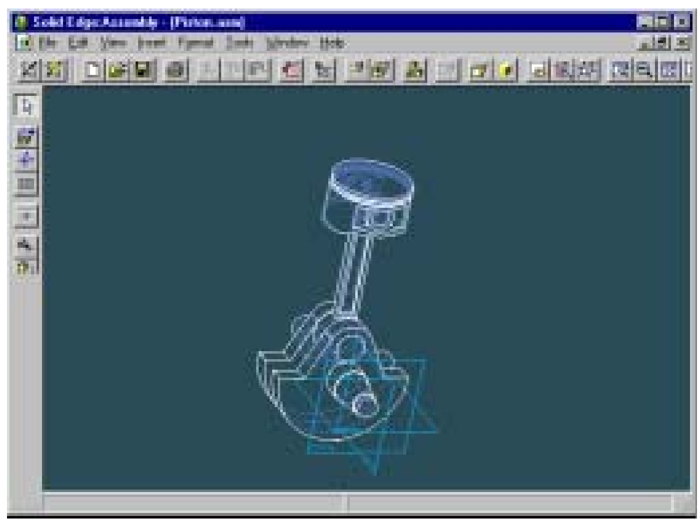
Figure 5-33
MSC.visualNastran Desktop Toolbar in Solid Edge

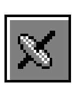


2. Open the file **Piston.asm** located in your **Program Files\visualNastran Desktop\Tutorials\Chapter 05\Exercise 5.3-Solid Edge** folder.

The CAD model of the piston assembly is displayed as shown in Figure 5-34.

Figure 5-34
Solid Edge CAD Model of Piston Assembly





3. Click the **Gyroscope** button in the MSC.visualNastran Desktop toolbar.

MSC.visualNastran Desktop for Solid Edge may prompt you to view a tutorial overview. (This happens only the very first time you Connect.) When you dismiss the online tutorial and click again on the gyroscope button, the **visualNastran for Solid Edge** dialog appears, as shown in Figure 5-35.

Figure 5-35
visualNastran for Solid Edge Dialog

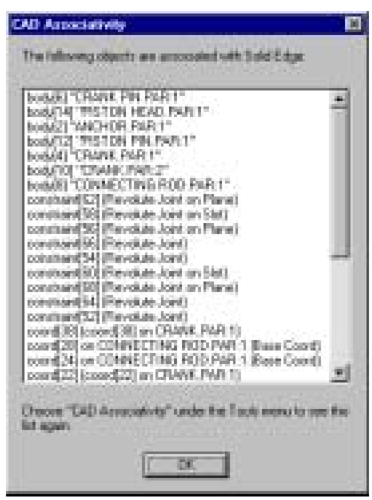
Note that the dialog allows you to select faceted geometry. The default is ACIS. See the online Help for more information on these choices, but you may ignore them for this exercise.

4. Click the **Build Model** button.

MSC.visualNastran Desktop for Solid Edge maps the assembly components and constraints into MSC.visualNastran Desktop bodies and joints, and creates a new, linked model named **Piston.wm3** in the directory where the CAD files are located. As MSC.visualNastran Desktop translates the geometry, the progress is displayed in the **visualNastran for Solid Edge** dialog.

When the export is complete, the MSC.visualNastran Desktop program opens and displays the CAD Associativity dialog, listing the MSC.visualNastran Desktop objects that are associated with objects in the Solid Edge model as shown in Figure 5-36.

Figure 5-36
CAD Associativity Dialog



5. Click **OK** to close the **CAD Associativity** dialog.

MSC.visualNastran Desktop prompts you to run the Constraint Navigator.

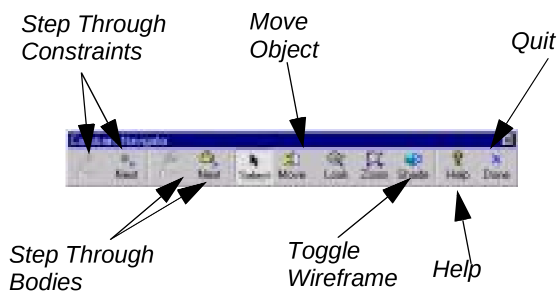
6. Click **Yes** to run the Constraint Navigator.

The linked model and the Constraint Navigator appear in the modeling window.

Using the Constraint Navigator — Solid Edge

The Constraint Navigator allows you to examine relationships among bodies, subassemblies, and constraints so that you can verify and modify your simulation model. The exercise in this section shows you how to use the Constraint Navigator controls to check your exported model. Refer to Figure 5-37.

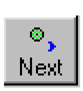
Figure 5-37
Constraint Navigator



When MSC.visualNastran Desktop exports the CAD model, it creates joints between parts of the assembly. The types of joints created depends on the constraints and geometry of the CAD model.

If the joints that MSC.visualNastran Desktop creates are not given the appropriate degrees of freedom, the parts cannot move appropriately when you run the simulation. To correct this problem, you should verify and adjust the joints between the parts of the assembly as necessary to correctly specify the motion desired.

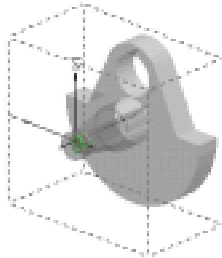
In the steps below, you will adjust the joints to give each the appropriate degrees of freedom.

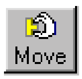


1. Click the **Next Constraint** button.

A revolute joint, **constraint[44]**, is isolated as shown in Figure 5-38.

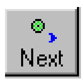
Figure 5-38
Isolated Constraint





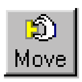
2. Click the **Move** button in the Constraint Navigator, click over the Crank, and drag the mouse.

This tests the kind of movement allowed by the constraint. The constraint is OK.



3. Click the **Next Constraint** button.

A revolute joint, **constraint[46]**, is isolated.



4. Use the **Move** tool to test the constraint by first moving the crank and then moving the crank pin.

The model requires a rigid connection between **Crank Pin.par:1** and **Crank.par:1**. That means changing this revolute joint into a rigid joint.

5. Double-click **constraint[46]** in the **Connections List**.

The **Properties** window appears as shown in Figure 5-39.

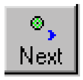
Figure 5-39
Properties Window (Constraint Page)

6. Select **Rigid Joint** on the **Constraint** page of the **Properties** window.

7. Under the **Rigid** tab of the **Properties** window, accept the **Optimized** setting.

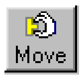
This combines the mass properties of the two connected bodies and computes the motion as if they were one body. Unless the constraint force is to be measured, optimized rigid joints should always be used because they improve simulation computational speeds.

8. Close the **Properties** window.



9. Click the **Next Constraint** button.

A revolute joint, **constraint[48]**, is isolated.



10. Use the **Move** tool to move the crank.

This model requires changing this constraint to an optimized rigid joint.

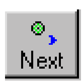
11. Double-click **constraint[48]** in the **Connections List**.

The **Properties** window appears. Click the **Constraint** tab if the **Constraint** page is not already displayed.

12. Select **Rigid Joint** in the **Constraint** page of the **Properties** window.

This changes the constraint.

13. Close the **Properties** window.



14. Click the **Next Constraint** button.

A parallel joint, **constraint[50]**, is isolated.

15. Use the **Move** tool to move the two cranks.

The model requires deleting this constraint and its two Coords.

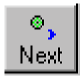
16. Hold down the Control key and select in the Connections List **coord[28] on Crank.par:2** and **coord[30] on Crank.par:1**, the two Coords connected to **constraint[50]**.

The two coords connected to the parallel joint, **constraint[50]**, should be selected when you are done.

17. Press the delete key.

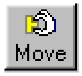
The **MSC.visualNastran Desktop** dialog appears to inform you that you are deleting an object linked to a CAD assembly.

18. Click **OK**.



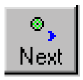
19. Click the **Next Constraint** button.

A revolute joint, **constraint[52]**, is isolated. (You may have to click the Next constraint button several times to reach **constraint[52]**.)



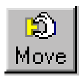
20. Use the **Move** tool to test the constraint.

This constraint is **OK**.



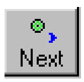
21. Click the **Next Constraint** button.

A revolute joint, **constraint[54]**, is isolated.



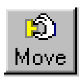
22. Use the **Move** tool to test the constraint.

This constraint is OK.



23. Click the **Next Constraint** button.

A revolute joint, **constraint[56]**, is isolated.



24. Use the **Move** tool to move the piston head.

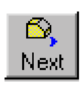
The model requires changing this constraint to an optimized rigid joint.

25. Double-click **constraint[56]** in the **Connections List**.

The **Properties** window appears. Click the **Constraint** tab if the **Constraint** page is not already displayed.

26. Select **Rigid Joint** in the **Constraint** page of the **Properties** window.

This changes the constraint.



27. Next, you can step through each body by clicking on the **Next** and **Previous Body** buttons in the Constraint Navigator to verify body motion.

28. Close the Constraint Navigator and press the Esc key to return to the **Select** tool.

Anchored Assemblies — Solid Edge

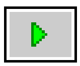
Based on the constraints and geometry in the Solid Edge CAD model, MSC.visualNastran Desktop may anchor some parts to the background when it creates the linked model. The first step in simulating the motion of the piston assembly is to make sure that it is properly anchored.

- If no bodies in the assembly are anchored, the piston is floating in space and gravity will cause it to fall when you run the simulation.
- If too many bodies are anchored, the piston won't be able to move appropriately when you run the simulation.

When MSC.visualNastran Desktop imports the piston assembly from Solid Edge, it anchors the body named **Anchor.par:1**.

1. Look at the **Object List** and note any bodies that have an anchor symbol attached to their icons.

One body in the linked model, **Anchor.par:1**, is anchored (fixed) by MSC.visualNastran Desktop.



2. Click the **Run** button in MSC.visualNastran Desktop's **Tape Player Control**, and let the simulation run for about 100 frames.

MSC.visualNastran Desktop begins to simulate the motion of the model. The piston assembly balances at first, then the piston head falls under the effect of gravity. Since the piston is almost in equilibrium, you will not see any movement at first.

The geometry of the assembly causes the piston head to swing like a pendulum around the crank pin, which is held in place by the anchor, as shown in Figure 5-40.

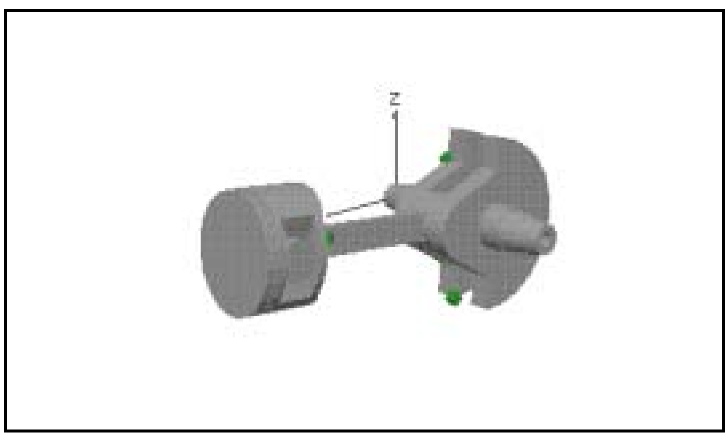
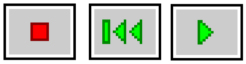


Figure 5-40
Piston Head Swinging Around the Crank Pin Like a Pendulum

Since this is the first time the simulation is being run, MSC.visualNastran Desktop calculates the dynamics and stores the data.



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

Depending on the speed of your computer, the animation may be faster this time because the history has already been calculated.

Adding a Motor — Solid Edge

Currently, the piston assembly is moving only in response to the effects of gravity. In this step, you will change the joint between the anchor and left side of the crank to a motor to simulate the force that drives the motion of the piston.

1. Select **Anchor.par:1** in the **Object List**.

The **Connections List** displays a list of the objects connected to this body.

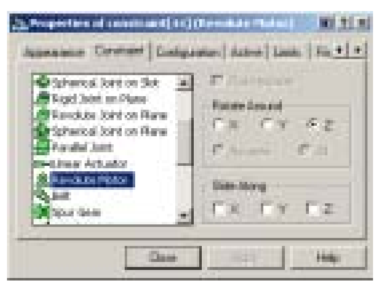
2. Double-click **constraint[44]**, the revolute joint that connects **Anchor.par:1** to **Crank.par:1**.

The **Properties** window for **constraint [44]** appears.

3. If necessary, click the **Constraint** tab in the **Properties** window. Then, select **Revolute Motor** from the list of available joint types, as shown in Figure 5-41.

The revolute joint is changed into a revolute motor.

Figure 5-41
Properties Window (Constraint Page) for Revolute Motor



4. Click the **Motor** tab.

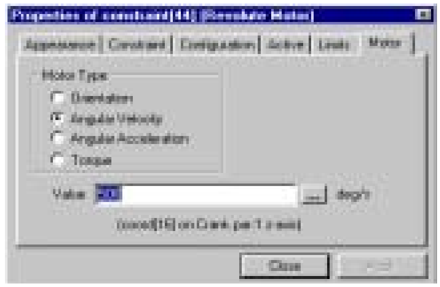
The **Motor** page of the **Properties** window appears.

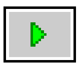
5. Accept **Angular Velocity** in the list of Motor Types.

6. Enter 500 as the value for angular velocity of this Revolute Motor, as shown in Figure 5-42.

You can close the **Properties** window now.

Figure 5-42
Properties Window (Motor Page)

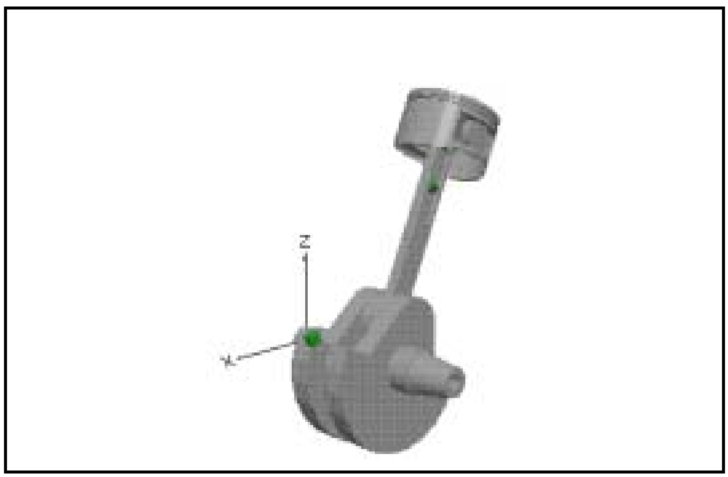


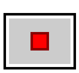


7. Click the **Run** button in MSC.visualNastran Desktop's **Tape Player Control**.

The piston assembly rotates all the way around the crank pin, as shown in Figure 5-43. It no longer swings like a pendulum, because the motor provides the driving torque to keep the piston rotating.

Figure 5-43
Piston Head Rotating All the Way Around the Crank Pin

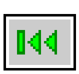




8. Click the **Stop** button.

Adjusting the Joints — Solid Edge

In this step, you will add a new constraint that limits the motion of the piston head, so that it can only move vertically.



1. Reset the simulation by clicking the **Reset** button.

The piston assembly returns to its starting position.

2. Click the **Rotate Around** tool in the **View** toolbar or press “T” on the keyboard so that you can see the top of the piston head and the rest of the drawing.

Your view should be similar to Figure 5-44.

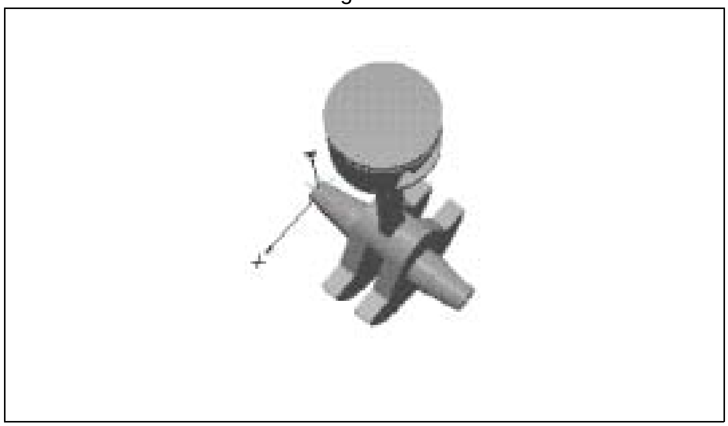
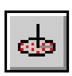
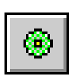


Figure 5-44
Top of the Piston Head



3. Click the **Coord** tool in the **Edit** toolbar, then place a Coord on the top surface of the piston head, near the center.

Blue lines appear around the piston head and cranks. This feature recognition helps guide placement of the Coord. When you place the Coord, it automatically snaps to the center of the piston head. The z-axis of the Coord should point upward and its red outline surrounds the top center of the piston head.



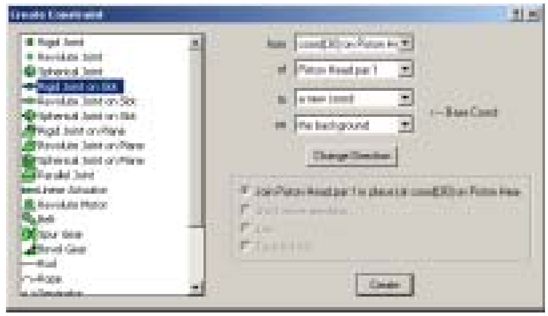
4. Make sure the Coord that you just created is selected. Then click the **Join/Create Constraint** button in the **Edit** toolbar.

The **Create Constraint** window appears.

5. Select **Rigid Joint on Slot** from the list of available constraint types, as shown in Figure 5-45, and click **Create**.

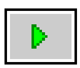
A rigid joint on slot appears on top of the piston head. This constraint connects the piston head to the background and allows the piston head to move only in the vertical (z-axis) direction.

Figure 5-45
Create Constraint Window



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

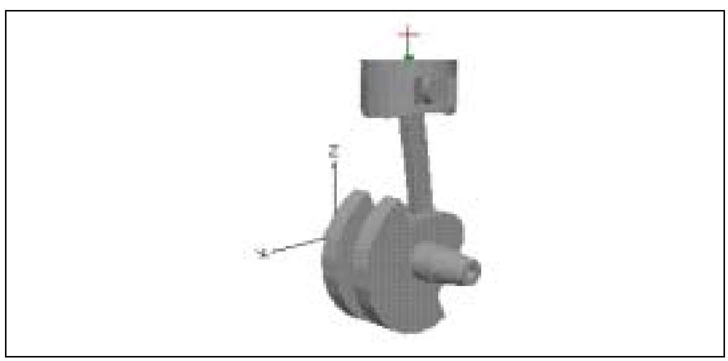
Your original view of the piston assembly (which provides a better view of the motion) is restored.

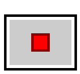


7. Click the **Run** button in MSC.visualNastran Desktop's **Tape Player Control**.

The piston head is pushed up and pulled down by the connecting rod as the crank rotates about the crank pin, as shown in Figure 5-46.

Figure 5-46
Piston Head Pushed Up and Pulled Down by the Connecting Rod





8. Click the **Stop** button.

Your model of the motion of the piston assembly is now complete!

Using CAD Associativity — Solid Edge

You've put a lot of work into adapting the CAD model to accurately simulate the motion of the piston assembly. What happens if the CAD model changes?

When you re-export the CAD model, any changes to it are automatically incorporated into your simulation of the motion. All of the changes and enhancements that you have already made in the simulation are left unchanged.

Change the Connecting Rod Length and Re-export the CAD Model

In this step, you will re-export the model to MSC.visualNastran Desktop. When you return to MSC.visualNastran Desktop, the changes you made will automatically be incorporated into your simulation of the motion.

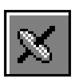
1. Reopen the Piston model in your CAD software.

2. Change the length of the Connecting Rod part from 150 mm to 250 mm.

If needed, refer to your CAD software User's Manual or online help for instructions on how to change the length of the connecting rod part.

3. Save your changes to the Connecting Rod part.

Solid Edge requires you to save your changes before you can re-export the model to MSC.visualNastran Desktop.



4. Click the **Gyroscope** button in the MSC.visualNastran Desktop toolbar.

The **visualNastran for Solid Edge** dialog appears, as shown in Figure 5-47.

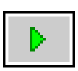
Figure 5-47
visualNastran for Solid Edge Dialog

5. Click the **Build Model** button.

MSC.visualNastran Desktop for Solid Edge re-exports the piston assembly components and constraints. When the process is done, the MSC.visualNastran Desktop program reopens the linked model, as shown in Figure 5-48.

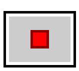
Figure 5-48
Changes in the Solid Edge CAD Model Reflected in MSC.visualNastran Desktop

Your changes to the CAD model are reflected in MSC.visualNastran Desktop. The length of the connecting rod has been changed.



6. Click the **Run** button in MSC.visualNastran Desktop's Tape Player Control.

Your simulation of the motion of the piston assembly runs with no problems.



7. Click the **Stop** button.

MSC.visualNastran Desktop preserves and protects your enhancements to the simulation model, while giving you the flexibility to change the underlying geometry of the CAD model as needed.

Using Sub-Assemblies — Solid Edge

You can use MSC.visualNastran Desktop's sub-assembly feature to help organize your design.

1. Choose **Sub-assembly** in the **Insert** menu.

The **Create Sub-assembly** dialog appears.

2. Enter Crank Parts in the text region of the **Sub-assembly** dialog.

The Crank Parts sub-assembly is added to the **Object List**.

3. In the **Object List**, drag and drop the Anchor, both Cranks, and the Crank Pin into the sub-assembly.

The drawing is unchanged by the re-organized **Object List**.

Exercise 5.4 Exploring SolidWorks Integration and Associativity

Objectives

In this exercise you will learn to

- Export a CAD model from SolidWorks.
- Verify and modify your simulation model using the Constraint Navigator.
- Change a constraint to a motor and add new constraints.
- Use CAD Associativity.
- Organize your design using subassemblies.

Support Files

- Tutorials\Chapter 05\Exercise 5.5-SolidWorks\Piston.sldasm

Exporting a CAD Model — SolidWorks

To activate MSC.visualNastran Desktop for SolidWorks, you need to enable it using the SolidWorks **Add-Ins** feature.

NOTE: You need to perform this procedure once after installing MSC.visualNastran Desktop for SolidWorks.

- Launch the SolidWorks program.
- Choose **Add-Ins...** in the SolidWorks **Tools** menu.
- Check the **visualNastran** checkbox.

1. Launch the SolidWorks program.

2. Open the file **Piston.sldasm** located in your **Program Files\visualNastran Desktop\Tutorials\Chapter 05\Exercise 5.4-SolidWorks** folder.

Note that an **MSC.visualNastran** menu appears in the SolidWorks menu bar, as shown in Figure 5-49.

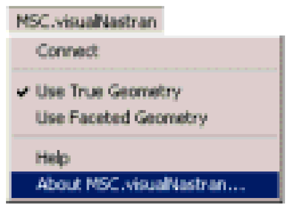


Figure 5-49
MSC.visualNastran menu in SolidWorks

The menu allows you to select true or faceted geometry. The default is faceted. See the Online Help for more information on these choices, but you may ignore them for this exercise.

The CAD model of the piston assembly is displayed as shown in Figure 5-50.

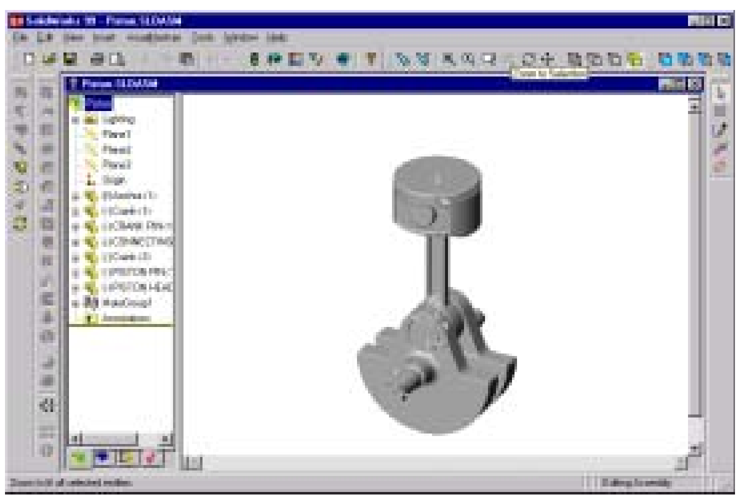


Figure 5-50
SolidWorks CAD Model of Piston Assembly

3. Choose **Connect** in the **MSC.visualNastran** menu.

MSC.visualNastran Desktop for SolidWorks may prompt you to view a tutorial overview. This happens only the very first time you Connect. When you dismiss the online tutorial and choose Connect again, MSC.visualNastran Desktop for SolidWorks maps the assembly components and constraints into MSC.visualNastran Desktop bodies and joints, and creates a new, linked model named **Piston.wm3** in the directory where the CAD files are located. As MSC.visualNastran Desktop translates the geometry, the progress is displayed in the **Preparing Simulation** dialog.

When the export is complete, the MSC.visualNastran Desktop program opens and displays the **CAD Associativity dialog**, listing the MSC.visualNastran Desktop objects that are associated with objects in the SolidWorks model as shown in Figure 5-51.

Figure 5-51
CAD Associativity Dialog

4. Click **OK** to close the **CAD Associativity** dialog.

MSC.visualNastran Desktop prompts you to run the Constraint Navigator.

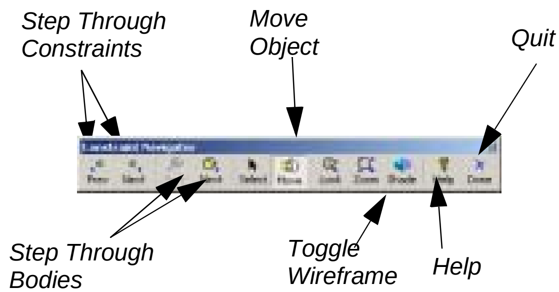
5. Click **Yes** to run the Constraint Navigator.

The linked model and the Constraint Navigator appear in the modeling window.

Using the Constraint Navigator — SolidWorks

The Constraint Navigator allows you to examine relationships among bodies, subassemblies, and constraints so that you can verify and modify your simulation model. The exercise in this section shows you how to use the Constraint Navigator controls to check your exported model. Refer to Figure 5-52.

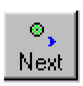
Figure 5-52
Constraint Navigator



When MSC.visualNastran Desktop exports the CAD model, it creates joints between parts of the assembly. The types of joints created depends on the constraints and geometry of the CAD model.

If the joints that MSC.visualNastran Desktop creates are not given the appropriate degrees of freedom, the parts cannot move appropriately when you run the simulation. To correct this problem, you should verify and adjust the joints between the parts of the assembly as necessary to correctly specify the motion desired.

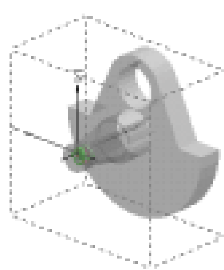
In the steps that follow, you will adjust the joints to give each the appropriate degrees of freedom.

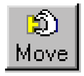


1. Click the **Next Constraint** button.

Concentric1, a revolute constraint, is isolated as shown in Figure 5-53.

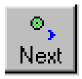
Figure 5-53
Isolated Constraint





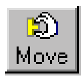
2. Click the **Move** button in the Constraint Navigator, click over the Crank, and drag the mouse.

This tests the kind of movement allowed by the constraint. This constraint is OK.



3. Click the **Next Constraint** button.

Concentric2, another revolute joint, is isolated.



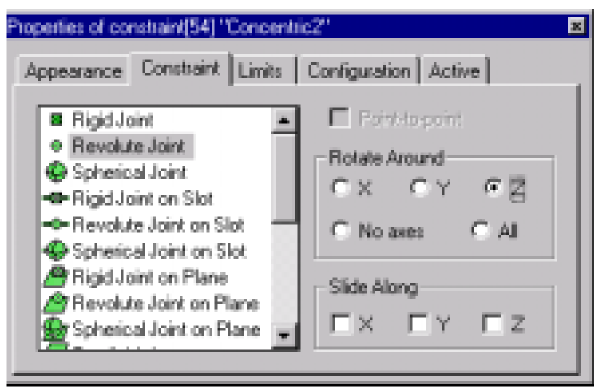
4. Use the **Move** tool to test the constraint by first moving the crank and then moving the crank pin.

The model requires a rigid connection between **CRANK PIN-1** and **Crank-1**. That means changing this revolute joint into a rigid joint.

5. Double-click the **Concentric2** revolute constraint in the **Connections List**.

The **Properties** window appears as shown in Figure 5-54.

Figure 5-54
Properties Window (Constraint Page)

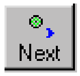


6. Click the **Rigid Joint** on the **Constraint** page of the **Properties** window.

7. Under the **Rigid** tab of the **Properties** window, accept the **Optimized** setting.

This combines the mass properties of the two connected bodies and computes the motion as if they were one body. Unless the constraint force is to be measured, optimized rigid joints should always be used because they improve simulation computational speeds.

8. Close the **Properties** window.

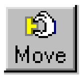


9. Click the **Next Constraint** button.

Distance3, a revolute joint on plane, is isolated.

10. Choose **Look At** in the **View** menu, then choose **Top View** in the **View** submenu.

This provides a better view of the constraint.



11. Use the **Move** tool to test the constraint.

This model requires deleting this constraint and its two Coords.

12. Hold down the Control key and select in the **Connections List** **coord[23] on Crank-1** and **coord[25] on CONNECTING ROD-1**, the two Coords connected to Distance3.

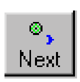
The rigid joint, **Distance3**, as well as its two Coords should be selected when you are done.

13. Press the delete key.

The **MSC.visualNastran Desktop** dialog appears to inform you that you are deleting an object linked to a CAD assembly.

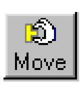
14. Click **OK**.

15. Choose **Look At** in the **View** menu, then choose **Previous View** in the **View** submenu.



16. Click the **Next Constraint** button.

Concentric3, a revolute joint on slot, is isolated.



17. Use the **Move** tool to test the constraint.

The model requires changing this constraint to a revolute joint.

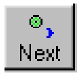
18. Double-click **Concentric3** in the **Connections List**.

The **Properties** window appears. Click the **Constraint** tab if the **Constraint** page is not already displayed.

19. Select **Revolute Joint** in the **Constraint** page of the **Properties** window.

This changes the constraint.

20. Close the **Properties** window.

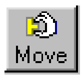


21. Click the **Next Constraint** button.

Parallel1, a parallel joint, is isolated.

22. Choose **Look At** in the **View** menu, then choose **Right View** in the **View** submenu.

This provides a better view of the constraint.



23. Use the **Move** tool to test the constraint.

This model requires deleting this constraint and its two Coords.

24. Hold down the Control key and select in the **Connections List** **coord[31] on Crank-1** and **coord[33] on CONNECTING ROD-1**, the two Coords connected to **Parallel1**.

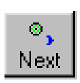
The rigid joint, **Parallel1**, as well as its two Coords should be selected when you are done.

25. Press the delete key.

The **MSC.visualNastran Desktop** dialog appears to inform you that you are deleting an object linked to a CAD assembly.

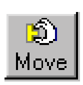
26. Click **OK**.

27. Choose **Previous View** in the **View** menu.



28. Click the **Next Constraint** button.

Concentric4, a revolute joint, is isolated.



29. Use the **Move** tool to test the constraint.

The model calls for an optimized rigid joint here.

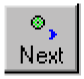
30. Double-click the **Concentric4** revolute constraint.

The **Properties** window appears.

31. Click **Rigid Joint** on the **Constraint** page of the **Properties** window.

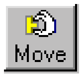
32. Under the **Rigid** tab of the **Properties** window, accept the **Optimized** setting.

33. Close the **Properties** window.



34. Click the **Next Constraint** button.

Parallel2, a parallel joint, is isolated.



35. Use the **Move** tool to test the constraint.

This model requires deleting this constraint and its two Coords.

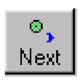
36. Hold down the Controlkey and select in the **Connections List** **coord[39] on Crank-1** and **coord[41] on Crank-2**, the two Coords connected to **Parallel2**.

The rigid joint, **Parallel2**, as well as its two Coords, should be selected when you are done.

37. Press the Delete key.

The **MSC.visualNastran Desktop** dialog appears to inform you that you are deleting an object linked to a CAD assembly.

38. Click **OK**.

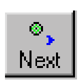


39. Click the **Next Constraint** button.

Concentric5, a revolute joint, is isolated.

40. Use the **Move** tool to test the constraint.

This joint is OK.



41. Click the **Next Constraint** button.

Concentric6, a revolute joint, is isolated.

42. Use the **Move** tool to test the constraint.

The model requires changing this constraint to an optimized rigid joint.

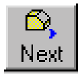
43. Double-click the **Concentric6** revolute constraint.

The **Properties** window appears.

44. Click **Rigid Joint** on the **Constraint** page of the **Properties** window.

45. Under the **Rigid** tab of the **Properties** window, accept the **Optimized** setting.

46. Close the **Properties** window.



47. Next, you can step through each body by clicking on the **Next** and **Previous Body** buttons in the Constraint Navigator to verify body motion.

48. Click **Done** to close the Constraint Navigator.

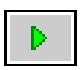
Anchored Assemblies — SolidWorks

NOTE: If a body is "fixed" in SolidWorks, it will be anchored in MSC.visualNastran Desktop. If a body "floats" in SolidWorks, it will not be anchored in MSC.visualNastran Desktop. By default, SolidWorks "fixes" the first body brought into an assembly.

The first step in simulating the motion of the piston assembly is to make sure that it is properly anchored.

- If no bodies in the assembly are anchored, the piston is floating in space and gravity will cause it to fall when you run the simulation.
- If too many bodies are anchored, the piston won't be able to move appropriately when you run the simulation.

When MSC.visualNastran Desktop imports the piston assembly from SolidWorks, it anchors the bodies appropriately. No modifications are necessary. You are ready to run an initial simulation in MSC.visualNastran Desktop.

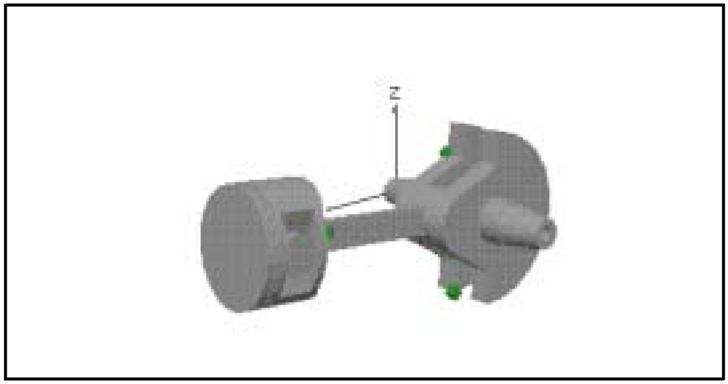


1. Click the **Run** button in MSC.visualNastran Desktop's **Tape Player Control**, and let the simulation run for about 100 frames.

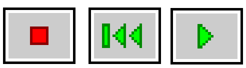
MSC.visualNastran Desktop begins to simulate the motion of the model. The piston assembly balances at first, then the piston head falls under the effect of gravity. Since the piston is almost in equilibrium, you will not see any movement at first.

The geometry of the assembly causes the piston head to swing like a pendulum around the crank pin, which is held in place by the anchor, as shown in Figure 5-55.

Figure 5-55
Piston Head Swinging Around the Crank Pin Like a Pendulum



Since this is the first time the simulation is being run, MSC.visualNastran Desktop calculates the dynamics and stores the data.



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

Depending on the speed of your computer, the animation may be faster this time because the history has already been calculated.

Adding a Motor — SolidWorks

Currently, the piston assembly is moving only in response to the effects of gravity. In this step, you will change the joint between the anchor and left side of the crank to a motor that drives the motion of the piston. The motor applies a torque to the crank.

1. Click on **ANCHOR-1** in the **Object List**.

The **Connections List** displays a list of the objects connected to this body.

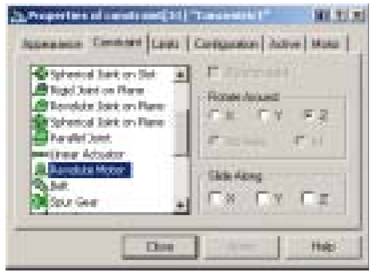
2. Double-click **Concentric1**, the revolute joint that connects **ANCHOR-1** to **CRANK-1**.

The Properties window for **Concentric1** appears.

3. If necessary, click the **Constraint** tab in the **Properties** window. Then, select **Revolute Motor** from the list of available joint types, as shown in Figure 5-56.

The revolute joint is changed into a revolute motor.

Figure 5-56
Properties Window (Constraint Page) for Revolute Motor



4. Click the **Motor** tab.

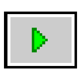
This displays the **Motor** page of the **Properties** window.

5. Accept **Angular Velocity** in the list of Motor Types.

6. Enter 500 as the value for angular velocity of this Revolute Motor, as shown in Figure 5-57.

You can close the **Properties** window now.

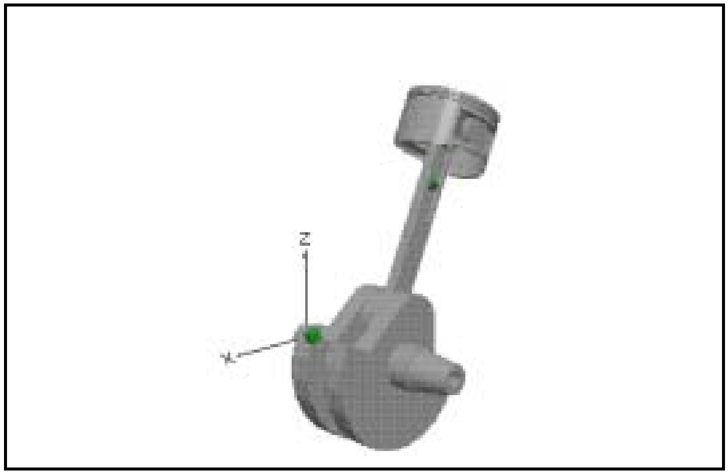
Figure 5-57
Properties Window (Motor Page)

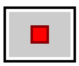


7. Click the **Run** button in MSC.visualNastran Desktop’s **Tape Player Control**.

The piston assembly rotates all the way around the crank pin, as shown in Figure 5-58. It no longer swings like a pendulum, because the motor provides the driving force to keep the piston rotating.

Figure 5-58
Piston Head Rotating All the Way Around the Crank Pin

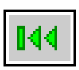




8. Click the **Stop** button.

Adjusting the Joints — SolidWorks

In this step, you will add a new constraint that limits the motion of the piston head, so that it can only move vertically.



1. Reset the simulation by clicking the **Reset** button.

The piston assembly returns to its starting position.

2. Press “Z” on the keyboard, or choose **Look At** in the **View** menu and choose **Top View** in the submenu.

3. Click the **Rotate Around** tool in the **View** toolbar or press “T” on the keyboard so that you can see the top of the piston head and the rest of the drawing.

Your view should be similar to Figure 5-59.

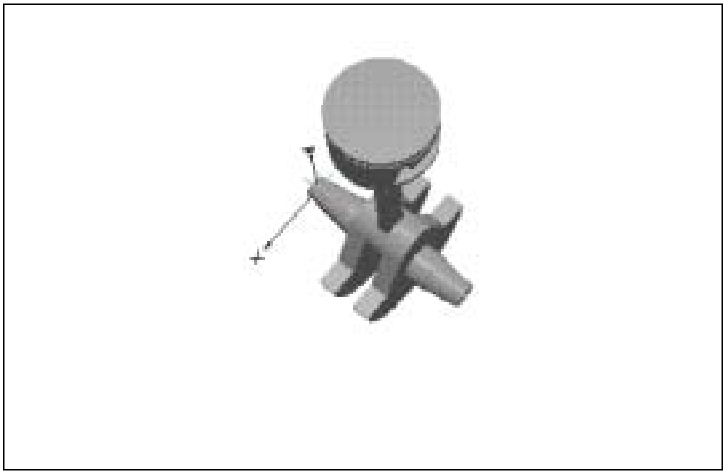
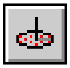
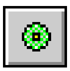


Figure 5-59
Top of the Piston Head



4. Click the **Coord** tool in the **Edit** toolbar, then place a Coord on the top surface of the piston head, near the center.

Blue lines appear around the piston head and cranks. This feature recognition helps guide placement of the Coord. The z-axis of the Coord should point upward and its red outline surrounds the top center of the piston head.



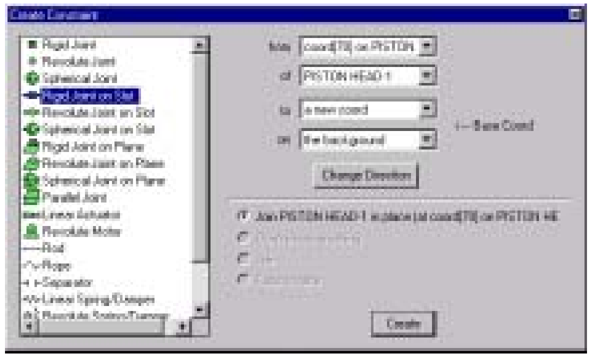
5. Make sure that the Coord that you just created is selected. Then click the **Join/Create Constraint** button in the **Edit** toolbar.

The **Create Constraint** window appears.

6. Select **Rigid Joint on Slot** from the list of available constraint types, as shown in Figure 5-45, and click **Create**.

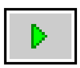
A rigid joint on slot appears on top of the piston head. This constraint connects the piston head to the background and allows the piston head to move only in the vertical (z-axis) direction.

Figure 5-60
Create Constraint Window



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

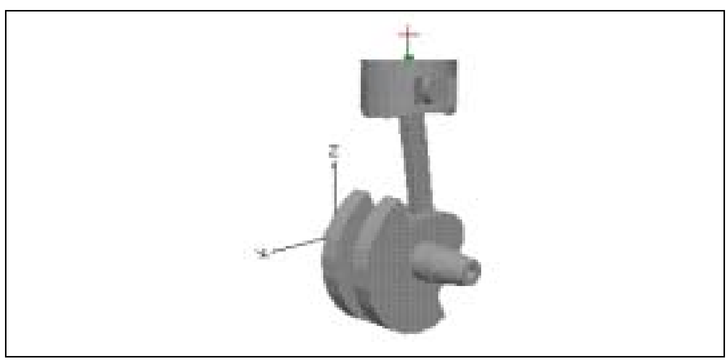
Your original view of the piston assembly (which provides a better view of the motion) is restored.

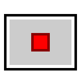


8. Click the **Run** button in MSC.visualNastran Desktop's **Tape Player Control**.

The piston head is pushed up and pulled down by the connecting rod as the crank rotates about the crank pin, as shown in Figure 5-46.

Figure 5-61
Piston Head Pushed Up and Pulled Down by the Connecting Rod





9. Click the **Stop** button.

Your model of the motion of the piston assembly is now complete!

Using CAD Associativity — SolidWorks

You've put a lot of work into adapting the CAD model to accurately simulate the motion of the piston assembly. What happens if the CAD model changes?

When you re-export the CAD model, any changes to it are automatically incorporated into your simulation of the motion. All of the changes and enhancements that you have already made in the simulation are left unchanged.

Change the Connecting Rod Length and Re-export the CAD Model

In this step, you will re-export the model to MSC.visualNastran Desktop. When you return to MSC.visualNastran Desktop, the changes you made will automatically be incorporated into your simulation of the motion.

1. Reopen the Piston model in your CAD software.
2. Change the length of the Connecting Rod part from 150 mm to 250 mm.

If needed, refer to your CAD software User's Manual or online help for instructions on how to change the length of the connecting rod part.

3. Choose **Connect** in the **MSC.visualNastran** menu.

MSC.visualNastran Desktop for SolidWorks re-exports the piston assembly components and constraints. When the process is done, the MSC.visualNastran Desktop program reopens the linked model, as shown in Figure 5-62.

Figure 5-62
Changes in the SolidWorks CAD Model Reflected in MSC.visualNastran Desktop

Your changes to the CAD model are reflected in MSC.visualNastran Desktop. The length of the connecting rod has been changed.

4. Click the **Run** button in MSC.visualNastran Desktop's **Tape Player Control**.

Your simulation of the motion of the piston assembly runs with no problems.

5. Click the **Stop** button.

MSC.visualNastran Desktop preserves and protects your enhancements to the simulation model, while giving you the flexibility to change the underlying geometry of the CAD model as needed.

Using Sub-Assemblies — SolidWorks

You can use MSC.visualNastran Desktop's sub-assembly feature to help organize your design.

1. Choose **Sub-assembly** in the **Insert** menu.

The **Create Sub-assembly** dialog appears.

2. Enter Crank Parts in the text region of the **Sub-assembly** dialog.

The Crank Parts sub-assembly is added to the **Object List**.

3. In the **Object List**, drag and drop the Anchor, both Cranks, and the Crank Pin into the sub-assembly.

The drawing is unchanged by the re-organized **Object List**.

Exercise 5.5 Exploring Autodesk Inventor Integration and Associativity

Objectives

In this exercise you will learn to

- Export a CAD model from Autodesk Inventor.
- Verify and modify your simulation model using the Constraint Navigator.
- Change a constraint to a motor and add new constraints.
- Use CAD Associativity.
- Organize your design using subassemblies.

Software

- MSC.visualNastran 4D or MSC.visualNastran Motion
- Autodesk Inventor

Support Files

- Tutorials\Chapter 5\Exercise 5.5-Inventor\Piston.iam

Exporting a CAD Model — Autodesk Inventor

1. Launch Autodesk Inventor.

MSC.visualNastran Desktop for Autodesk Inventor prompts you to view a tutorial overview. This happens only the very first time you open Autodesk Inventor with MSC.visualNastran Desktop's Autodesk Inventor Integration installed. You can work through the tutorial now or at a later time.

2. Open the file **Piston.iam** located in your **Program Files\MSC.visualNastran Desktop\Tutorials\Chapter 05\Exercise 5.5- Inventor** folder.

The CAD model of the piston assembly is displayed as shown in Figure 5-63.

3. From the **Tools** menu, select **MSC Analysis Tools**. Select **Connect to MSC.visualNastran Desktop...** from the submenu. (See Figure 5-64).

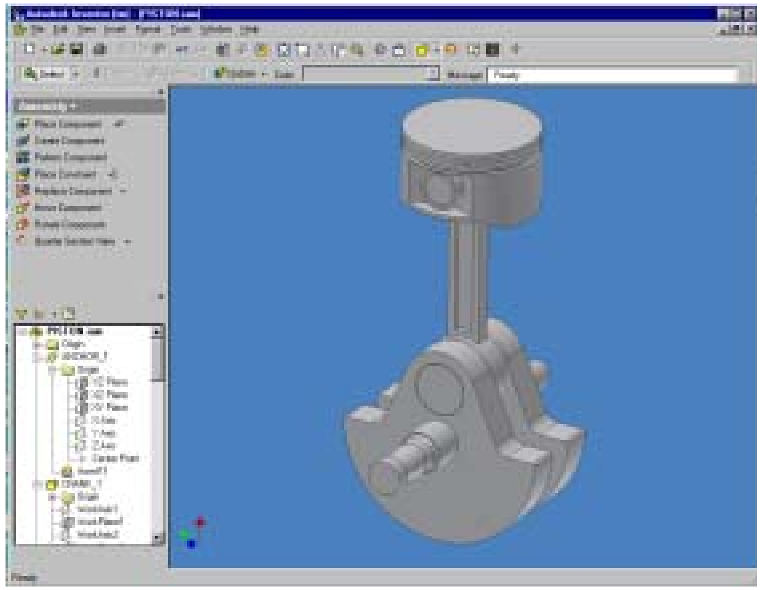


Figure 5-63
Inventor CAD Model of Piston Assembly

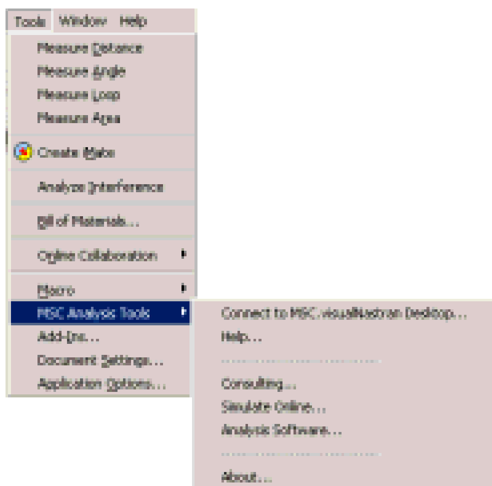


Figure 5-64
MSC Analysis Tools Menu in Inventor

MSC.visualNastran Desktop for Autodesk Inventor maps the assembly components and constraints into MSC.visualNastran Desktop bodies and joints, and creates a new, linked model named **Piston.wm3** in the directory where the CAD files are located. As MSC.visualNastran Desktop translates the geometry, the progress is displayed in the **Preparing Simulation** dialog.

When the export is complete, the MSC.visualNastran Desktop program opens and displays the **CAD Associativity** dialog, listing the MSC.visualNastran Desktop objects that are associated with objects in the Inventor model as shown in Figure 5-65.

Figure 5-65
CAD Associativity Dialog

4. Click **OK** to close the **CAD Associativity** dialog.

MSC.visualNastran Desktop prompts you to run the Constraint Navigator.

5. Click **Yes** to run the Constraint Navigator.

The linked model appears with the Constraint Navigator in the modeling window.

Using the Constraint Navigator — Autodesk Inventor

The Constraint Navigator allows you to examine relationships among bodies, subassemblies, and constraints so that you can verify and modify your simulation model. The exercise in this section shows you how to use the Constraint Navigator controls to check your exported model. Refer to Figure 5-66.

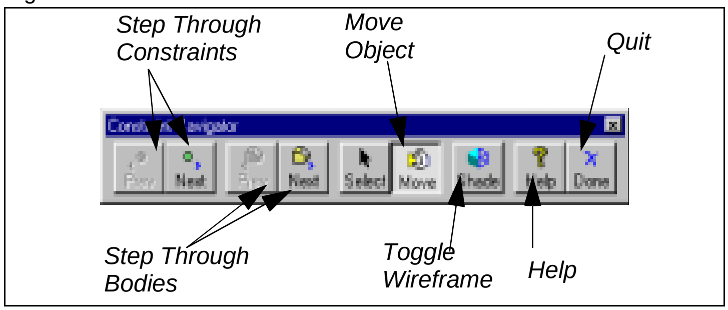
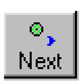


Figure 5-66
Constraint Navigator

When MSC.visualNastran Desktop exports the CAD model, it creates joints between parts of the assembly. The types of joints created depends on the constraints and geometry of the CAD model.

If the joints that MSC.visualNastran Desktop creates are not given the appropriate degrees of freedom, the parts cannot move appropriately when you run the simulation. To correct this problem, you should verify and adjust the joints between the parts of the assembly as necessary to correctly specify the motion desired.

In the steps that follow, you will adjust the joints to give each the appropriate degrees of freedom.



1. Click the **Next Constraint** button.

A revolute joint, **Insert11**, is isolated as shown in Figure 5-67.

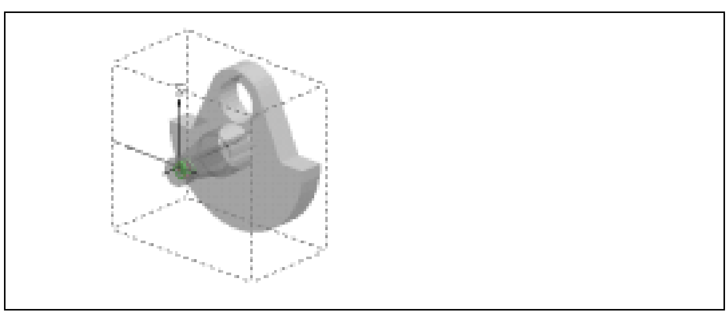
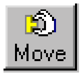
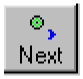


Figure 5-67
Isolated Constraint



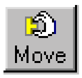
2. Click the **Move** button in the **Constraint Navigator,** click over the Crank, and drag the mouse.

This tests the kind of movement allowed by the constraint. The constraint is OK.



3. Click the **Next Constraint** button.

A revolute joint, **Mate02**, is isolated.



4. Use the **Move** tool to test the constraint by first moving the crank and then moving the crank pin.

The model requires a rigid connection between **CRANKPIN_1** and **CRANK_1**. That means changing this revolute joint into a rigid joint.

5. Double-click **Mate02** in the **Connections List**.

The **Properties** window appears as shown in Figure 5-68.

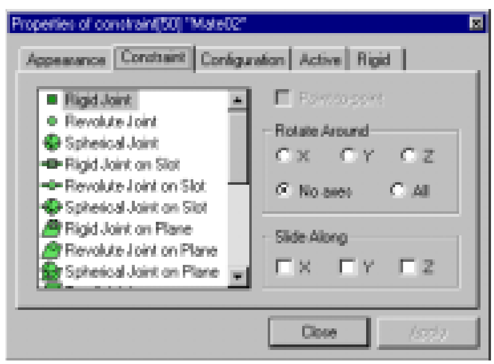


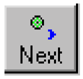
Figure 5-68
Properties Window (Constraint Page)

6. Click the **Rigid Joint** on the **Constraint** page of the **Properties** window.

7. Under the **Rigid** tab of the **Properties** window, accept the **Optimized** setting.

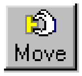
This combines the mass properties of the two connected bodies and computes the motion as if they were one body. Unless the constraint force is to be measured, optimized rigid joints should always be used because they improve simulation computational speeds.

8. Close the **Properties** window.



9. Click the **Next Constraint** button.

A revolute joint on slot, **Mate05**, is isolated.



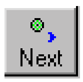
10. Use the **Move** tool to test the constraint.

This model requires changing the constraint to a revolute joint.

11. Double-click **Mate05** in the **Connections List**.

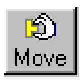
12. Select **Revolute Joint** in the **Constraint** page of the **Properties** window.

13. Close the **Properties** window.



14. Click the **Next Constraint** button.

A revolute joint on plane, **Mate03**, is isolated.



15. Use the **Move** tool to test the constraint.

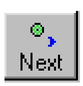
The model requires deleting this constraint and its two coords.

16. Hold down the Control key, and in the **Connections List**, select the two coords connected to **Mate03**.

17. Press the delete key.

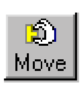
A dialog appears to warn you that you are deleting an object linked to a CAD assembly.

18. Click **OK**.



19. Click the **Next Constraint** button.

A revolute joint, **Mate07**, is isolated.



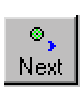
20. Use the **Move** tool to test the constraint.

The model requires changing this constraint to a rigid joint.

21. Double-click **Mate07** in the **Connections List**.

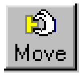
22. Select **Rigid Joint** in the **Constraint** page of the **Properties** window.

23. Close the **Properties** window.



24. Click the **Next Constraint** button.

A revolute joint on plane, **Flush04**, is isolated.



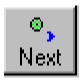
25. Use the **Move** tool to test the constraint.

The model requires deleting this constraint and its two coords.

26. Hold down the Control key, and in the **Connections List**, select the two coords connected to **Flush04**.

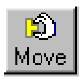
27. Press the delete key.

28. Click **OK**.



29. Click the **Next Constraint** button.

A revolute joint on slot, **Mate08**, is isolated.



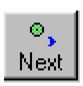
30. Use the **Move** tool to test the constraint.

This model requires changing this constraint to a revolute joint.

31. Double-click **Mate08** in the **Connections List**.

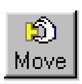
32. Select **Revolute Joint** in the **Constraint** page of the **Properties** window.

33. Close the **Properties** window.



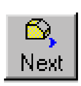
34. Click the **Next Constraint** button.

A revolute joint, **Mate10**, is isolated.



35. Use the **Move** tool to test the constraint.

This constraint is OK.



36. Next, you can step through each body by clicking on the **Next** and **Previous Body** buttons in the Constraint Navigator to verify body motion.

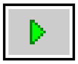
37. Close the Constraint Navigator by clicking the **Done** button.

Anchored Assemblies — Autodesk Inventor

Based on the constraints and geometry in the Autodesk Inventor CAD model, MSC.visualNastran Desktop may anchor some parts to the background when it creates the linked model. The first step in simulating the motion of the piston assembly is to make sure that it is properly anchored.

- If no bodies in the assembly are anchored, the piston is floating in space and gravity will cause it to fall when you run the simulation.
- If too many bodies are anchored, the piston won't be able to move appropriately when you run the simulation.

In this piston assembly, **ANCHOR_1** is fixed.



1. Click the **Run** button in MSC.visualNastran Desktop's **Tape Player Control**.

There is no movement. This is because gravity is acting in the "wrong" direction. By default, gravity acts on the z axis in MSC.visualNastran Desktop, specifically in the "-z" direction. If the z-axis was not in an upward orientation when the CAD assembly was created, then the direction of gravity must be changed before MSC.visualNastran Desktop can properly simulate the motion.

2. Use the **Tape Player Control** to stop and reset the simulation.
3. Choose **Simulation Settings** in the **World** menu.

The **Simulation Settings** window appears.

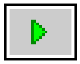
4. Click **Gravity** under **Simulation Settings** in the left pane of the window.

The **Gravity** page appears as shown in Figure 5-69.

Figure 5-69
Settings Window (Gravity Page)

5. Under **Gravity Direction and Magnitude**, select the radio button for the “-X” direction.

6. Click **Close**.

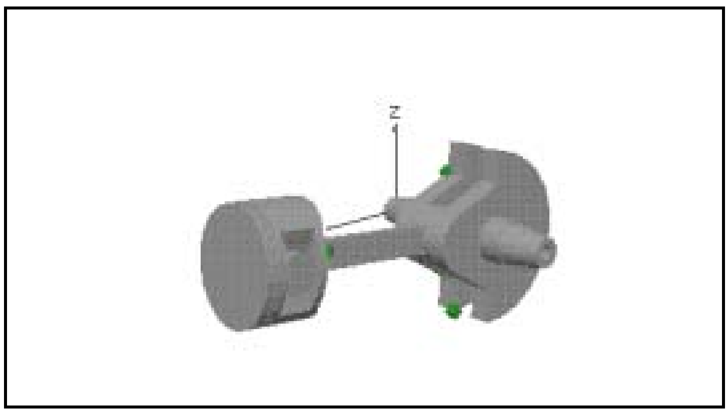


7. Click the **Run** button in MSC.visualNastran Desktop’s **Tape Player Control**.

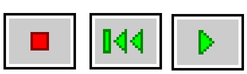
MSC.visualNastran Desktop begins to simulate the motion of the model. The piston assembly balances at first, and then the piston head falls under the effect of gravity. Since the piston is almost in equilibrium, you will not see any movement at first. You will begin to see movement after the simulation runs 120 frames.

The geometry of the assembly causes the piston head to swing like a pendulum around the crank pin, which is held in place by the anchor, as shown in Figure 5-70.

Figure 5-70
Piston Head Swinging Around the Crank Pin Like a Pendulum



Since this is the first time the simulation is being run, MSC.visualNastran Desktop calculates the dynamics and stores the data.



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

Depending on the speed of your computer, the animation may be faster this time because the history has already been calculated.

9. Click the **Stop** button.

Adding a Motor — Autodesk Inventor

Currently, the piston assembly is moving only in response to the effects of gravity. In this step, you will change the joint between the anchor and left side of the crank to a motor to simulate the force that drives the motion of the piston.

1. In the **Object List**, click **ANCHOR_1**.

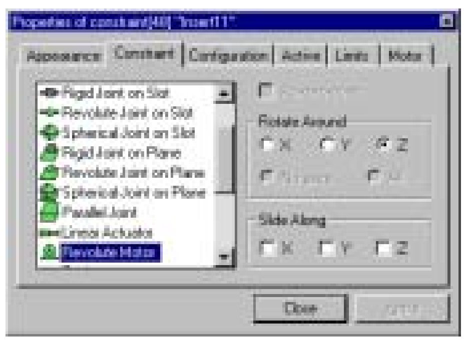
2. In the **Connections List**, double-click **Insert11**, the revolute joint that connects **ANCHOR_1** to **CRANK_1**.

The **Properties** window for **Insert11** appears.

3. If necessary, click the **Constraint** tab in the **Properties** window. Then, select **Revolute Motor** from the list of available joint types, as shown in Figure 5-71.

The revolute joint is changed into a revolute motor.

Figure 5-71
Properties Window (Constraint Page) for Revolute Motor



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

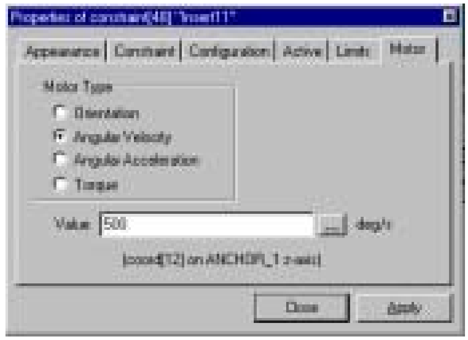
This brings up the motor characteristics page.

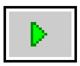
5. Accept **Angular Velocity** in the list of Motor Types.

6. Enter 500 as the value for angular velocity of this Revolute Motor, as shown in Figure 5-72.

You can close the **Properties** window now.

Figure 5-72
Properties WIndow (Motor Page)

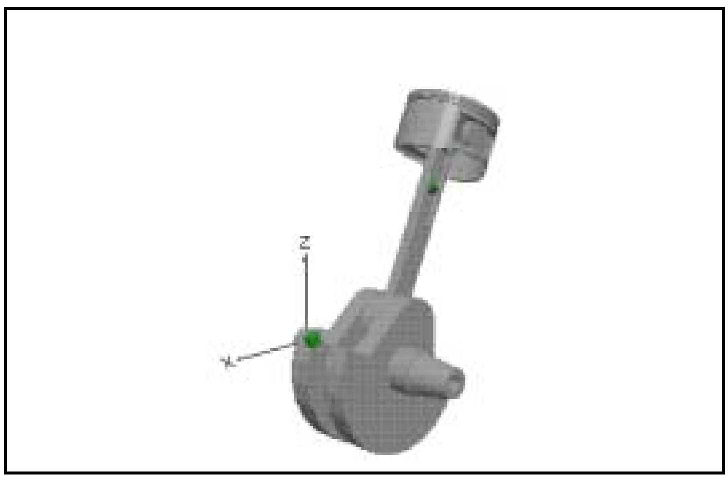


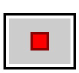


7. Click the **Run** button in MSC.visualNastran Desktop's **Tape Player Control**.

The piston assembly rotates all the way around the crank pin, as shown in Figure 5-73. It no longer swings like a pendulum, because the motor provides the driving force to keep the piston rotating.

Figure 5-73
Piston Head Rotating All the Way Around the Crank Pin

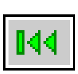




8. Click the **Stop** button.

Adjusting the Joints — Autodesk Inventor

In this step, you will add a new constraint that limits the motion of the piston head, so that it can only move vertically.



1. Reset the simulation by clicking the **Reset** button.

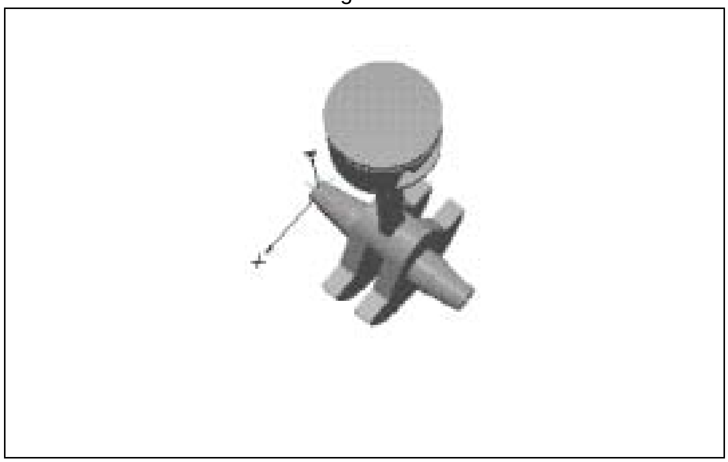
The piston assembly returns to its starting position.

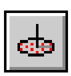
2. Press “X” on the keyboard, or choose **Look At** in the **View** menu and choose **Front View** in the submenu.

3. Click the **Rotate Around** tool in the **View** toolbar and rotate the model so that you can see the top of the piston head and the rest of the drawing.

Your view should be similar to Figure 5-74.

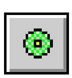
Figure 5-74
Top of the Piston Head





4. Click the **Coord** tool in the **Edit** toolbar, then place a Coord on the top surface of the piston head, near the center.

Blue lines appear around the piston head and cranks. This feature recognition helps guide placement of the Coord. When you place the Coord on the blue circular outline on top of the piston head, it automatically snaps to the center of the piston head. The z-axis of the Coord should point upward and its red outline surrounds the top center of the piston head.



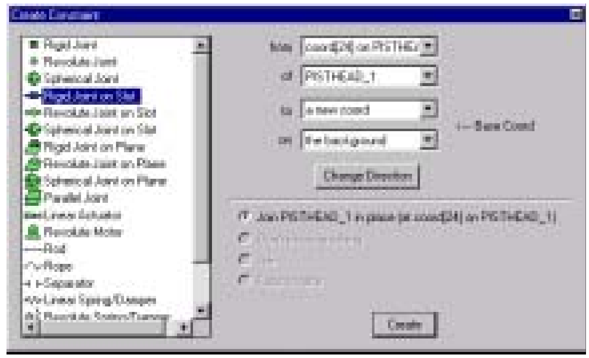
5. Make sure that the Coord that you just created is selected. Then click the **Join/Create Constraint** button in the **Edit** toolbar.

The **Create Constraint** window appears.

6. Select **Rigid Joint on Slot** from the list of available constraint types, as shown in Figure 5-75. Then click **Create**.

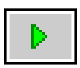
A rigid joint on slot appears on top of the piston head. This constraint connects the piston head to the background and allows the piston head to move only in the vertical direction.

Figure 5-75
Create Constraint Window



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

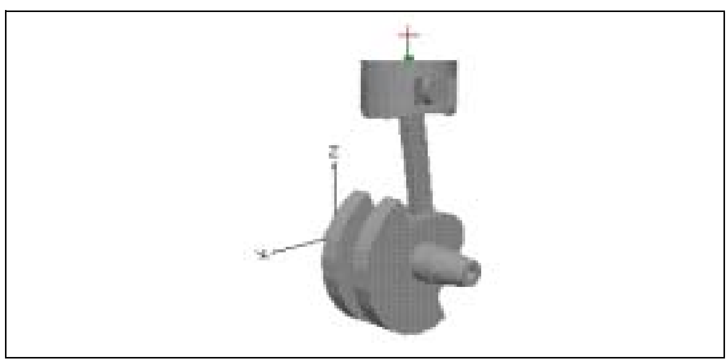
Your original view of the piston assembly (which provides a better view of the motion) is restored.

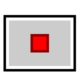


8. Click the **Run** button in MSC.visualNastran Desktop’s **Tape Player Control**.

The piston head is pushed up and pulled down by the connecting rod as the crank rotates about the crank pin, as shown in Figure 5-76.

Figure 5-76
Piston Head Pushed Up and Pulled Down by the Connecting Rod





9. Click the **Stop** button.

Your model of the motion of the piston assembly is now complete!

Using CAD Associativity — Autodesk Inventor

You’ve put a lot of work into adapting the CAD model to accurately simulate the motion of the piston assembly. What happens if the CAD model changes?

When you re-export the CAD model, any changes to it are automatically incorporated into your simulation of the motion. All of the changes and enhancements that you have already made in the simulation are left unchanged.

Change the Connecting Rod Length and Re-export the CAD Model

In this step, you will change the piston in Autodesk Inventor and re-export the model to MSC.visualNastran Desktop. When you return to MSC.visualNastran Desktop, the changes you made will automatically be incorporated into your simulation of the motion.

1. Reopen the Piston model in your CAD software.
2. Change the length of the Connecting Rod part from 150 mm to 250 mm.

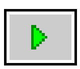
If needed, refer to your CAD software User’s Manual or online help for instructions on how to change the length of the connecting rod part.

3. Choose **Tools>MSC Analysis Tools>Connect to MSC.visualNastran Desktop...**.

MSC.visualNastran Desktop for Autodesk Inventor re-exports the piston assembly components and constraints. When the process is done, the MSC.visualNastran Desktop program reopens the linked model, as shown in Figure 5-77.

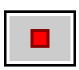
Figure 5-77
Changes in the Inventor CAD Model Reflected in MSC.visualNastran Desktop

Your changes to the CAD model are reflected in MSC.visualNastran Desktop. The length of the connecting rod has been changed.



4. Click the **Run** button in MSC.visualNastran Desktop’s **Tape Player Control**.

Your simulation of the motion of the piston assembly runs with no problems.



5. Click the **Stop** button.

MSC.visualNastran Desktop preserves and protects your enhancements to the simulation model, while giving you the flexibility to change the underlying geometry of the CAD model as needed.

Using Sub-Assemblies — Autodesk Inventor

You can use MSC.visualNastran Desktop’s sub-assembly feature to help organize your design.

1. Choose **Sub-assembly** in the **Insert** menu.

The **Create Sub-assembly** dialog appears.

2. Enter **Crank Parts** in the text region of the **Sub-assembly** dialog and click **OK**.

The Crank Parts sub-assembly is added to the **Object List**.

3. In the **Object List**, drag and drop the Anchor, both Cranks, and the Crank Pin into the sub-assembly.

The drawing is unchanged by the re-organized **Object List**.