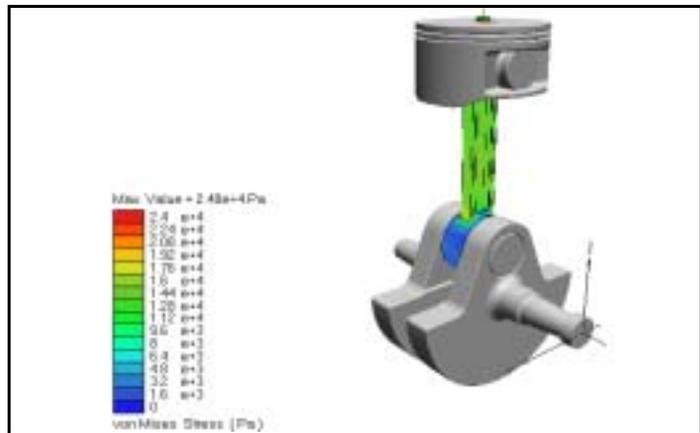
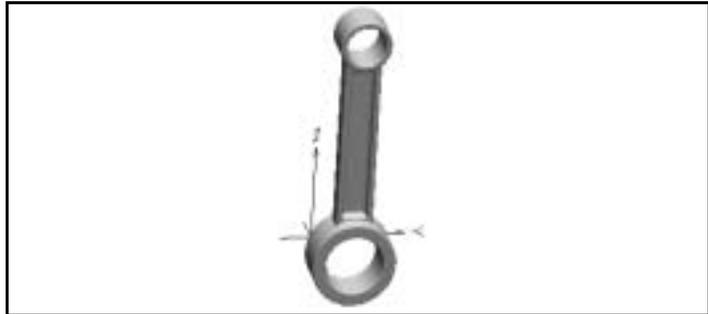


CHAPTER 8

Stress Simulation



Exercise 8-1

Steady State Stress Simulation

Objectives

In this exercise you will learn to

- Apply a distributed load and restraint to a part.
- Perform steady state stress simulation.

Software

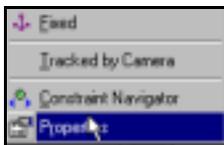
MSC.visualNastran 4D or MSC.visualNastran Desktop FEA

Support Files

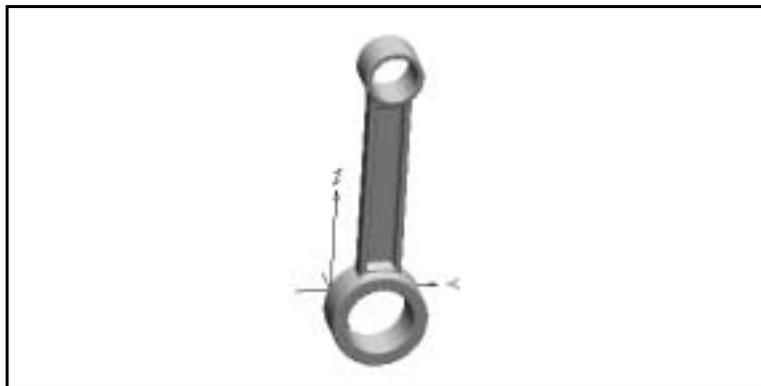
- Tutorials\Chapter 08\Exercise 8.1\Connecting Rod.wm3
1. Launch MSC.visualNastran Desktop.
 2. Choose **Open** from the **File** menu.
 3. Browse the **Tutorials\Chapter 08\Exercise 8.1** folder and open the **Connecting Rod.wm3** file.

Prepare the Simulation

Before running the stress simulation, you must include the part in the finite element analysis, load it, and restrain it.



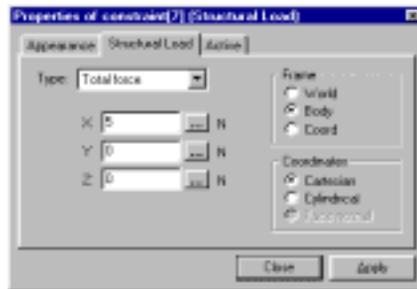
1. Right click on **Connecting Rod-1** and choose **Properties** from the pop-up menu, or double-click **Connecting Rod-1** in the **Object List**.
2. Click the **FEA** tab of the **Properties** window. (To display this page, you first may need to select the **FEA** checkbox in the **Properties List** of the **Object Manager**.)
3. Select the **Include in FEA** setting.
4. Enter a **Default Mesh Size** of 15 mm and click the **Apply** button.
5. Close the **Properties** window.
6. Rotate the connecting rod to expose the hole surfaces.



7. Select the **Structural Load** tool on the **Sketch** toolbar.

8. Click at a point on the inside surface of the top hole to apply the distributed load.
9. Right-click the distributed force in the **Connections List** and choose **Properties**.
10. Click the **Structural Load** tab to display the **Structural Load** page of the **Properties** window.

Figure 8-1
Properties Window
(Distributed Load Page)



11. Select the **Total Force** setting. Enter 5, 0, 0 in the **X, Y, Z** text fields, respectively.
12. Select **Body** frame and **Cartesian** coordinates.
13. Close the **Properties** window.
14. Select the **Restraint** tool on the **Sketch** toolbar.
15. Click at a point on the inside surface of the lower hole to apply the restraint.



Perform Stress Analysis

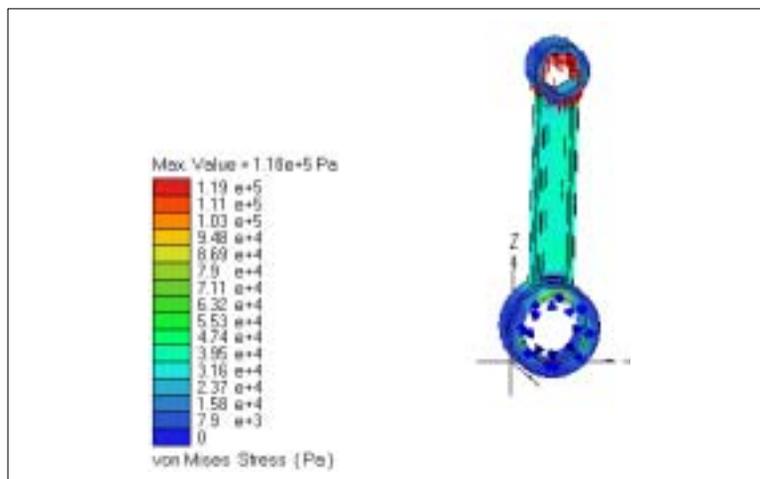
Now you are ready to run the stress analysis on the connecting rod.



1. Select the **Solve FEA** icon. MSC.visualNastran Desktop performs a stress simulation.

The results are displayed when the analysis is complete.

Figure 8-2
MSC.Nastran Results



Exercise 8-2

Dynamic Stress Simulation

Objectives

In this exercise you will learn to

- Perform dynamic stress simulation.

Software

MSC.visualNastran 4D

Support Files

- Tutorials\Chapter 08\Exercise 8.2\Piston.wm3
1. Launch MSC.visualNastran Desktop (if it not already running).
 2. Choose **Open** from the **File** menu.
 3. Browse the **Tutorials\Chapter 08\Exercise 8.2** folder and open the **Piston.wm3** file.

Perform Stress Simulation

Next, you will designate the connecting rod as an FEA body and set up the dynamic stress simulation.

1. Right click the connecting rod and choose **Include in FEA** from the pop-up menu.
2. Under the **World** menu, select **Simulation Settings**.

Figure 8-3
Simulation Settings Window



3. In the FEA section, select the **Auto-compute FEA at every frame** setting. Close the window.



4. Click the **Run** button in the **Tape Player** control.

NOTE: You may receive a warning regarding redundant constraints. Please ignore the warning for now and continue with the exercise.

The piston begins to rotate while MSC.Nastran calculates the stress on the connecting rod at every frame. The program will continue calculating the dynamic FEA results until the **Stop** button is pressed.

5. Click the **Stop** button in the **Tape Player** control after several frames have been calculated.

Figure 8-4
FEA Simulation Results

