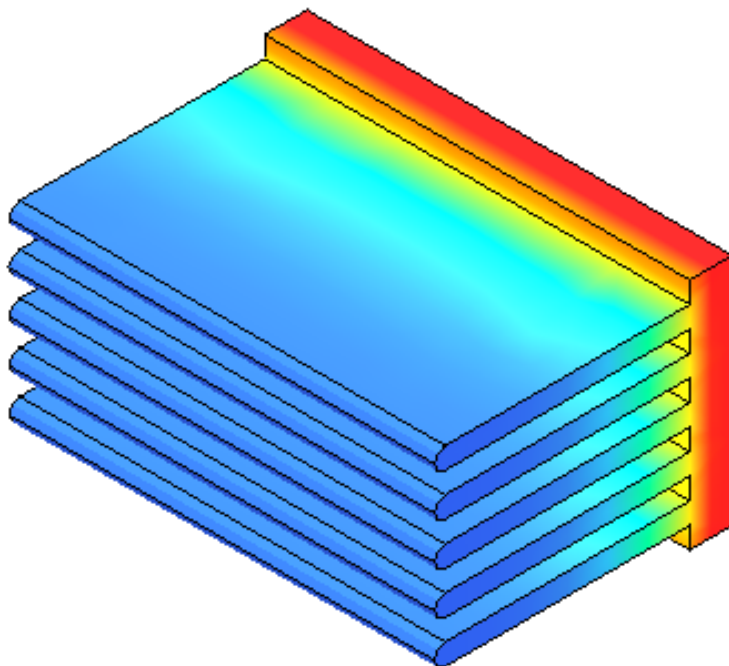


CHAPTER 11

Heat Transfer



Exercise 11-1

Create Thermal Constraints

Objectives

In this exercise you will learn to

- Add the thermal toolbar to your workspace.
- Set thermal constraints.

- Place a thermal load on an object.

Software

- MSC.visualNastran 4D, MSC.visualNastran Desktop FEA

Support Files

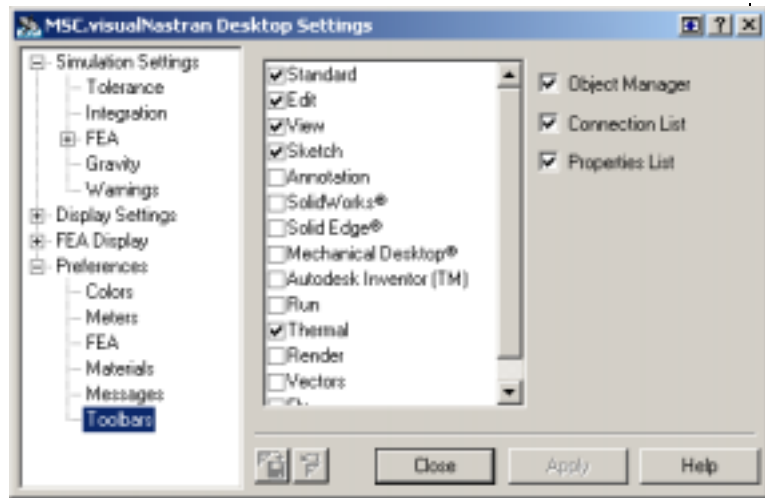
- Tutorials\Chapter 11\Exercise 11.1\heat_transfer.wm3
1. Launch MSC.visualNastran Desktop.
 2. Choose **Open** from the **File** menu.
 3. Browse the **Tutorials\Chapter 11\Exercise 11.1** folder and open the file **heat_transfer.wm3**.

Add the Thermal Toolbar

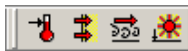


1. Click the **Display Settings** button.
2. Expand the menu under **Preferences**. Select **Toolbars** and click the **Thermal** checkbox, as shown in Figure 11-1.

Figure 11-1
Settings Window



3. Click **Close**.



The Thermal toolbar now appears in the modeling window.

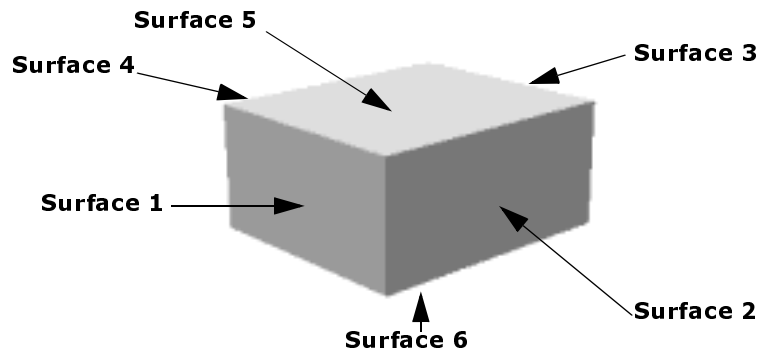
Add Heat Boundary Conditions to Various Surfaces



1. Click the **Prescribed Temperature** button in the **Thermal** toolbar.
2. Select **Surface 1**. Note that the faces are highlighted as the cursor moves over them before selection.

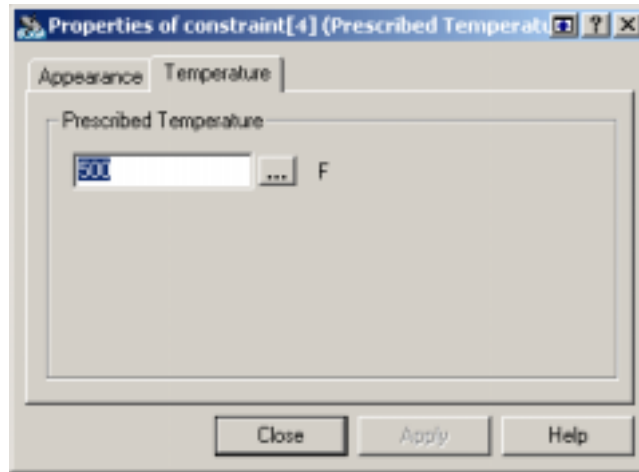
NOTE: You will refer to Figure 11-2 throughout this exercise as you assign thermal constraints to the box surfaces shown.

Figure 11-2
Surface Reference



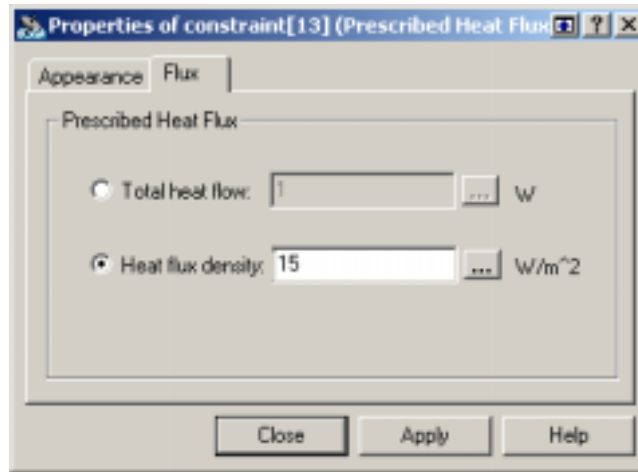
3. In the **Object List** double-click the **Prescribed Temperature** icon, **constraint[6]**, to open the **Properties** window.
4. Select the **Temperature** tab if not already selected.
5. Enter 500 degrees and click **Close**.

Figure 11-3
*Prescribed Temperature
Constraint Window*



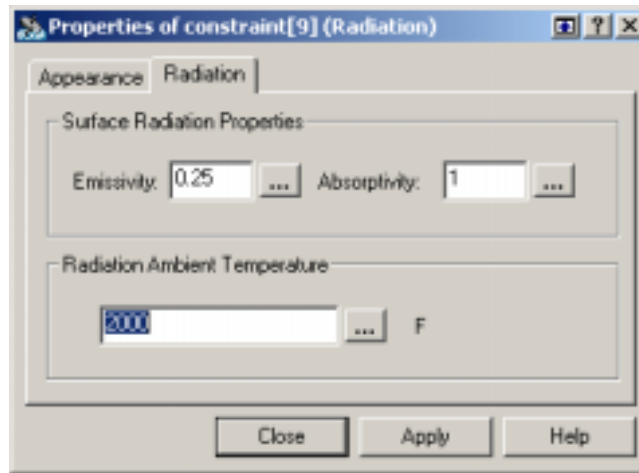
6. Click the **Prescribed Heat Flux** button in the **Thermal** toolbar.
7. Select **Surface 5**.
8. In the **Object List** double-click the **Prescribed Heat Flux** icon, **constraint[13]**, to open the **Properties** window.
9. Click the **Flux** Tab if not already selected.
10. Select the **Heat flux density** radio button and enter 15 in the **Heat flux density** box and click **Close**.

Figure 11-4
Prescribed Heat Flux Constraint Window



11. Click the **Surface Radiation** button in the **Thermal** toolbar.
12. Select **Surface 2**.
13. In the **Object List** double-click the **Surface Radiation** icon, **constraint[18]**, to open the **Properties** window.
14. Go to the **Radiation** page by clicking the **Radiation** tab.
15. Enter 0.25 in the **Emissivity** box and 2000 degrees in **Radiation Ambient Temperature** box.

Figure 11-5
Radiation Constraint Window



16. Close the **Properties** window



17. Use the **Rotate around** button to manipulate the box so that surfaces 3, 4, and 6 are visible.



18. Double-click the **Convective Heat Flux** button in the **Thermal** toolbox.



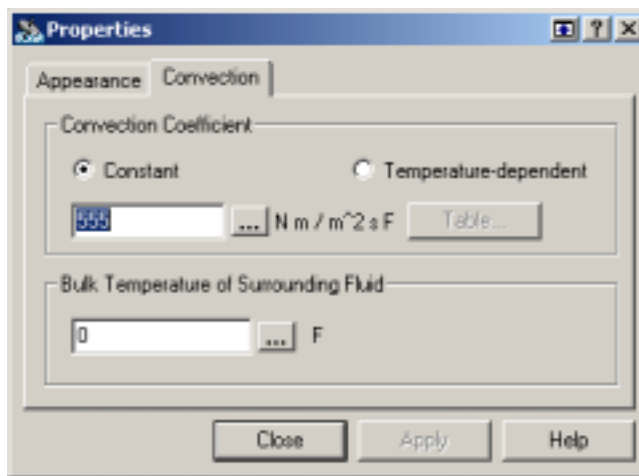
Double-clicking a constraint puts the constraint selection into "sticky mode" for continuous selection of multiple surfaces.

19. Select **Surfaces 3, 4, and 6**.

20. In the **Object List** double-click **constraint[23]** to open the **Properties** window.

21. Click the **Convection** tab if it is not already selected.

Figure 11-6
Convection Properties Window



22. Select the **Constant** radio button.

23. Enter 555 in the fluid **Convection Coefficient** box.

24. Enter 0 degrees in the **Bulk Temperature of Surrounding Fluid**.

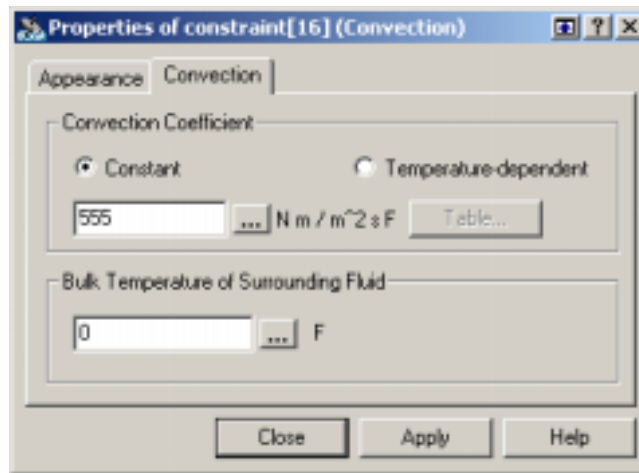
25. In the **Object List**, click **constraint[26]** to view its properties.



You can edit the properties of several objects without closing the **Properties** window. As you select different objects in the **Object Manager** or modeling window, the properties of the selected object are displayed in the **Properties** window.

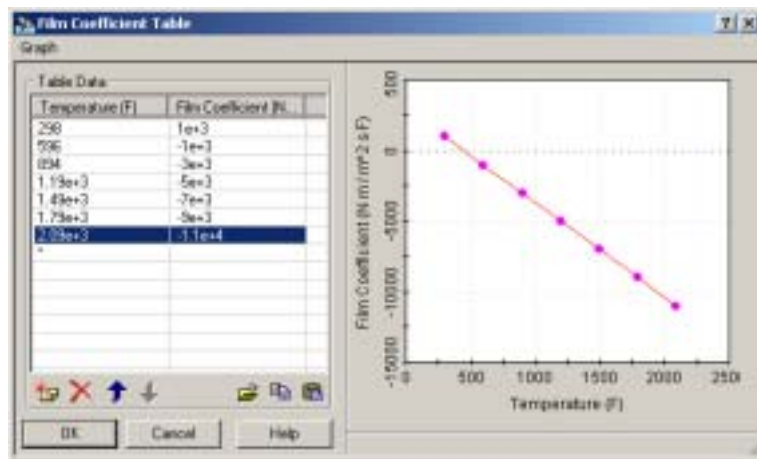
26. Select the **Constant** radio button.
27. Enter 555 in the fluid **Convection Coefficient** box.
28. Enter 0 degrees in the **Bulk Temperature of Surrounding Fluid**.

Figure 11-7
Convection Constraint Window



29. In the **Object List**, click **constraint[29]** to view its properties.
30. Select the **Temperature-dependent** radio button and click **Table**.

Figure 11-8
Film Coefficient Table



31. In the **Film Coefficient Table** dialog, click the **Insert data** button several times. This will generate a set of linear data automatically, or you can insert your own values either by hand or by importing data from a file.
32. Click **OK** to close the **Film Coefficient Table** dialog.
33. Close **Properties** window.

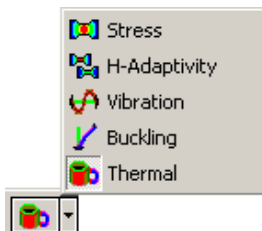


Values in the table data can be edited by the keyboard or by dragging the graph points with the mouse.



34. Select the **FEA Thermal Solve** button.

Figure 11-9
FEA Pull-down Menu




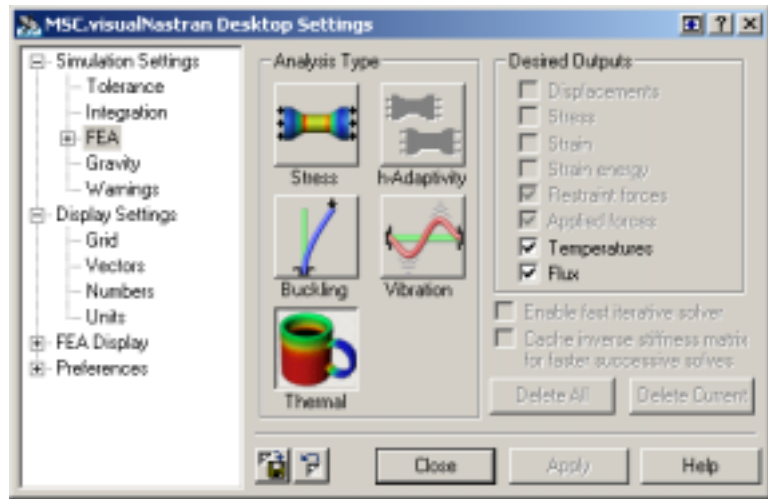
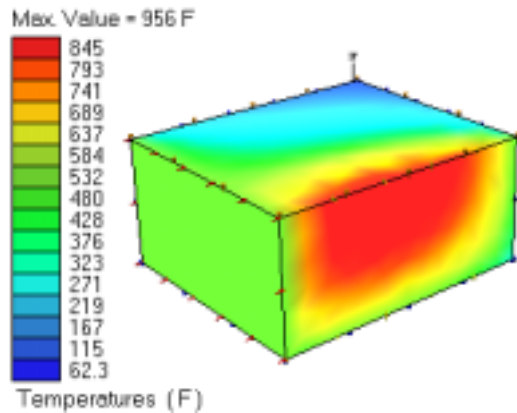
 Alternatively, **Simulation Settings** can be used to select the **Thermal** button for **FEA** simulation settings. See Figure 11-10.

Figure 11-10
FEA Mode in Settings Window



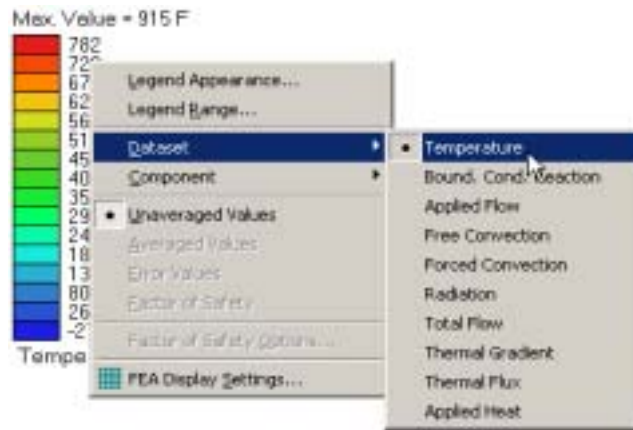
The **FEA Analysis Dialog** is displayed while MSC.Nastran performs a thermal simulation. When the analysis is complete, the results are displayed on the screen as shown in Figure 11-11.

Figure 11-11
Results of Heat Transfer FEA



35. To customize the display results, right-click the **FEA Contour Legend**. From the **Dataset** submenu, select the data to display.

Figure 11-12
Customizing Heat Transfer
Display



Exercise 11-2

Heat Transfer Analysis on an Assembly

Objectives

In this exercise you will learn to

- Rigidly join bodies.
- Run heat transfer FEA on an assembly.

Software

- MSC.visualNastran 4D, MSC.visualNastran Desktop FEA

Support File

- Tutorials\Chapter 11\Exercise 11.2\heat_transfer_assy.wm3

1. Launch MSC.visualNastran Desktop if not already open.
2. Choose **Open** from the **File** menu.
3. Browse the **Tutorials\Chapter 11\Exercise 11.2** folder and open the **heat_transfer_assy.wm3**

NOTE: Heat transfer analysis of an assembly requires special consideration at the bond face.

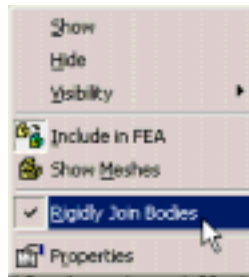
1. In the **Object List**, select the assembly.
2. Right-click and select **Include in FEA**.

Figure 11-13
Object Window



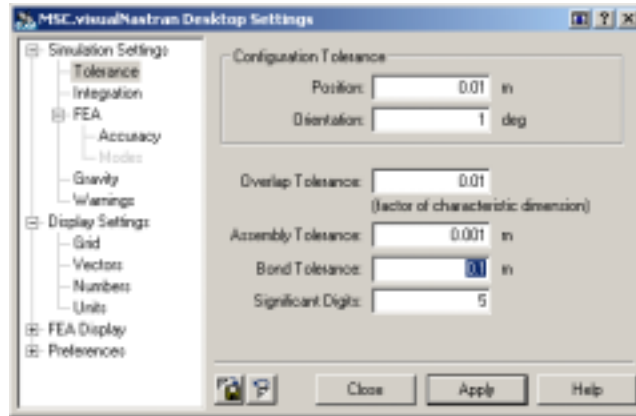
3. Right-click again, and select **Rigidly Join Bodies**.

Figure 11-14
Rigidly Join Bodies Pull down menu



4. Select the **Simulation Settings** button from the **View** toolbar.
5. Select **Tolerance** and enter 0.1 m in the **Bond Tolerance** box. This tolerance for the rigid bond between **body[1]** and **body[2]** encompasses both bodies.
6. Close the **Settings** dialog.

Figure 11-15
*Changing Bond Tolerance in
Settings Window*

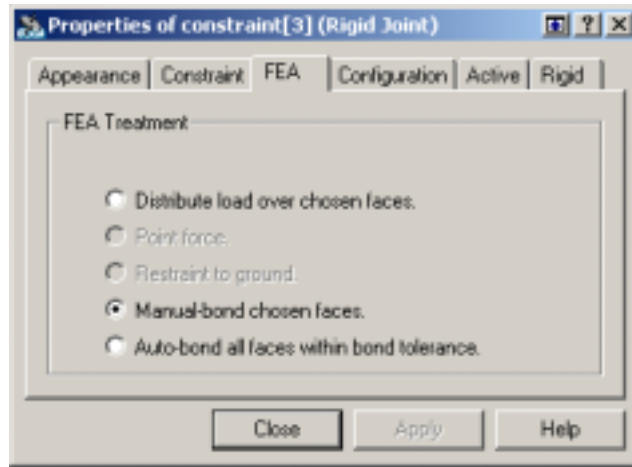


7. Double-click the rigid joint, **constraint[3]** to display its **Properties** window.
8. Click the **FEA** tab and select **Manual-bond chosen faces**. (If the FEA tab is not present in the **Properties** window, then select the FEA checkbox in the **Properties List** of the **Object Manager**.)

With the setting **Auto-bond all faces within the bond tolerance**, MSC.visualNastran Desktop will automatically select the two faces within the bond tolerance. With **Manual-bond chosen faces**, you can select any two surfaces independent of the bond tolerance.

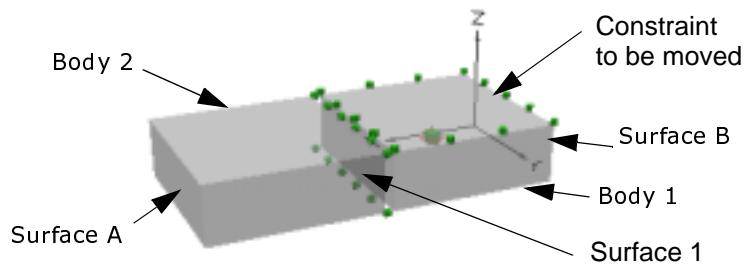
9. Close the **Properties** window.

Figure 11-16
Rigid Joint Constraint
Properties Window



One surface on each body will be high-lighted, as shown in Figure 11-17. If one of two faces high-lighted is not the desired face --- in this case, the mating faces of the assembly (Surface 1) --- you will be able to modify it in the following steps.

Figure 11-17
Moving the Bond Constraint



10. Right-click the bond constraint (labeled "Constraint to be moved" in Figure 11-17) on the upper surface of **body[1]**.
11. Select **Distribute on Face**.

Figure 11-18
Distribute on Face Pull down Menu



12. Click **Surface 1** (the mating face) to move the bond constraint.

13. Click the **Prescribed Temperature** button on **Thermal** toolbar.

14. Select **Surface A** as shown in Figure 11-17.

15. In the **Object List**, double-click the **Prescribed Temperature** constraint, **constraint[8]**, to open the **Properties** window.

16. Select the **Temperature** tab if not already selected.

17. Enter 1000 degrees and click **Close**.



18. Click the **Prescribed Temperature** button in the **Thermal** toolbar.

19. Select **Surface B** as shown in Figure 11-17. You may have to rotate the body to select the surface.

20. In the **Object List**, double-click the **Prescribed Temperature** constraint, **constraint[11]**, to open the **Properties** window.

21. Select the **Temperature** tab if not already selected.

22. Enter 0 degrees and click **Close**.



23. Click the **FEA Thermal Solve** button from the **Tape Player Control** to begin the analysis. You may first have to click the arrow to the right of the **FEA Solve** button to select the **Thermal** icon.

24. See Figure 11-19 for results of thermal FEA on an assembly and compare it to your own results.

Figure 11-19
*FEA Results of Heat Transfer
Analysis on an Assembly*

