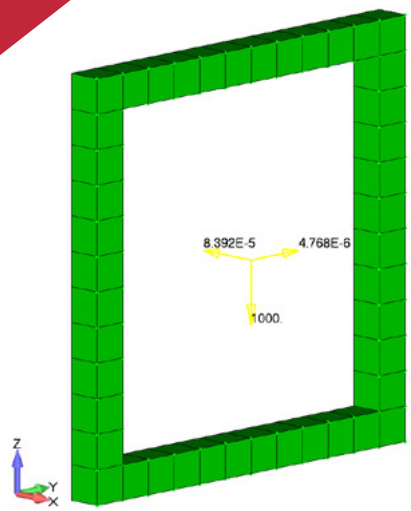
Femap II.I

WHITEPAPER

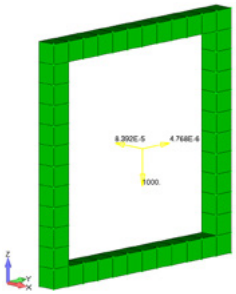
Freebody Section Forces in Femap



ATA ENGINEERING, INC.

www.ata-plmsoftware.com
844-756-7638 (844-PLM-SOFT)

Freebody Section Forces in Femap



Software:
Femap 11.1

Overview

Have you ever wanted to calculate the interface forces in a section of a structure? Have you ever wanted to take a portion of your finite element model and create a freebody diagram? Have you ever wanted to create a breakout model but been unsure about what boundary conditions to use? Femap is a powerful finite element analysis pre- and post-processor that provides these capabilities.

This whitepaper is part of a series of free Siemens PLM Software training resources provided by ATA. For more whitepapers, tutorials, videos, and macros, visit ATA's PLM Software website: http://www.ata-plmsoftware.com/resources.

Freebody Section Forces in Femap

Plotting Freebody Forces on Your Finite Element Model

This white paper begins with the assumption that you have a finite element model prepared for analysis and have an analysis set created in Femap. In this simple example, a 70-inch-long square box beam has been created with a wall thickness of one inch. The beam is fixed at one end and has a 1000 pound force applied at the other in the vertical direction. This model is shown in Figure 1.

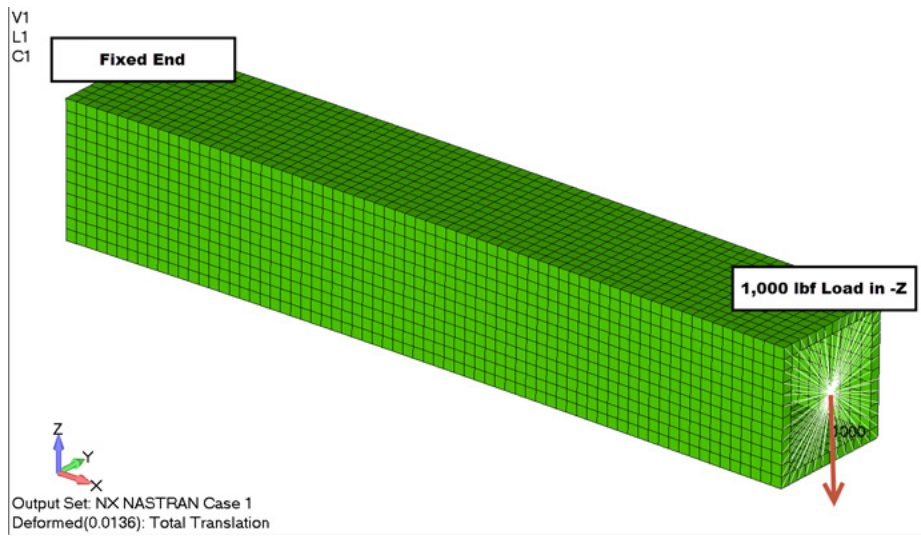


Figure 1: ▶
Cantilever 70-inch-long square box beam with one-inch-thick walls.

Before submitting the model for analysis, edit the output requests for the analysis set and turn on Force Balance output in the Nodal category. This is shown in Figure 2. Once this output request has been activated, submit the job for analysis and read the results into Femap.

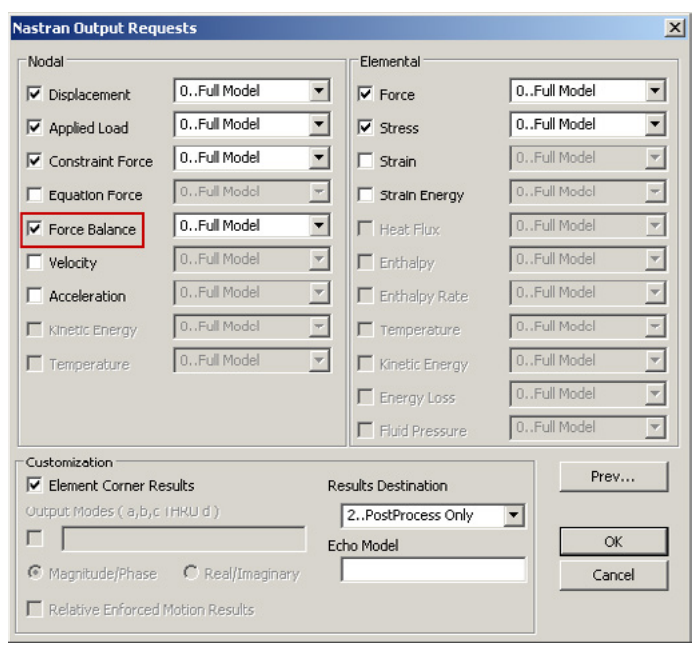


Figure 2: ▶
The nodal force balance output must be enabled to generate freebody section-cut diagrams.

Freebody Section Forces in Femap

When the results are available for post-processing, navigate into the PostProcessing Toolbox and expand the Freebody category. (The PostProcessing Toolbox is often accessible via a tab in the standard Model Info pane; if it is not visible, go to Tools – PostProcessing Toolbox to activate it.) Click the button boxed in Figure 3 to create a new freebody display. In the next window, click New Freebody, and then accept all the defaults on the following window, as shown in Figure 4. These defaults can be changed in certain circumstances to obtain proper output, but for this example the defaults will suffice.

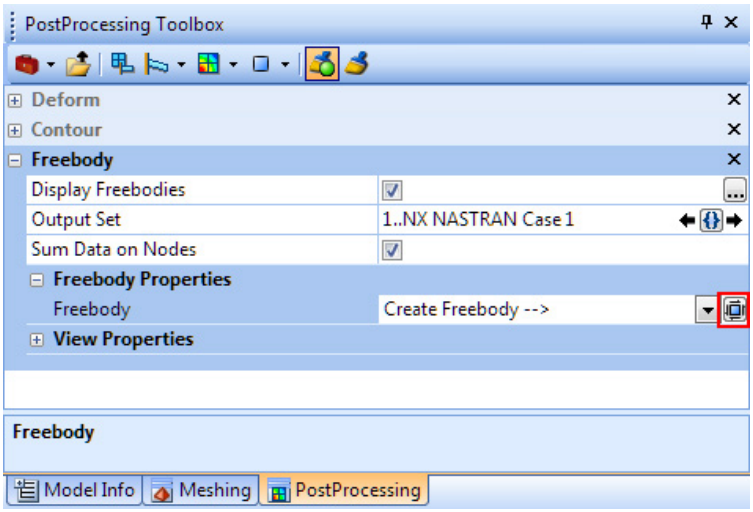


Figure 3: ▲
Creating a new freebody display in the PostProcessing Toolbox.

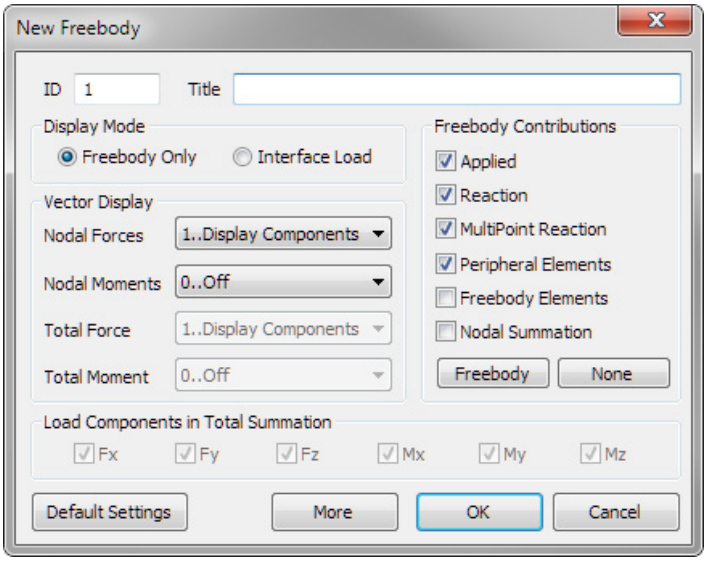


Figure 4: ▲
Freebody defaults.

Once you have accepted the defaults and created a new Freebody, the Freebody Properties window will show many more options. Several of the important options are briefly mentioned here. The coordinate system pull-down menu lets you select a coordinate system in which to transform the freebody loads; as the default is the basic coordinate system, you will want to change this option if your model is not aligned with the basic coordinate axes.

The Display Mode option lets you select either Freebody or Interface Load. Freebody displays freebody forces on all selected grid points, without calculating a resultant. The Interface Load selection displays resultant forces and moments at a specified location. The following example illustrates the difference between the two approaches.

Suppose we want to determine the internal forces at a location 24 inches from the fixed end of the beam. The elements at this location are shown in yellow in Figure 5, and a group that contains these elements and the nodes on the front face of these elements can be created.

Freebody Section Forces in Femap

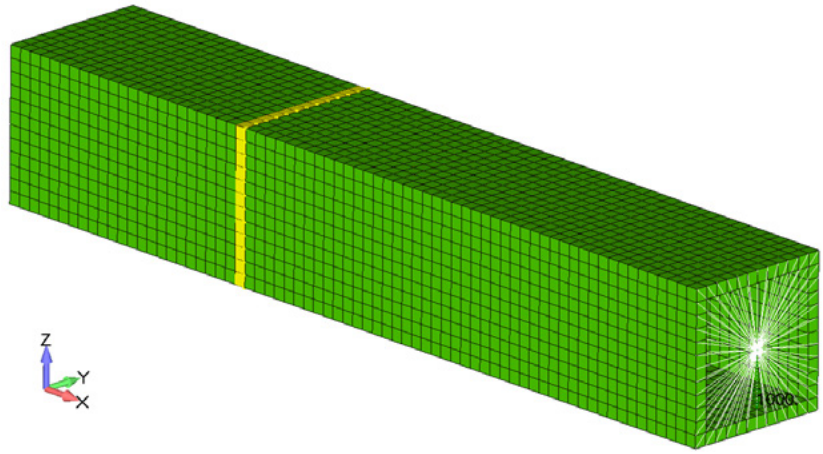


Figure 5: ▶
Group of elements and nodes located 24 inches from the fixed end of the cantilevered beam.

In the PostProcessing Toolbox, click the Freebody Elements selection button as shown in Figure 6 and use the group created above to easily input the desired elements. After clicking OK in the "Entity Selection" dialog box, arrows will pop up in your display. Depending on your screen resolution, this collection of arrows may appear cluttered, so we can zoom in on just one corner of the group of elements in more detail.

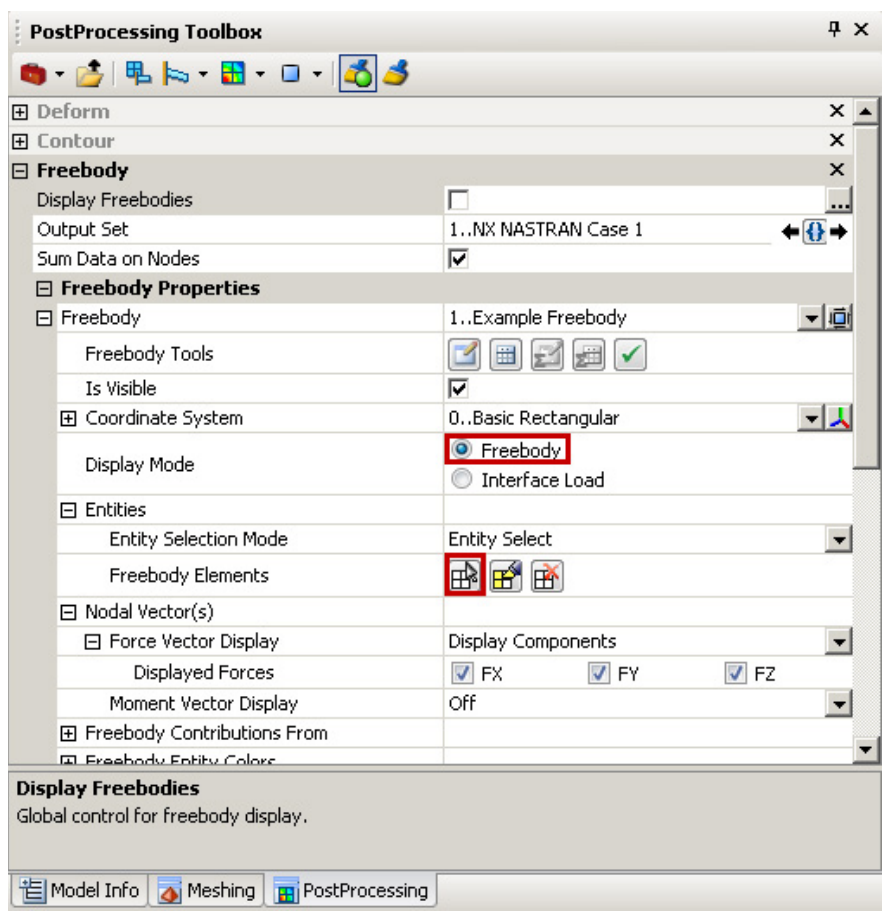


Figure 6: ▶
With the display mode set to "Freebody," the desired elements to consider in the freebody calculations must be selected.

Freebody Section Forces in Femap

This zoomed-in corner of the box beam is shown in Figure 7. Because this model is composed of 3D solid elements, each node has three force vectors associated with it. If we had been using a different element formulation (e.g., shell elements), there would also be moments associated with each node. Assuming the defaults were selected as shown in Figure 4, the value shown adjacent to each arrow represents the force on that node exerted by all other adjacent elements not in the freebody group. One observation that can be made in this example is that the out-of-plane forces are larger at the outer top surface of the beam (145.68 lbf) than they are at the inner top surface of the beam (137.11 lbf), which agrees with the theoretical expectation for a beam in bending where the normal stresses are maximum at the extreme top and bottom surfaces.

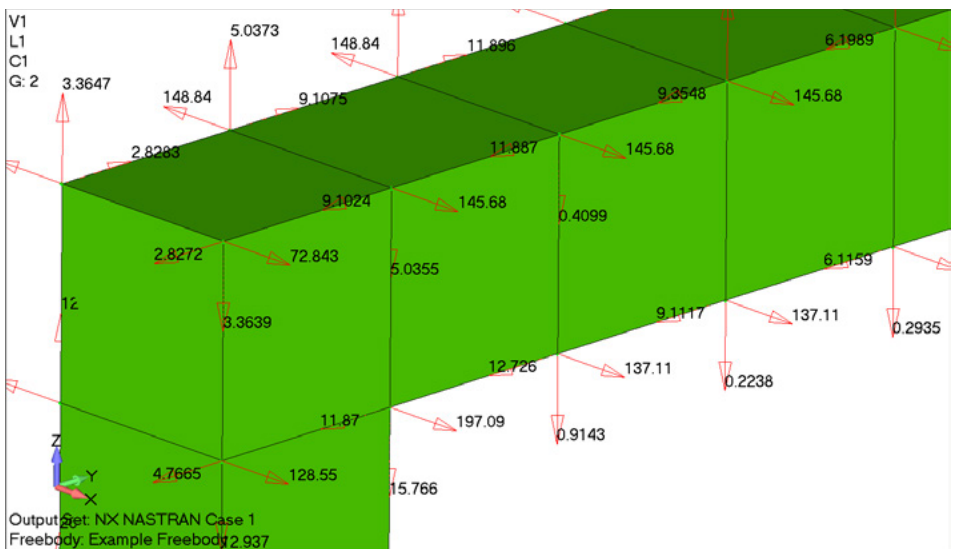
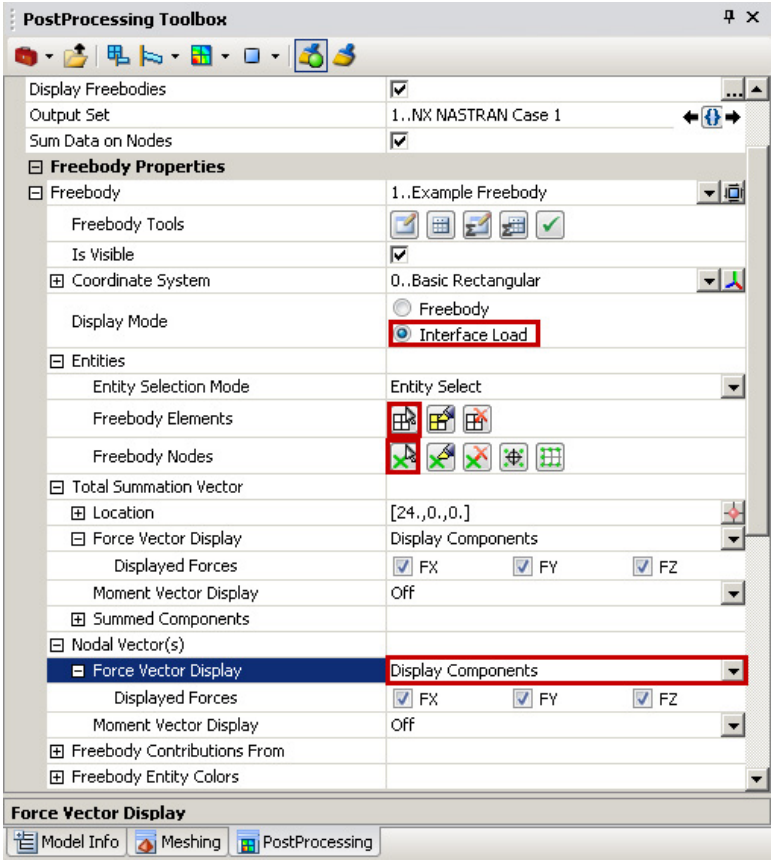


Figure 7: ▶
Corner of box beam showing the output of the Freebody display tool for the Freebody display mode.

The previous example used the Freebody display mode, which shows the forces on all nodes attached to the elements selected. The Interface Load display mode allows you to be more selective and see only the forces on specific nodes attached to elements you have selected. When you change the display mode from Freebody to Interface Load, you must select nodes in addition to elements. The appearance of the Interface Load display is shown in Figure 8.

Freebody Section Forces in Femap

Figure 8: ▶
With the display mode set to "Interface Mode," the desired nodes and elements to consider in the freebody calculations must be selected.



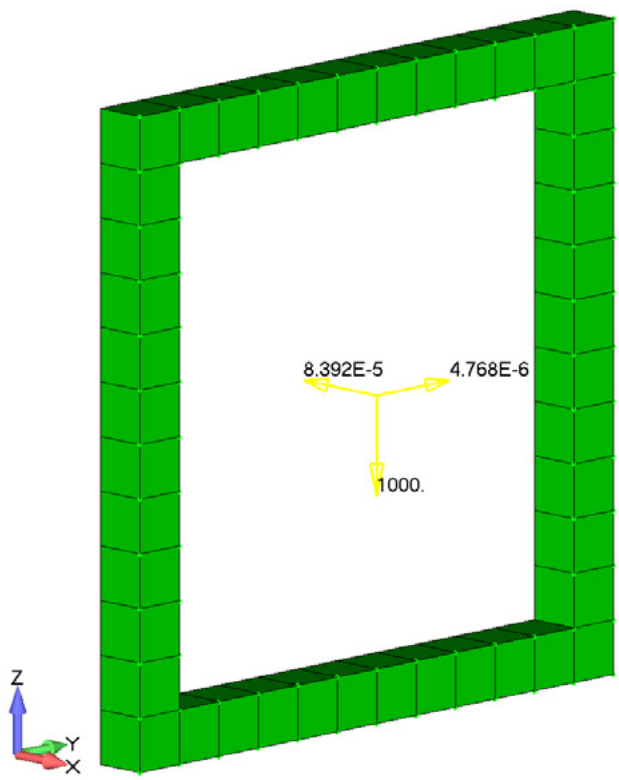
In the node and element entity selection dialog boxes, we can use the group we created previously. Also shown in the figure is the Nodal Force Vector Display. Set this to Off in the corresponding pull-down menu; this will shut off the vector display at every node. If you were to leave this option set to On, the graphics window display would appear identical to Figure 7. It is pointed out here because you may be interested in this display in the future, but we will turn it off for the sake of illustrating a unique feature of the Interface Load mode.

After you select the nodes, Femap will ask whether you want to calculate the summation vectors about a point located at the average position of all the nodes you have selected. If your FEM is symmetric, as in this example, this is an acceptable choice to make. However, if your mesh is asymmetric (e.g., if your mesh is biased to one side or the other of your cross-section), then you may want to manually place the summation point at the centroid of the cross-section.

Freebody Section Forces in Femap

Once you have accepted all values in the dialog boxes and returned to the PostProcessing Toolbox, the graphics window should look similar to Figure 9. The vectors in this figure show the resultant forces summed from all nodes that were previously selected. Because we selected all nodes that exist on this cross-section, the force in the Z direction is −1000 lbf, which is equal to the applied load on the beam. For an externally applied shear load, this is exactly the result predicted by beam theory. You can also visualize internal moments in a similar fashion by setting the Moment Vector Display in the Freebody Properties window.

Figure 9: ▶
Resultant internal forces at an interface located 24 inches from the fixed end of the cantilever beam.



Conclusion

This white paper demonstrates a powerful Femap capability that can be used to display internal section forces and moments in a structure. This tool allows you to examine load transfer paths or to examine the forces across a specific interface in your structure. A useful extension of this tool is to determine section loads at an interface of interest in your model and then use those loads as boundary conditions in another model. Such information can be used to evaluate load transfer paths or develop the acceleration of breakout models.

www.ata-plmsoftware.com

ATA Engineering

ATA ENGINEERING, INC.

Solution Partner

PLM

SIEMENS

www.ata-e.com

ata-engineering

@ataengineering

sales@ata-e.com

858.480.2000

www.ata-plmsoftware.com

San Diego
Corporate Headquarters

Albuquerque

Denver

Huntsville

Los Angeles

San Francisco

Washington, D.C.