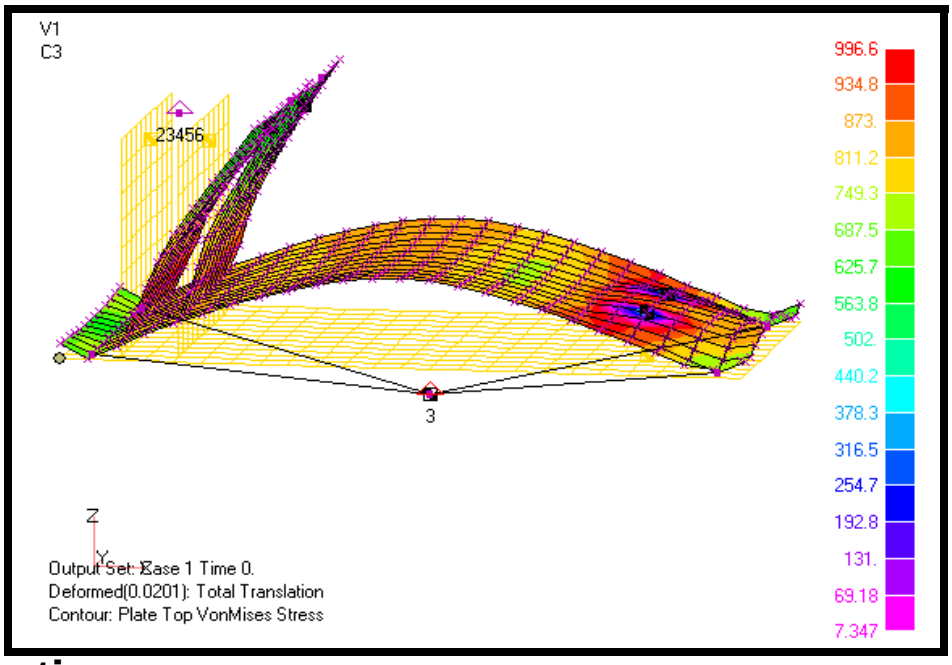
WORKSHOP 13b

Response Spectrum Application



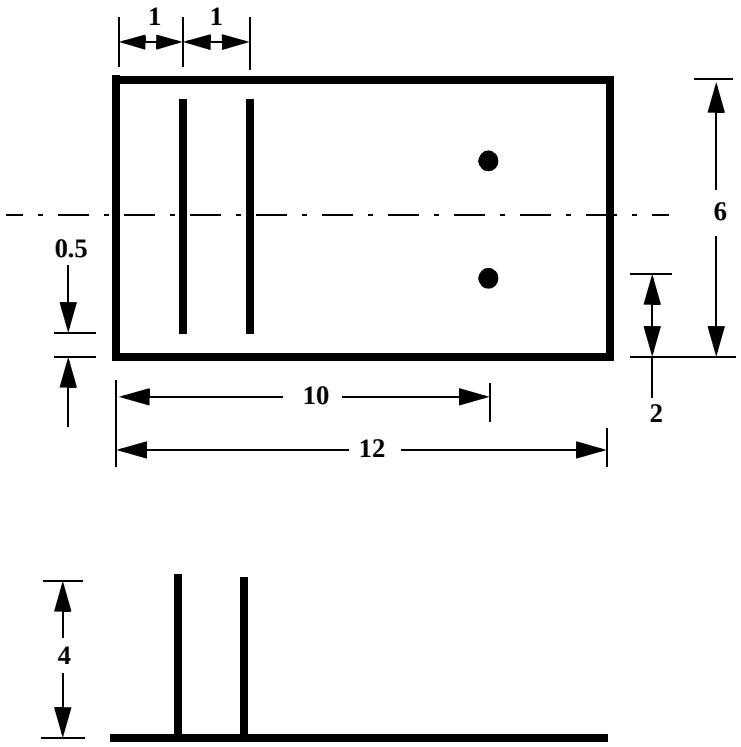
Objectives:

- Open an existing model of a circuit board.
- Create boundary conditions for a response analysis.
- Import the shock response spectra from a previous analysis.
- Run a modal analysis with shock response spectra applied.
- View the results.

Model Description:

Starting with the circuit board model from workshop problem 12a, a shock response analysis will be performed.

Figure 13b.1 - Printed Circuit Board Dimensions



The printed circuit board is attached to a computer chassis at its corners. The chassis properties data are given below. Determine the first 10 modal frequencies of the circuit board.

Large Mass Properties:

Mass, M = 10000 lbs

Exercise Procedure:

1. Start up MSC.Nastran for Windows V4.0 and open an existing model.

   Double click on the icon labeled **MSC.Nastran for Windows V4.0.**

   Open the model created in workshop 12a. On the *Open Model File* form, select the file and **Open.**

   Filename: **Circuit_board.MOD**

   Open

2. Change the view of the model to wireframe.

   View/Select... (F5)

   Model Style: ● **Draw Model**

   Deformed Style: ● **None**

   Contour Style: ● **None**

   OK

   View/Rotate... (F8)

   ZX Front

   OK

   File/Save As...

   Filename: **prob13b**

   OK

3. Create a node that will be used for a large base mass.

   Model/Node...

   X: **6** *Y:* **3** *Z:* **-1**

   ID: **2000**

   Parameter...

   ❑ TX ❑ TY ❑ TZ ❑ RX ❑ RY ❑ RZ

   OK

OK

Cancel

4. Create a property for the large base mass.

   From the pulldown menu, select **Model/Property**.

   Model/Property...

   Title: **large_mass**

   Elem/Property Type...

   Other Elements: ● **Mass**

   OK

   Mass, M or Mx: **10000**

   OK

   Cancel

   Model/Element...

   Type...

   Other Elements: ● **Mass**

   OK

   ID: **2000**

   Property: **4.. large_mass**

   Node: **2000**

   OK

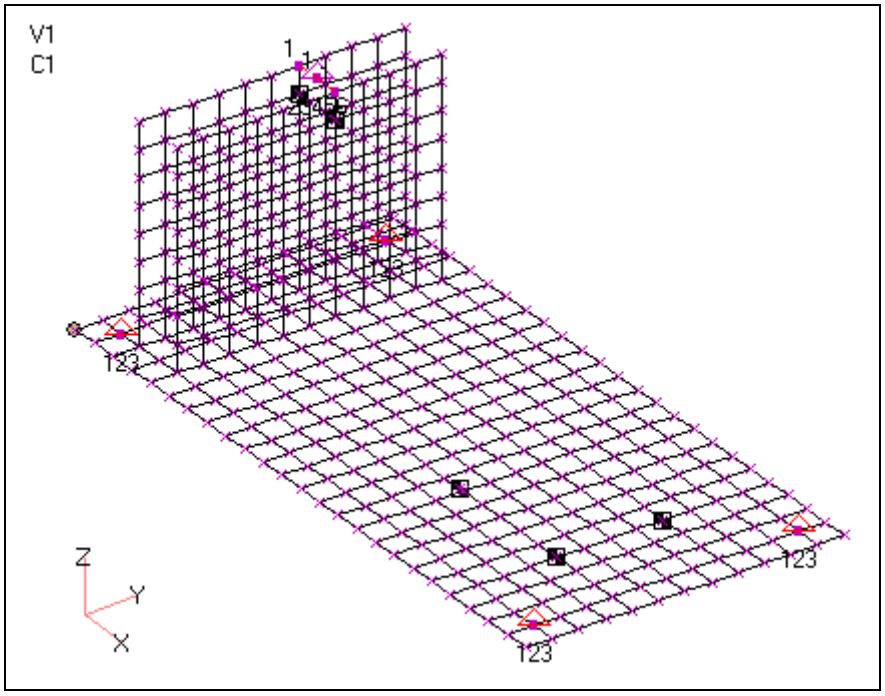
   Cancel

5. Connect the node to the circuit board.

   View/Rotate... (F8)

   Dimetric

   OK



Model/Element...

Type...

Other Elements: ● **Rigid**

OK

Independent: Node: **2000**

☒ **Tx** ❑ **Rx**
☒ **Ty** ❑ **Ry**
☒ **Tz** ❑ **Rz**

Nodes...

<select the 4 node locations on the circuit board that contain the pinned single point constraints - ID #'s 73, 95, 303, 325>

OK

OK

Cancel

6. Define the constraints on the node with the large mass.

Model/Constraint/Set...

ID: **2**

Title: **free_z**

OK

Model/Constraint/Nodal...

<select node 2000>

OK

☒ Tx ☒ Ty ❑Tz
☒ Rx ☒ Ry ☒ Rz

OK

Cancel

7. Create a constraint set to “support” rigid body motion.

Model/Constraint/Set...

ID: **3**

Title: **support_z**

OK

Model/Constraint/Nodal...

<select large mass node - ID # 2000>

OK

❑ Tx ❑ Ty ☒ Tz
❑ Rx ❑ Ry ❑ Rz

OK

Cancel

8. Import the response spectra from problem 13a.

File/Import/FEMAP Neutral...

Filename: **prob_13a_functions.NEU**

OK

OK

Six functions were imported. The response spectra were functions 4, 5, and 6. Recall that the damping for these functions were 0., 0.025, and 0.05, respectively. We must now create a function to define this relationship.

9. Define function number vs. critical damping and modal damping for the response analysis.

Model/Function...

Title: **function_vs_damping**

Type: **16.. Function vs. Critical Damping**

ID:	**4**	*Y:*	**0**	**More**
ID:	**5**	*Y:*	**0.025**	**More**
ID:	**6**	*Y:*	**0.05**	**More**

OK

Title: **modal_damping**

Type: **7.. Critical Damping vs. Freq**

X:	**1**	*Y:*	**0.01**	**More**
X:	**10**	*Y:*	**0.01**	**More**
X:	**20**	*Y:*	**0.02**	**More**
X:	**50**	*Y:*	**0.05**	**More**
X:	**100**	*Y:*	**0.05**	**More**

OK

Cancel

10. Submit the file for analysis.

File/Export/Analysis Model...

Type: **2..Normal Modes/Eigenvalue**

OK

Filename: **Prob13b**

Write

☒ **Run Analysis**

Advanced...

Response Spectrum...

Spectrum: ● **Acceleration**

Spectrum Function ID: **7.. function_vs_damping**

SUPORT set: **3.. support_z**

Damping Func: **8.. modal_damping**

OK

Eigenvalues and Eigenvectors

Number Desired: **11**

OK

OK

In the **NASTRAN Case Control** window under *Output Requests*, turn all requests off except:

☒ **Displacement** ☒ **Acceleration**

☒ **Constraint Force** ☒ **Element Stress**

☒ **Constraint** **2.. Free_z**

OK

☒ **WTMASS** **0.00259**

OK

Yes

11. Postprocess the results.

View/Select... (F5)

Deformed Style: ● **Deformed**

Contour Style: ● **Contour**

Deformed and Contour Data...

Use the pulldown menus to make the following selections.

Output Set:	**11.. Case 1 Time 0.**
Deformation:	**1..Total Translation**
Contour:	**7033.. Plate Top Von Mises Stress**

OK

OK

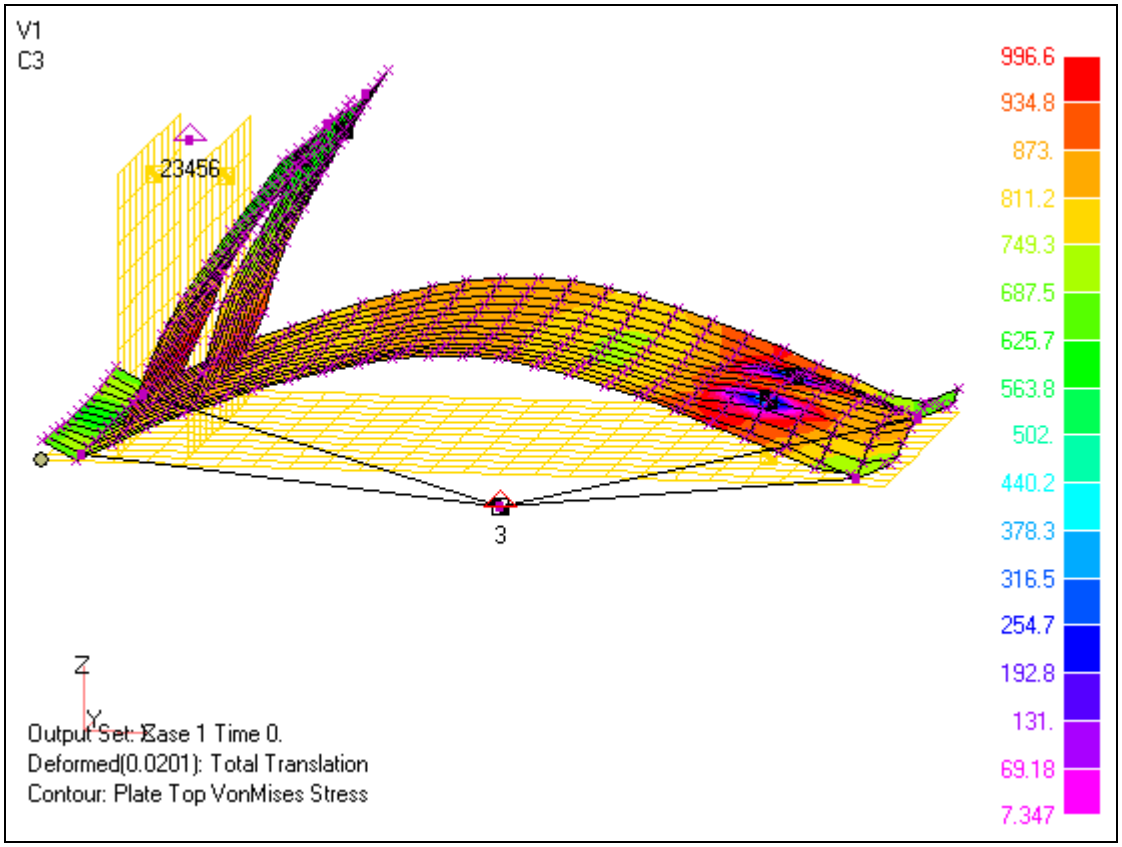
View/Rotate... (F8)

X: **-80**

Y: **0**

Z: **-10**

The results should look like the following:



When finished, exit MSC.Nastran for Windows.

File/Exit

Yes

This concludes the exercise.