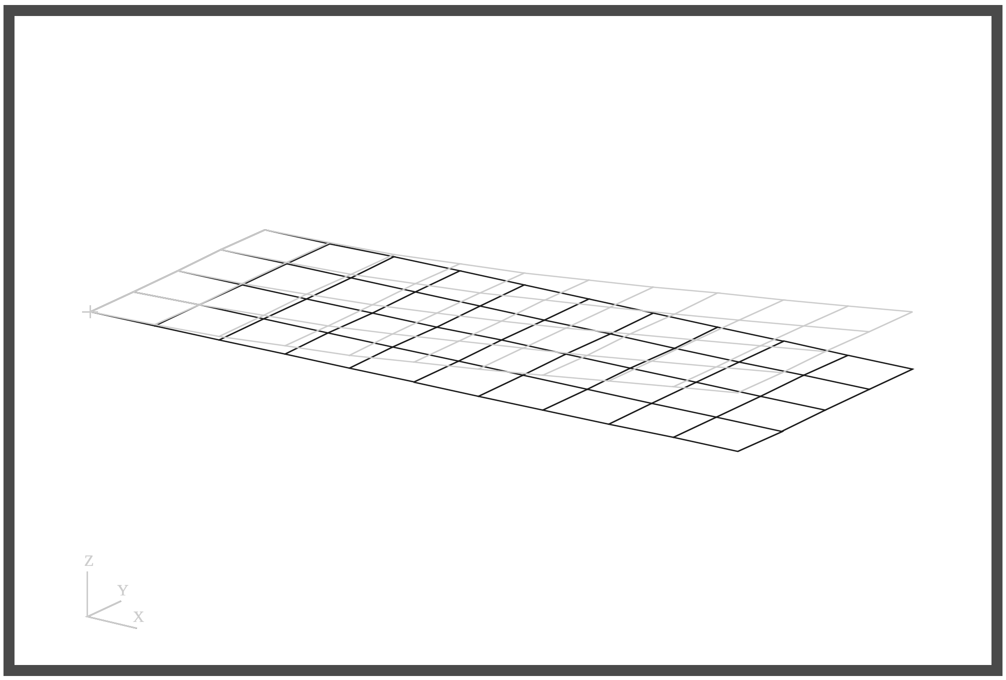
WORKSHOP PROBLEM 10

Modal Random Response Analysis



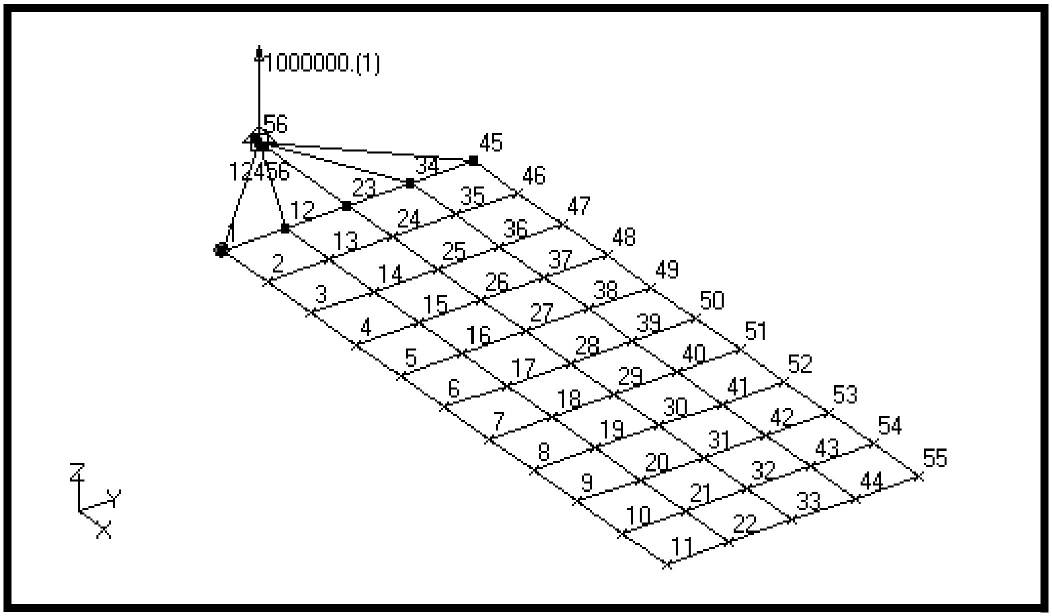
Objectives

- Import analysis model of a flat rectangular plate.
- Define an input acceleration power spectrum.
- Use the large mass method for enforced acceleration.
- Submit the analysis in MSC.Nastran for Windows.
- Compute response power spectra.

Model Description:

Using the modal method, determine selected random response spectra of the flat rectangular plate created in Workshop1, excited by the input acceleration spectra density defined below. Use the modal damping table also defined below. Use a frequency step of 20 Hz between 20 and 1000 HZ; in addtion, specify five evenly spaced excitation frequencies beween ±5% of the natural frequencies between 20 and 1000 Hz.

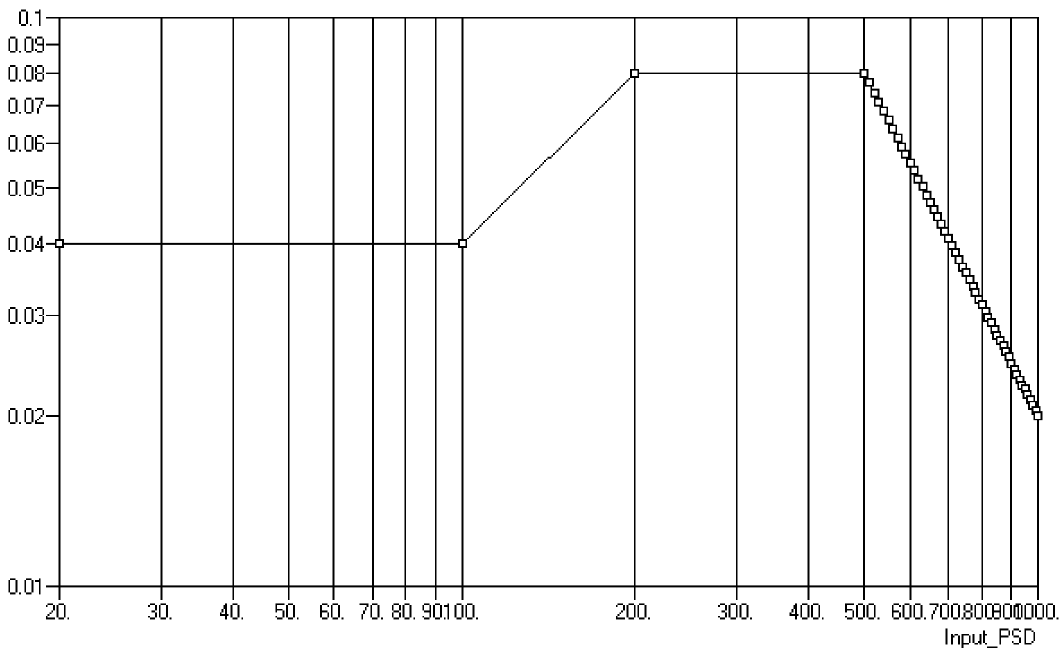
Figure 10.1 - Loads and Boundary Conditions



The input acceleration spectra density in the Z direction is defined by:

Frequency, Hz	Level/Slope
20 - 100	0.04 g^2/Hz
100 - 200	+3 db/octave
200 - 500	0.08 g^2/Hz
500 - 1000	-6 db/octave
1000	0.02
Overall	7.29 g RMS

Figure 10.2 - Input Acceleration Spectra

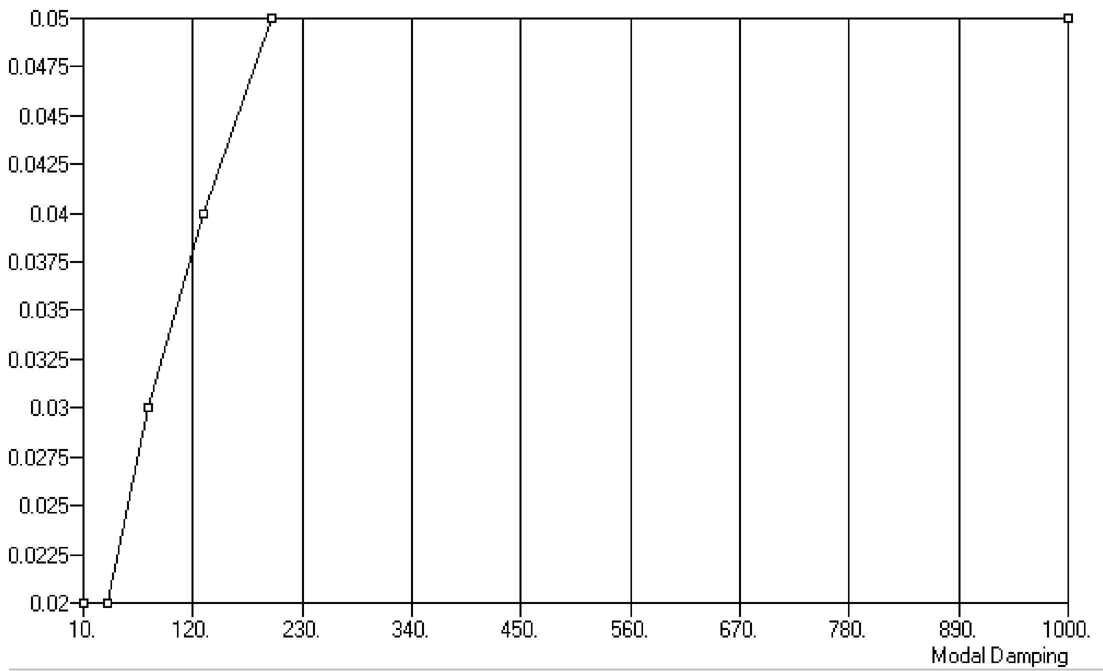


Note: All segments are linear on a log-log plot.

The modal damping is defined as:

f	ζ
10	0.02
35	0.02
75	0.03
130	0.04
200	0.05
1000	0.05

Figure 10.3 - Modal Damping



Exercise Procedure:

1. Start up MSC.Nastran for Windows 4.0 and begin to create a new model.

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

On the *Open Model File* form, select **New Model.**

Open Model File:	**New Model**

2. Import **prob1.DAT.**

File/ Import/Analysis Model...

● **Nastran**	**MSC/Nastran**

OK

Change the directory to **C : \temp**.

File name:	**prob1.DAT**

Open

When asked, "Ok, to Adjust all massess by PARAM, WTMASS factor of 0.00259?", answer **No**. This information will be entered during analysis.

No

To reset the display of the model do the following:

View/Redraw

View/Autoscale

View/Rotate...	**Dimetric**

OK

3. Create the frequency dependant functions for the input load, PSD and modal damping.

Model/Function...

Title:	**Unit Load**

To select the type, click on the list icon next to the data box and select vs. Frequency.

Type: **3..vs. Frequency**

Data Entry: ● **Single Value**

X: 0 **Y: 1**

More

X: 1000 **Y: 1**

More

OK

ID: **2**

Title: **Input_PSD**

Type: **3..vs. Frequency**

Data Entry: ● **Single Value**

X: 20 **Y: 0.04**

More

X: 100 **Y: 0.04**

More

X: 200 **Y: 0.08**

More

This equation is required to achieve a line on a log-log plot from 500 to 1000 Hz.

Data Entry: ● **Equation**

Delta X: **10**

X: 500 **Y: 0.08*SQR(500/!x)**

To X: 1000

More

OK

ID: **3**

Title: **Modal Damping**

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

Data Entry: ● **Single Value**

X: 10 **Y: 0.02**

More

X: 35 **Y: 0.02**

More

X: 75 **Y: 0.03**

More

X: 130 **Y: 0.04**

More

X: 200 **Y: 0.05**

More

X: 1000 **Y: 0.05**

More

OK

Cancel

Now, plot these functions.

View/Select...

XY Style ● **XY Function**

Model Data

Function **2..Input_PSD**

OK

OK

View/Options...

Catagory **PostProcessing**

Options	**XY Y Range/Grid**
Axis Range	**2..Max Min**
Minimum	**0.01**
Maximum	**0.1**
Apply	
Options	**XY Axis Style**
Plot Type	**2..Log-Log**
Apply	
Options	**XY Curve 1**
Curve Style	**2..Lines with Points**
Apply	
OK	

Refer to Figure 10.2 to verify the input. Next, plot the Modal Damping input.

View/Select...	
XY Style	● **XY Function**
Model Data	
Function	**3..Modal Damping**
OK	
OK	
View/Options...	
Catagory	● **PostProcessing**
Options	**XY Axis Style**
Plot Type	**0.. Rectilinear**
Apply	
Options	**XY Y Range/Grid**
Plot Type	**0..Automatic**
Apply	

OK

Refer to Figure 10.3 to verify the input

4. Create the Base Mass and load for a one g enforced acceleration.

View/Select...

Model Style ● **Draw Model**

OK

Model/Load/Set...

Title: **1..Z Direction Base Shake**

OK

Model/Load/Dynamic Analysis...

Solution Method: ● **Modal Frequency**

Modal Damping Table: **3.. Modal Damping**

PSD: **2.. Input PSD**

Advanced...

Mass Formation: ● **Coupled**

OK

Enforced Motion...

Enter coordinates for Base Mass.

X: **-1** **Y:** **1** **Z:** **0**

OK

Select Nodes on Base.

Select Nodes: **1, 12, 23, 34, 45**

OK

Create loads on Nodes.

Acceleration ☒ **AZ:** **1**

Function Dependence: **1.. Unit Load**

OK

Mass/Accel Scale Factor...

Select the new node, refer to Figure 10.1

Mass: **1.0**

Factor: **1000000**

OK

OK

Create a new constraint set on the base mass that is free in the Z direction.

Model/Constraint/Set...

ID: **2**

Title: **Z Direction Base Shake**

OK

Model/Constraint/Nodal

ID: **56**

OK

DOF: ☒ **TX** ☒ **TY** ☐ **TZ**

☒ **RX** ☒ **RY** ☒ **RZ**

OK

Cancel

View/Redraw to see the current load set and constraint set.

5. Run a normal modes solution to create a list of natural frequencies between 20 and 1000 Hz.

File/Export/Analysis Model...

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

OK

Change the directory to **C : \temp**.

File name:	**prob10_modes**
Write	
	☒ **Run Analysis**
Advanced...	
Modal Solution Method:	● **Lanczos**
Range of Interest to:	**1000**
Mass:	● **Coupled**
OK	
Problem ID:	**Modal Analysis of a Plate**
OK	

Under *Output Requests*, unselect all except:

	☒ **Displacement**
OK	

Under *PARAM*, enter the following:

☒ *WTMASS*	**.00259**
OK	

When asked if you wish to save the model, respond **Yes**.

Yes	
File name:	**prob10**
Save	

When the MSC.Nastran manager is through running, MSC.Nastran for Windows will be restored on your screen, and the *Message Review* form will appear. To read the messages, you could select **Show Details**. Since the anaysis ran smootly, we will not bother with the details this time.

Continue

6. Create the table of solution frequencies based on the normal modes we just computed.

Model/Load/Dynamic Analysis...

Modal Freq...

Modal Results

First Freq: **2.. Mode 2 133.6996 Hz**

Last Freq: **4.. Mode 4 843.8918 Hz**

Number of Points per Existing Mode: **5**

Frequency Band Spread (+/-): **5**

OK

OK

Now add additional solution frequencies every 20 Hz from 20 to 1000Hz.

Modify/Edit/Function...

Entitiy ID: **4..Modal Frequency Table**

OK

Data Entry: ● **Linear Ramp**

Delta X: **20**

X **20** **Y**

To X **1000** **Y**

More

OK

7. Now, run the random response analysis.

File/Export/Analysis Model...

Type: **6.. Random Response**

OK

Change the directory to **C : \temp**.

File name: **prob10**

Write

☒ **Run Analysis**

Advanced...

Select Load Sets to Analyze:

ID: **1**

OK

Under Range of Interest Enter.

from (Hz): **0**

to (Hz): **2000**

OK

OK

Under output request unselect all entities.

OK

In the **NASTRAN Output for Random Analysis** form, **Nodal Output Requests** select:

Acceleration: ☒ **T3**

In the **Element Stresses** section under **Plates**:

Bottom: ☒ **X Normal**

OK

Select Nodes for output:

Select All

OK

Select Elements for output:

Select All

OK

Under *PARAM*, enter:

☒ *WTMASS* **0.00259**

OK

When asked if you wish to save the model, respond **Yes**.

Yes

When asked if OK to read PSD vs. Frequency Function, respond **Yes**.

Yes

When the MSC.Nastran manager is through running, MSC.Nastran will be restored on your screen, and the *Message Review* form will appear. To read the messages, you could select **Show Details**. Since the anaysis ran smoothly, we will not bother with the details this time.

Continue

8. To view the results as a contour plot:

View/Select...

Contour Style: ● **Contour**

Deformed and Contour Data...

Output Set: **5.. X-Y Plot Summary**

Contour: **1581..T3 Acceleration RMS**

OK

OK

Use the Dynamic Query Tool to find the T3 Acceleration RMS Level of Nodes 56, 1, 11 and 33. To turn on the Query Tool click on the button in the lower right corner of the window that has the label of OFF. Select the Node tool then hold the cursor over the selected Nodes.

Node	T3 Acceleration RMS
56	__________
1	__________
11	__________
33	__________

Note: These are in the units of in/sec^2 not g!

To view stress RMS levels as a contour plot:

View/Select...

Deformed and Contour Data

Contour: **87321..Plate Bot X Stress RMS**

OK

OK

Use the Dynamic Query tool to find the Plate Bot X Stress RMS at Nodes 23, 26 and 29.

Node	Plate Bot X Stress RMS
23	__________
26	__________
29	__________

Turn off the Dynamic Query Tool.

9. Finally, to create XY Plots from some of the Response Power Spectra.

View/Select...

Under **XY Style** pick:

● **XY of a function**

Model Data...

Under **Function**, Select: **60. ACC3 PSD Node 56**

OK

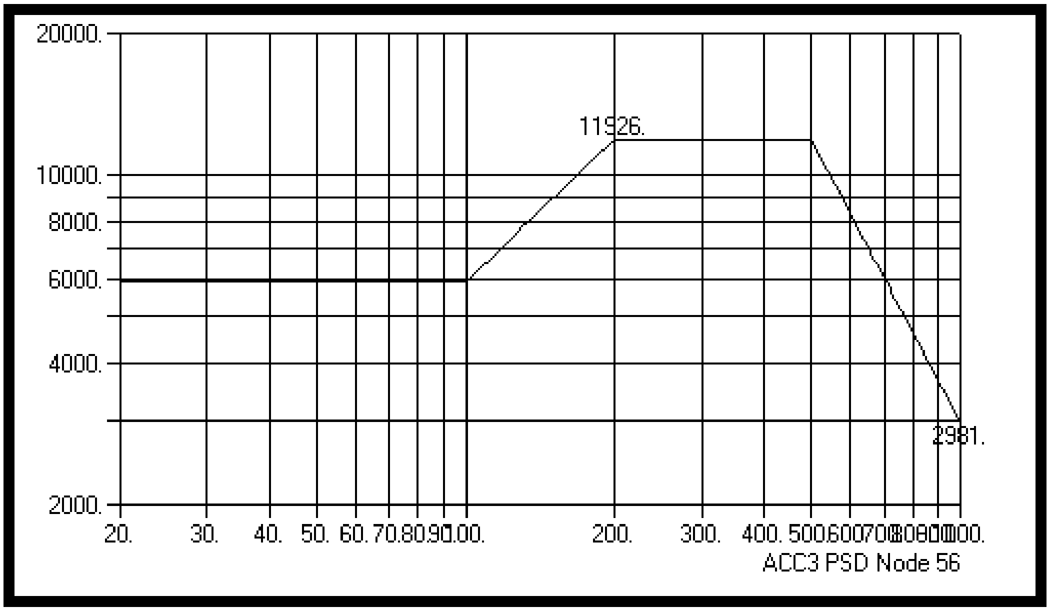
OK

To convert the plot to log-log, select:

View/Options...

Chategory:	● **Postprocessing**
Options:	**XY Axes Style**
Plot type:	**2.. Log log**
Apply	
Options:	**XY Curve1**
Data Labels:	**4.. Max/Min Value**
Curve Style	**1..Lines**
Apply	
Options:	**XY Y Range/Grid**
Axis Range:	**2.. Max Min**
Minimum:	**2000**
Maximum:	**20000**
OK	

Figure 10.4 - <u>T3 Acceleration Spectrm for Node 56</u>



To see the T3 Acceleration PSD for Node 11 Select:

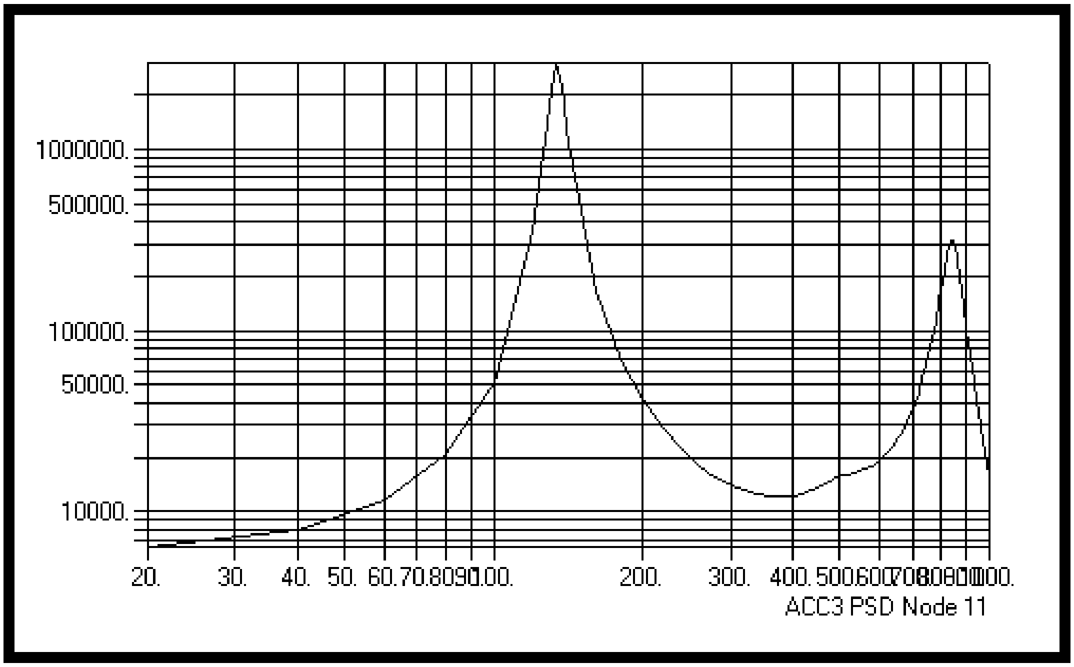
View/Select...

Model Data...	
Select:	**15.. ACC3 PSD Node 11**
OK	
OK	

View/Options...

Options:	**XY Y Range/Grid**
Axis Range:	**0.. Automatic**
Options:	**XY Curve 1**
Data Labels:	**0..No Labels**
OK	

Figure 10.5 - T3 Acceleration for Node 11



To plot the Plate Bot X Stress PSD select:

View/Select...

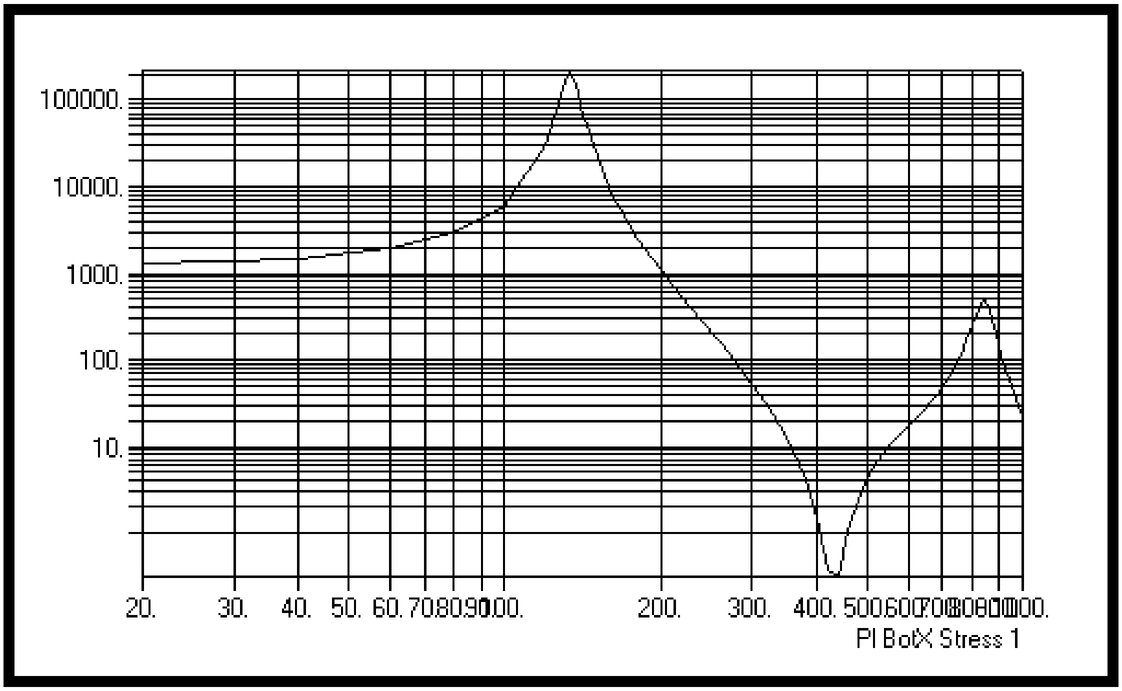
| **Model Data...** |

Select: | **61.. PL Bot X Stress 1** |

| **OK** |

| **OK** |

Figure 10.6 - Plate Bottom X Stress Spectrum



When finished, exit MSC.Nastran for Windows.

File/Exit

This concludes this exercise.

Node	T3 Accel RMS
56	**2816.369**
1	**2816.369**
11	**10037.80**
33	**10160.67**

Node	Plate Bot X Stress RMS
23	**2329.982**
26	**1381.136**
29	**598.5251**