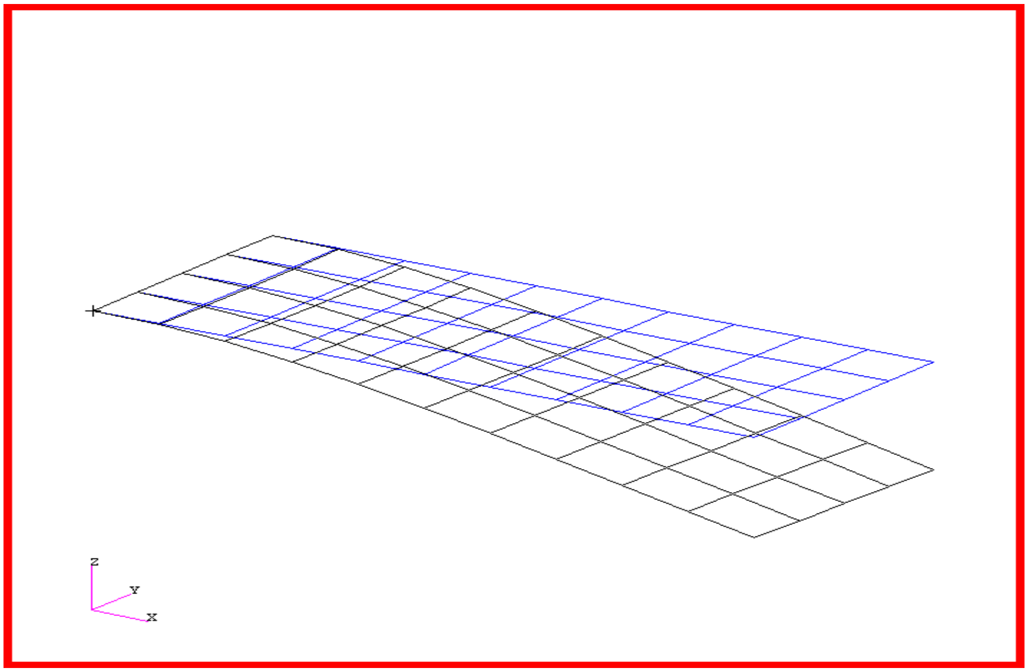
WORKSHOP PROBLEM 6

Modal Frequency Response Analysis



Objectives:

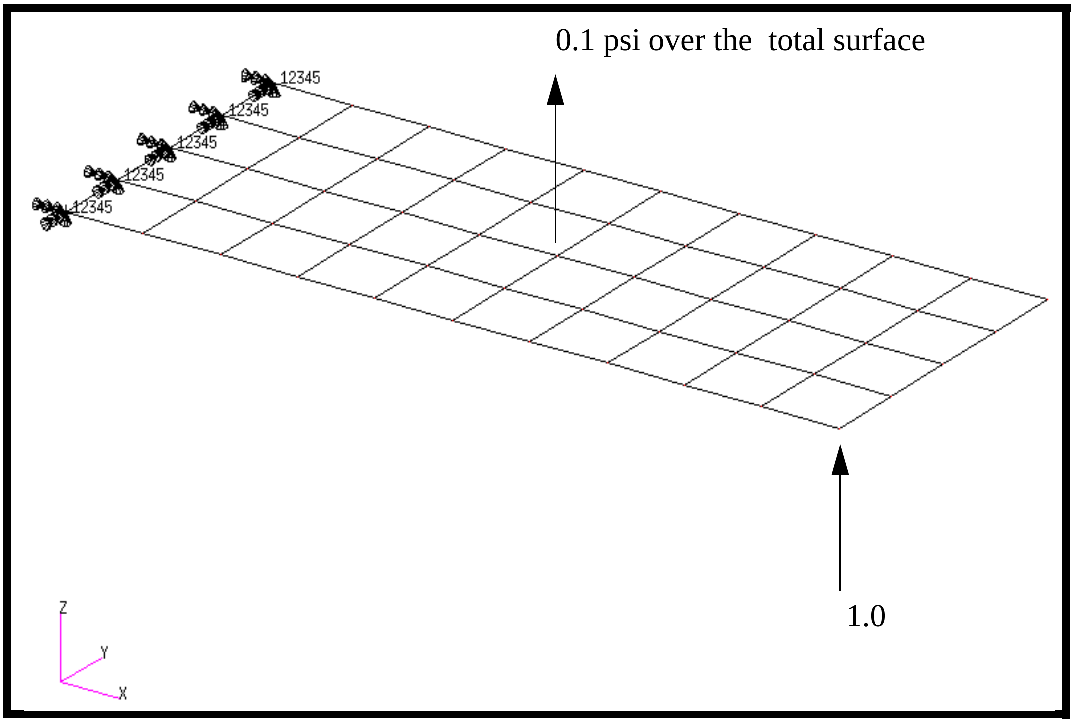
- Create a geometric representation of a flat rectangular plate.
- Use the geometry model to define an analysis model comprised of plate elements.
- Define a frequency-varying excitation.
- Submit the file for analysis in MSC.Nastran for Windows.
- Compute nodal displacements for desired frequency domain.

Model Description:

Using the modal method, determine the frequency response of the flat rectangular plate, created in Workshop 1, excited by a 0.1 psi pressure load over the total surface of the plate and a 1.0 lb. force at a corner of the tip lagging 45^o. Use a modal damping of $\xi = 0.03$. Use a frequency step of 20 hz between a range of 20 and 1000 hz; in addition, specify five evenly spaced excitation frequencies between the half power points of each resonant frequency between the range of 20-1000 hz.

Below is a finite element representation of the flat plate. It also contains the loads and boundary constraints.

Figure 6.1- Loads and Boundary Conditions



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 ask, "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. Run Normal Modes to create a list of natural frequencies between 20 and 1000 hz..

File/Export/Analysis Model...

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

OK

File name: **prob6_modes**

Write

☒ **Run Analysis**

Advanced...

Modal Solution Method: ● **Lanczos**

Range of Intrest 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

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

Yes

File name: **prob6**

Save

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 analysis ran smoothly, we will not bother with the details this time.

Continue

5. Create the frequency dependent function for the frequency response of the unit load.

Model/Function...

Title:	**frequency_varying_load**

To select the type, click on the list icon next to the databox and select **vs. Frequency.**

Type:		**3..vs. Frequency**	
Data Entry:		● **Single Value**	
X	**0**	**Y**	**1**
More			
X	**1000**	**Y**	**1**
More			
OK			

Create the frequency dependent critical damping.

ID:	**2**
Title:	**Modal_damping**

To select the function, click on the list icon next to the databox and select **Critical damping vs. Freq.**

Type:		**7..Critical damping vs Freq**	
Data Entry:		● **Single Value**	
X	**0**	**Y**	**0.03**
More			
X	**1000**	**Y**	**0.03**
More			
OK			
Cancel			

6. Create the model loading.

Before creating the appropriate loading, a load set needs to be created. Do so by performing the following.

Model/Load/Set...

Title: **frequency_dependent_load**

OK

Now, define the dynamic analysis parameters.

Model/Load/Dynamic Analysis...

Solution Method: ● **Modal Frequency**

Under *Equivalent Viscous Damping Modes*, select the following:.:

Modal damping: **2.. Modal_damping**

Modal Freq...

Frequency Band Spread (+/-): **3**

OK

Advanced...

Mass Formulation: ● **Coupled**

OK

OK

Modify the Solution Frequency Table.

Modify/Edit/Function...

Entitiy ID: **3.. Modal Frequency Table**

OK

Data Entry: ● **Linear Ramp**

Delta X: **20**

X	20	**Y**	1
To X	1000	**Y**	1

MORE

OK

7. Now, define the 1 psi time-varying pressure.

Model/Load/Elemental...

Select All

OK

(highlight)	**Pressure**
Method:	● **Constant**

Under *Load*, input the following. To select the Function Dependence, click on the list icon next to the databox and select **frequency_varying_load.**

Pressure/Value:	**0.1**
Pressure/ Function Dependence:	**1..frequency_varying_load**

OK

Face:	**1**

OK

Cancel

8. Next, define the unit load.

Model/Load/Nodal...

Select **Node 11.**

OK

To select the function dependence, click on the list icon next to the databox and select **frequency_varying_load**.

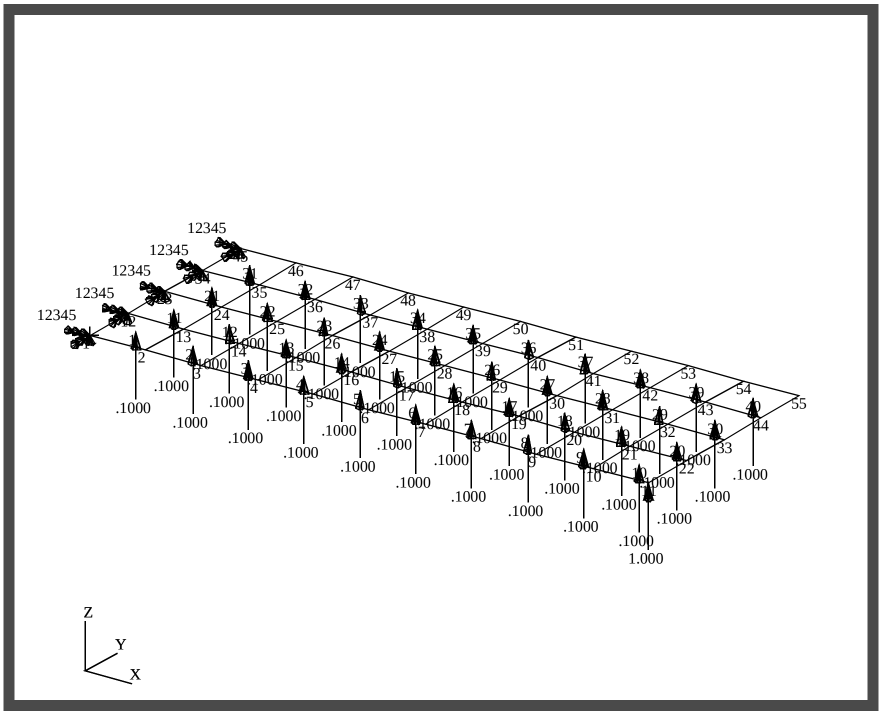
Function Dependence:	**1..frequency_varying_load**
FZ ☒	**1**

Phase: **-45**

OK

Cancel

Figure 6.2 - The model should appear similar to the following.



9. Delete the modes Data..

Delete/Output/Set...

Select All

OK

YES

10. Finally, create the input file for analysis.

File/Export/Analysis Modal...

Analysis Type: **4..Frequency/Harmonic Response**

OK

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

File name: **prob6**

Write

☒ **Run Analysis**

Advanced...

Solution Type: ● **Modal**

Under *Range of Interest* enter the following:

From (Hz): **0**

To (Hz): **2000**

OK

Problem ID: **Modal Frequency Response**

OK

Under *Output Requests*, unselect all except:

☒ **Displacement**

OK

Under *PARAM*, enter the following:

☒ *WTMASS* **.00259**

OK

11. When asked if you wish to save the model, 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 analysis ran smoothly, we will not bother with the details this time.

Continue

12. List the results of the analysis.

To list the displacement results at Node 11, select the following:

List/Output/Query...

Output Set:	**12..Case 12 Freq. 140**
Category:	**1..Displacement**
Entity:	● **Node**
ID:	**11**

OK

Repeat this process for all relevant node locations and frequencies. Answer the following questions using the results. The answers are listed at the end of the exercise.

Displacement at Node 11

Frequency		Displacement (T3)
140	=	__________
440	=	__________

Displacement at Node 33

Frequency		Displacement (T3)
140	=	__________
600	=	__________

Displacement at Node 55

Frequency		Displacement (T3)
140	=	__________

1000 = __________

13. Finally, create the XY plot of the deformed data.

Create the XY plot.

View/Select...

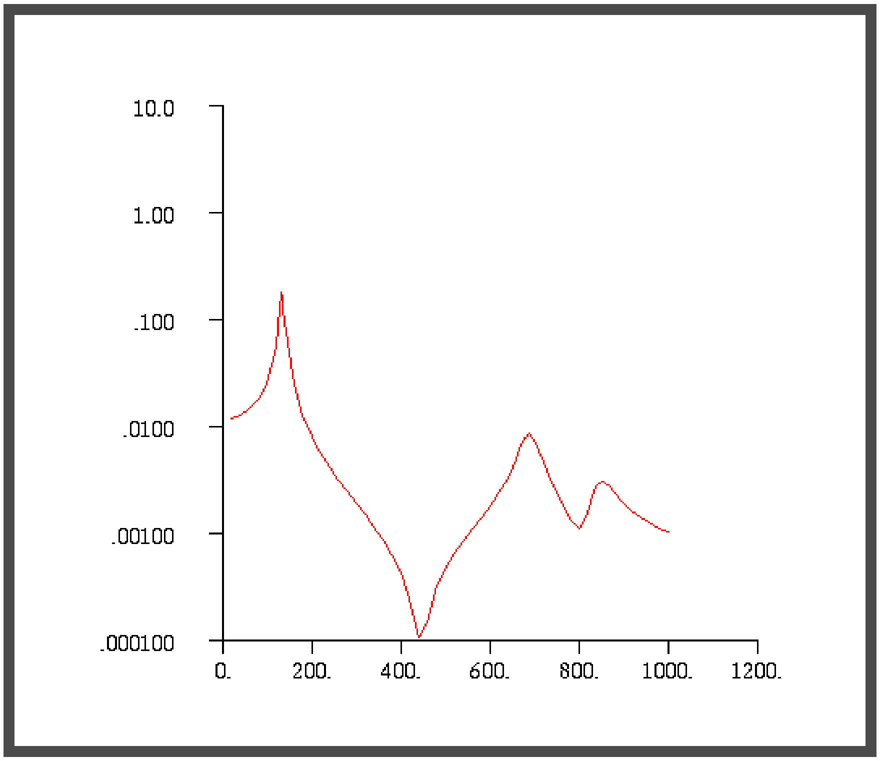
XY Style:	● **XY vs. Set Value**
XY Data...	
Category:	**0..Any Output**
Type:	**0..Value or Magnitude**
Output Set:	**1..Case 1 Freq 20**
Output Vector:	**4..T3 Translation**
Output Location/ Node:	**11**
OK	
OK	

To view the plots in semi-log scale, do the following steps.

View/Options...

Category:	● **PostProcessing**
Options:	**XY Axes Style**
Plot Type:	**1..Semi-Log (Y-Axis)**
OK	

Figure 6.3 - Displacement Response at Loaded Corner (Node 11)



To unpost the XY plot.

View/Select...

Model Style: ● **Draw Model**

OK

Now repeat this process to generate the XY plots of T3 displacement at Node 33 and 55.

Figure 6.4 - Displacement Response at Tip Center (Node 33)

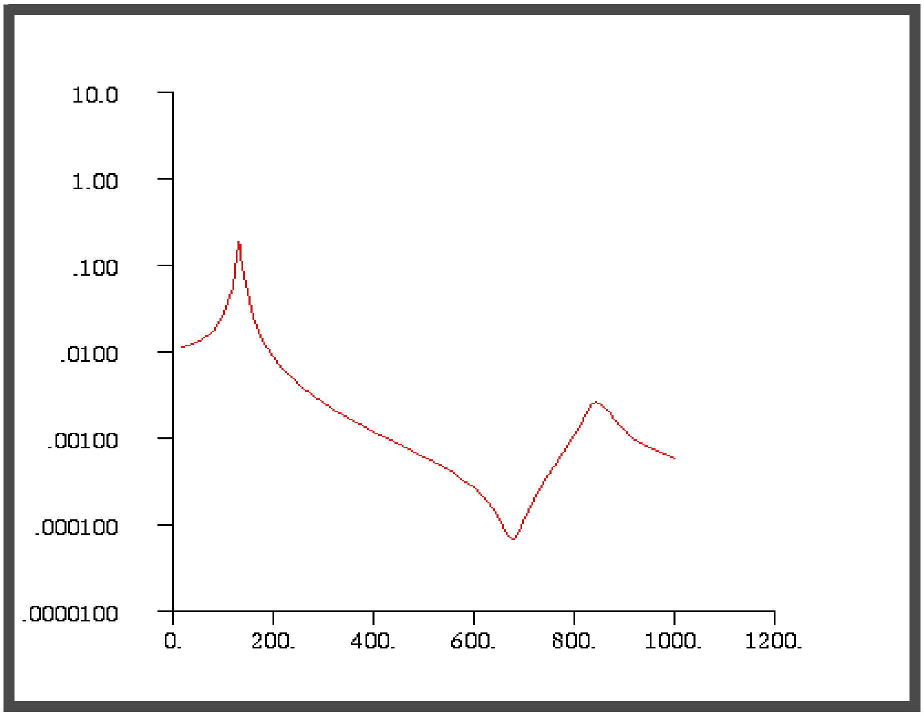
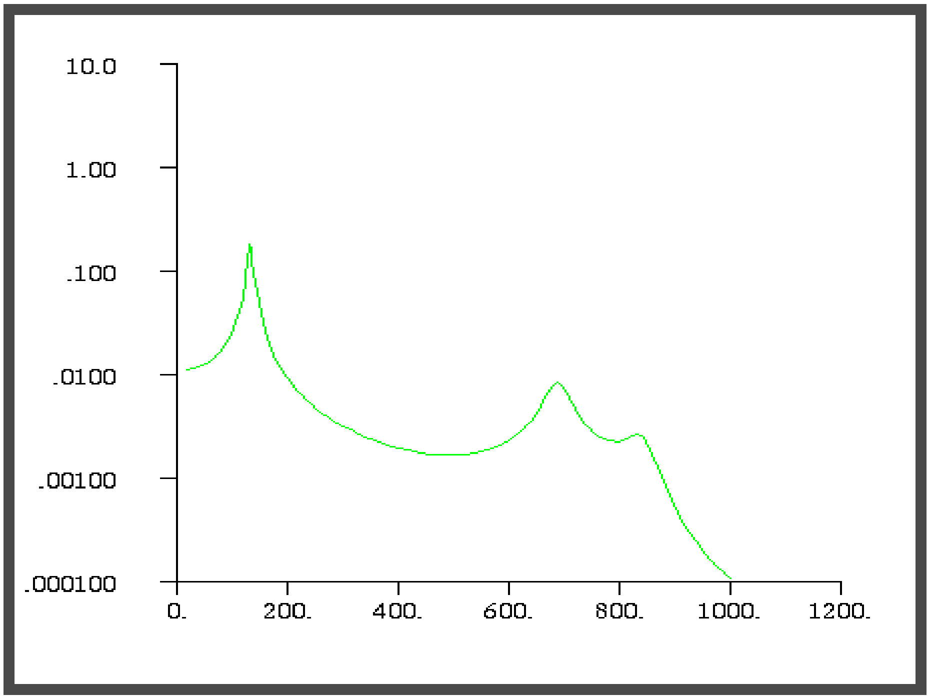


Figure 6.5 - Displacement Response at Opposite Corner (Node 55)



When finished, exit MSC.Nastran for Windows.

File/Exit

This concludes this exercise.

Nodal Displacement at Node 11

Frequency	T3
140	**0.0880**
440	**0.000348**

Nodal Displacement at Node 33

Frequency	T3
140	**0.0885**
600	**0.000285**

Nodal Displacement at Node 55

Frequency	T3
140	**0.0887**
1000	**0.000128**