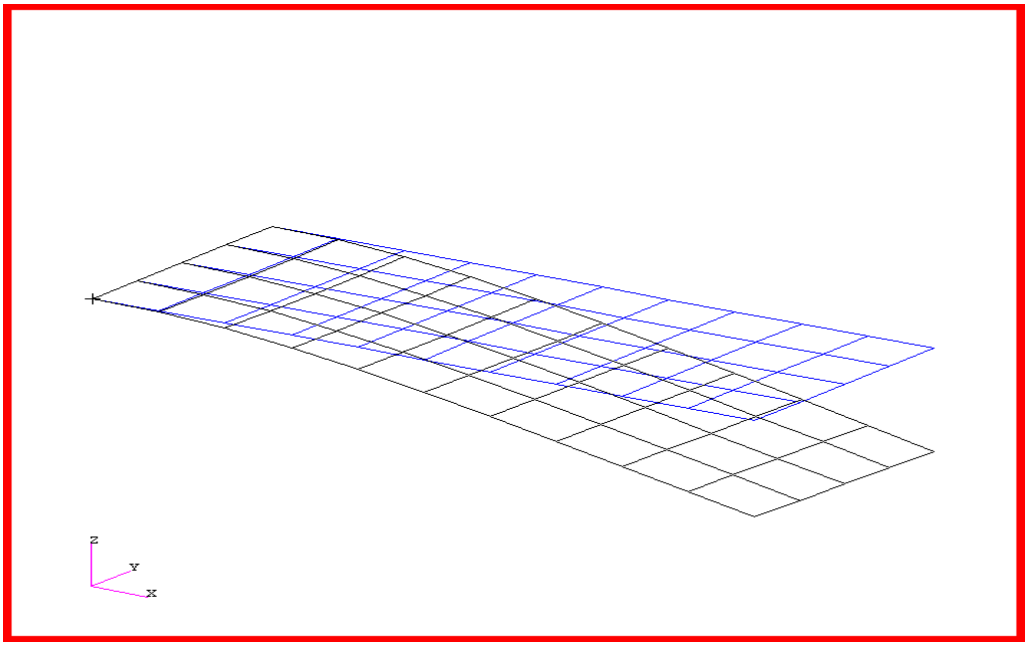
WORKSHOP PROBLEM 5

Direct 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 frequency-varying excitation.
- Run an MSC/NASTRAN modal frequency response analysis.
- Visualize analysis results.

Model Description:

Using the direct method, determine the frequency response of a 5x2 flat rectangular plate under frequency-varying excitation. This example structure shall be excited by a unit load at a corner of the tip. Use a frequency step of 20Hz between the range of 20 and 100Hz. Use structural damping of g=0.06.

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

Figure 5.1 - Grid Coordinates and Element Connectivities.

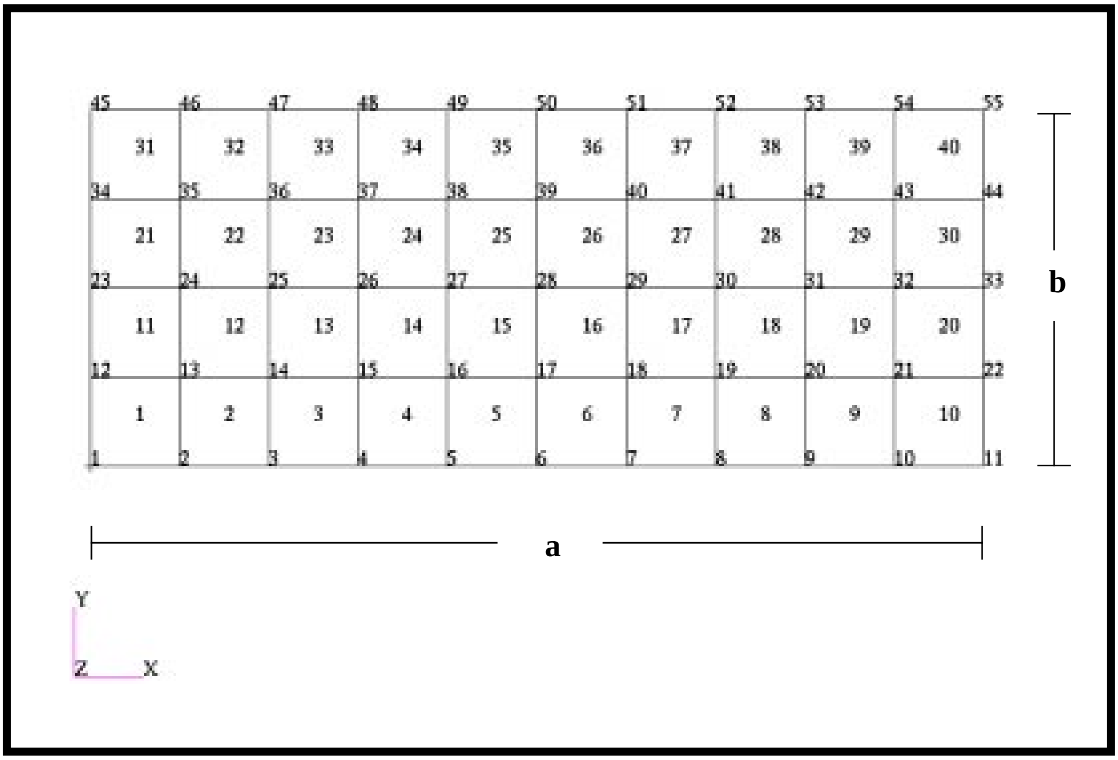


Figure 5.2 - Loads and Boundary Conditions

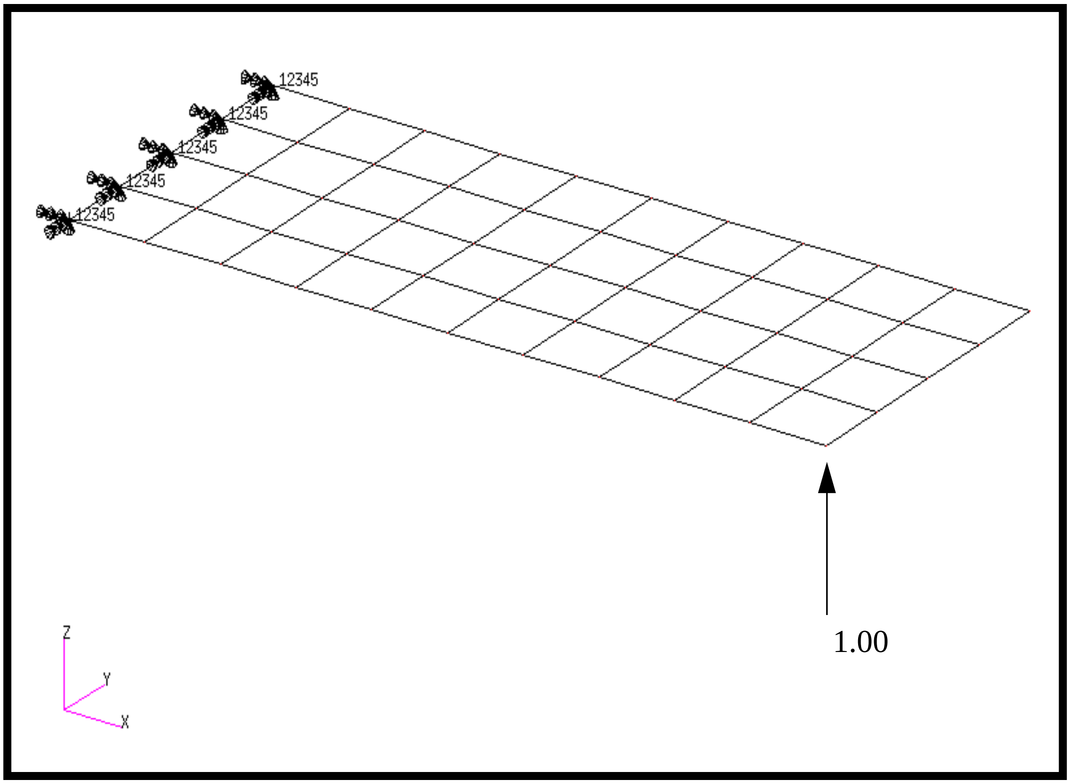


Table 5.1 - Properties

Length (a)	**5 in**
Height (b)	**2 in**
Thickness	**0.100 in**
Weight Density	**0.282 lbs/in^3**
Mass/Weight Factor	**2.59E-3 sec^2/in**
Young's Modulus	**30.0E6 lbs/in^2**
Poisson's Ratio	**0.3**

Exercise Procedure:

1. Start up MSC/NASTRAN for Windows 3.0 and begin to create a new model.

Double click on the icon labeled MSC/NASTRAN for Windows V3.0.

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. Create the frequency dependent function for the transient 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 a second function for frequency solution.

Title: **output_frequency**

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

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

Data Entry: ● **Linear Ramp**

Delta X: **20**

X **20** **Y** **1**

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

More

OK

Cancel

4. 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	● **Direct Frequency**

Under *Equivalent Viscous Damping*, input the following:

Overall Structural Damping Coeff (G):	**0.06**

Under *Frequency Response*, select the Solution Frequencies. To do this, click on the list icon next to the databox and select **output_frequency**.

Frequencies:	**2..output_frequency**
Advanced...	
Mass Formulation:	● **Coupled**
OK	
OK	

5. Now, define the unit load.

Model/Load/Nodal...

Select **Node 11.**

OK	
Type:	**Force**

Direction: ● **Component**

Method: ● **Constant**

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**

OK

Cancel

6. Finally, create the input file for analysis.

File/Export/Analysis Model...

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

OK

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

File name: **freq**

Write

☒ **Run Analysis**

Advanced...

Solution Type: ● **Direct**

OK

Problem ID: **Direct frequency response**

OK

Under *Output Requests*, unselect all except:

☒ **Displacement**

OK

Under *PARAM*, enter the following:

☒ *WTMASS* **.00259**

OK

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

Yes

File name: **freq**

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

8. List the results of the analysis.

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

List/Output/Query...

Output Set: **7..Case 7 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	=	___________
380	=	___________

Displacement at Node 33

Frequency		Displacement (T3)
140	=	___________
600	=	___________

Displacement at Node 55

Frequency		Displacement (T3)
140	=	___________
1000	=	___________

9. Finally, create the XY plot of the deformed data. First you may want to remove the labels and load and boundary constraint markers.

View/Options...

Quick Options...

Labels Off

Deselect the following:

☐ **Load - Force**

☐ **Constraint**

Done

OK

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	

Fig 5-3: Displacement Response at Node 11

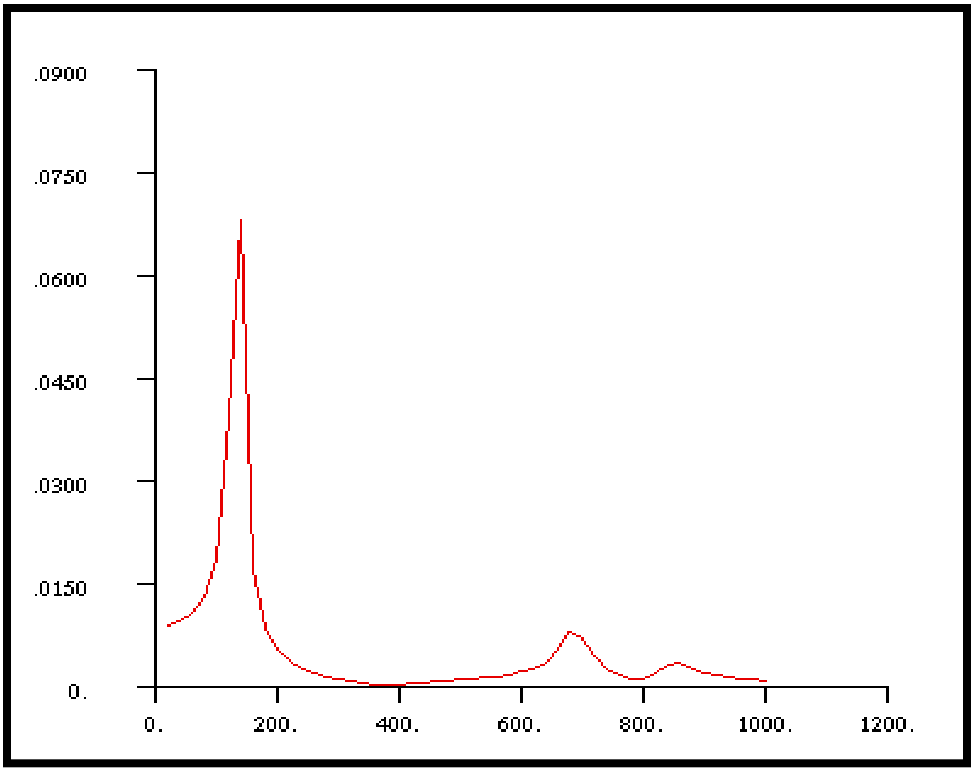


Fig 5-4: Displacement Response at Node 33

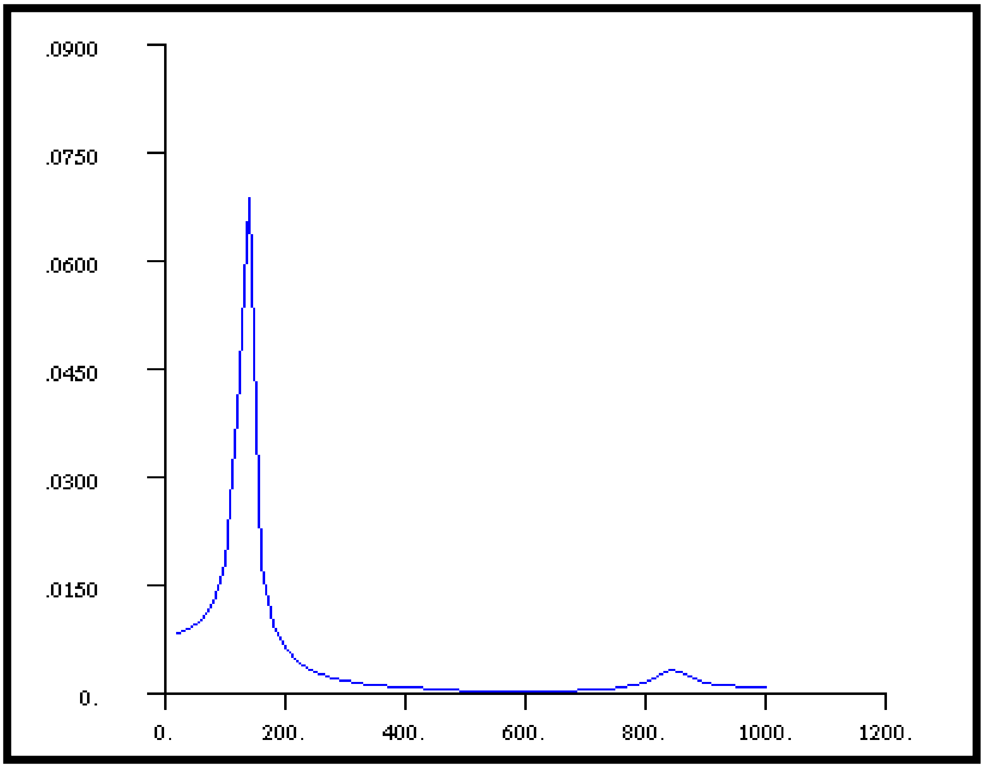
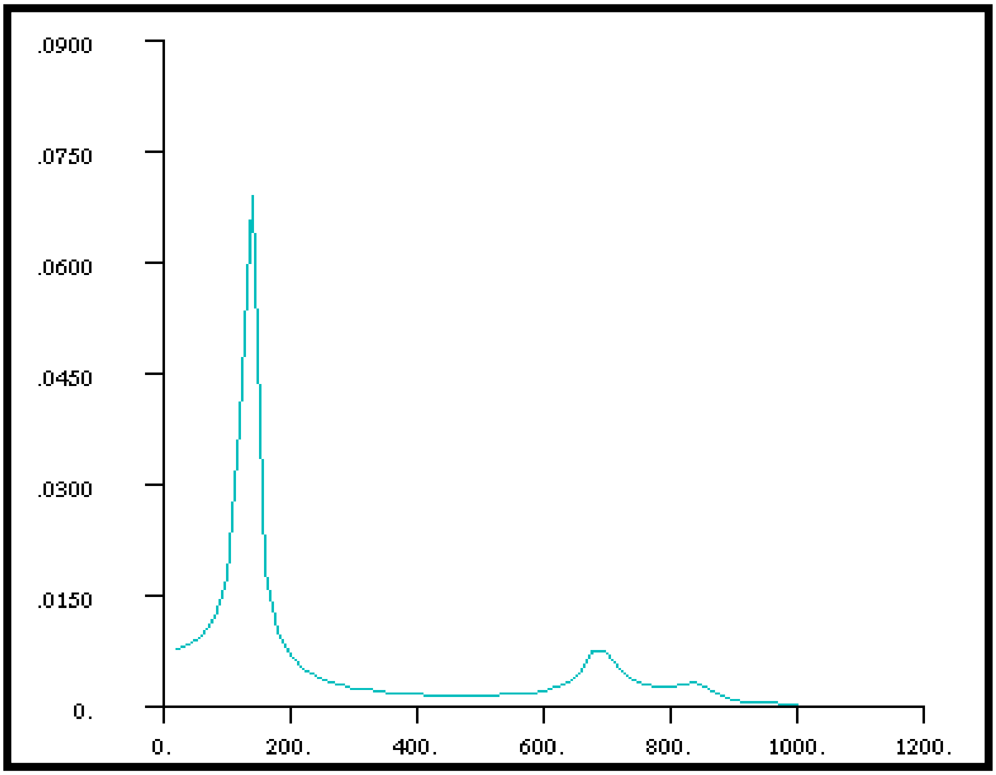


Fig 5-6: Displacement Response at Node 55



To remove the result 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.

When finished, exit MSC/NASTRAN for Windows.

File/Exit

This concludes this exercise.

Nodal Displacement at Node 11

Frequency	**T3**
140	**0.067946**
380	**0.00010012**

Nodal Displacement at Node 33

Frequency	**T3**
140	**0.068582**
600	**0.000049816**

Nodal Displacement at Node 55

Frequency	**T3**
140	**0.068966**
1000	**0.00021447**