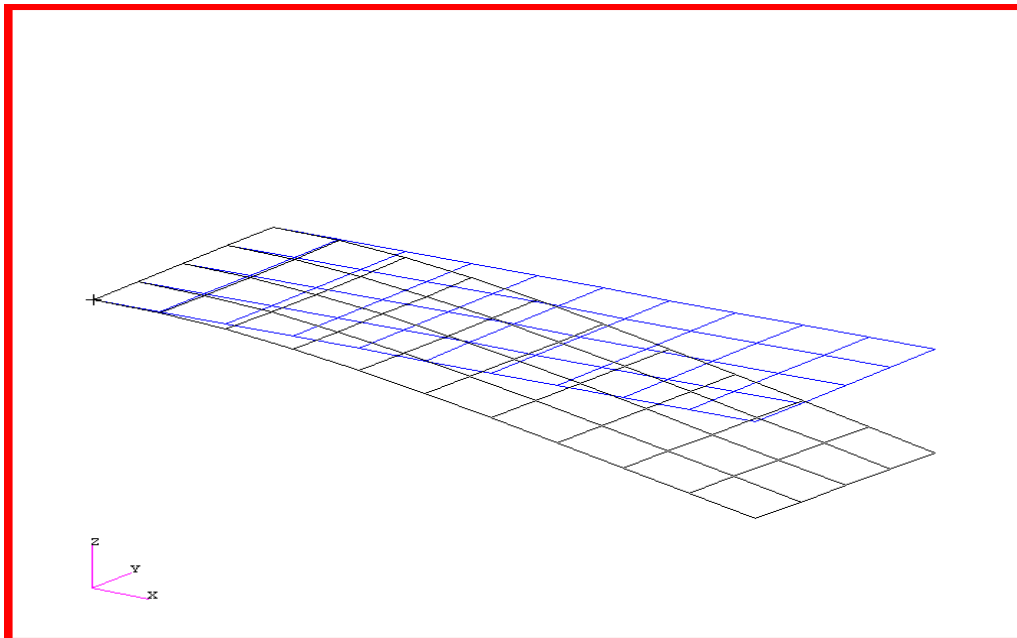

WORKSHOP PROBLEM 5

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 for Windows 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 1000Hz. 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.

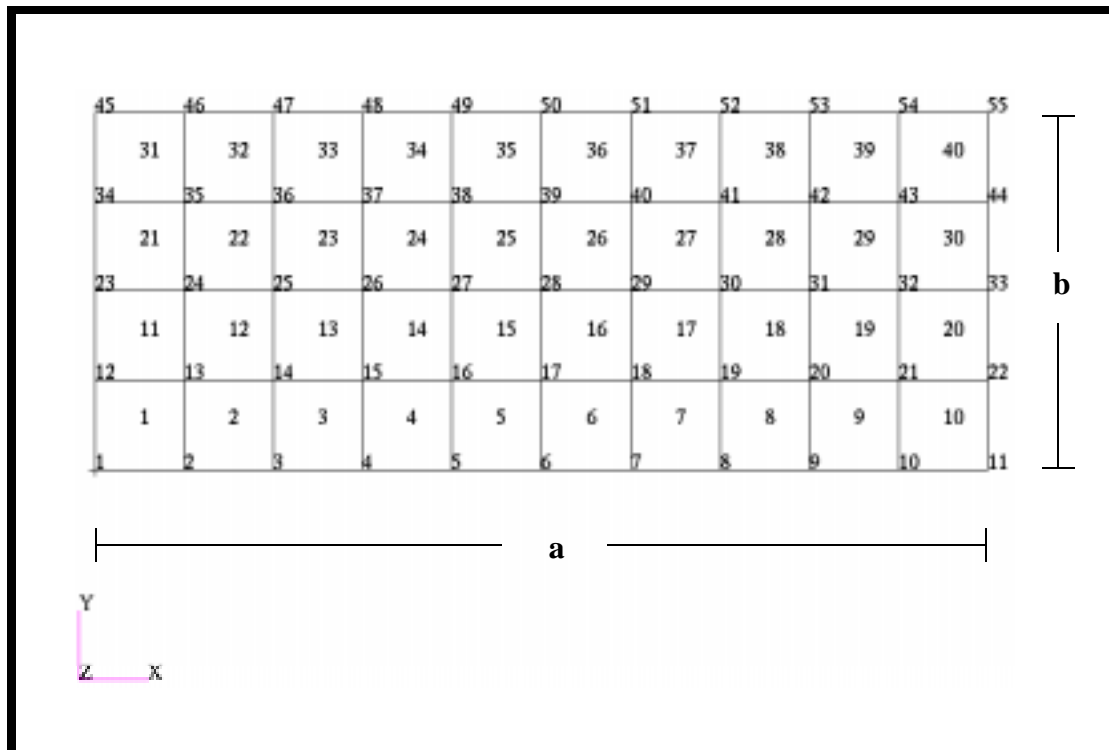


Figure 5.2 - Loads and Boundary Conditions

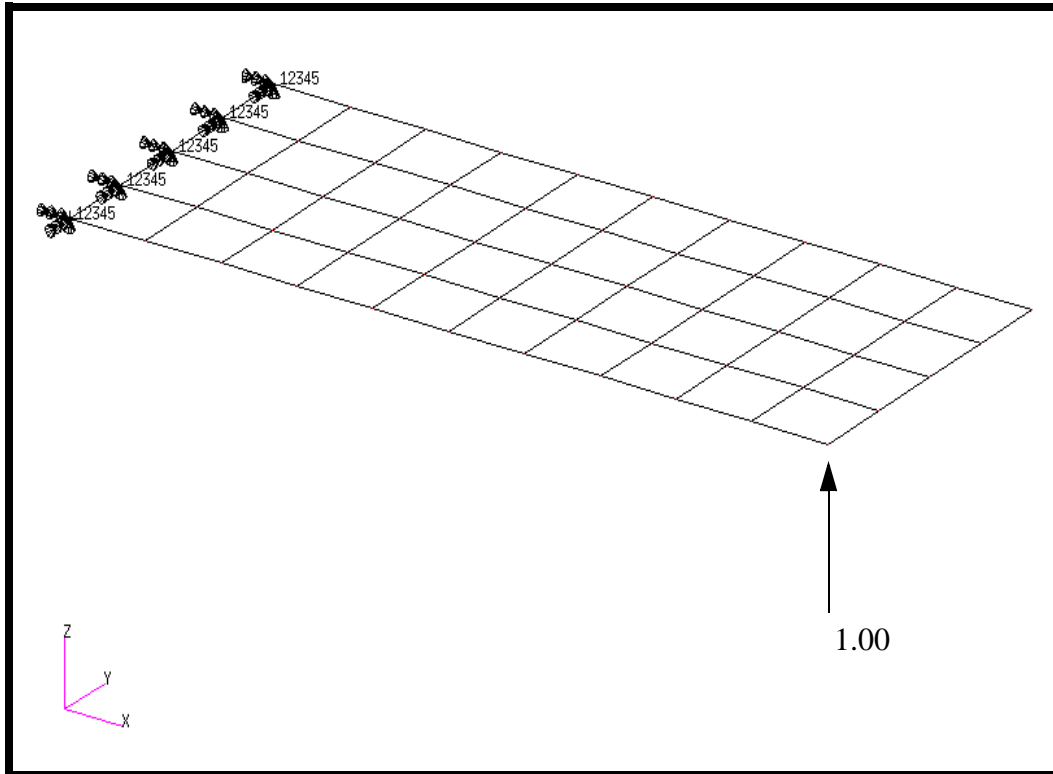


Table 5.1 - Properties

Length (a)	5 in
Height (b)	2 in
Thickness	0.100 in
Weight Density	0.282 lbs/in³
Mass/Weight Factor	2.59E-3 sec²/in
Young's Modulus	30.0E6 lbs/in²
Poisson's Ratio	0.3

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:

2. Import **prob1.DAT**.

File/Import/Analysis Model...

Nastran

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

File name:

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

To reset the display of the model do the following:

View/Redraw

View/Autoscale

View/Rotate...

-
3. Create the frequency dependent function for the frequency response of the unit load.

Model/Function...

Title:

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

Type:

Data Entry: **Single Value**

X

Y

X

Y

Create a second function for frequency solution.

Title:

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

Type:

Data Entry: **Linear Ramp**

Delta X:

X

Y

To X

To Y

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:

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):*

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

Frequencies:

Mass Formulation:

Coupled

5. Now, define the unit load.

Model/Load/Nodal...

Select **Node 11**.

Type:

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:

FZ

OK

Cancel

6. Finally, create the input file for analysis.

File/Export/Analysis Model...

Type:

OK

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

File name:

Write

Run Analysis

Advanced...

Solution Type: **Direct**

OK

Problem ID:

OK

Under *Output Requests*, unselect all except:

Displacement

OK

Under *PARAM*, enter the following:

WTMASS

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

File name:

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.

8. List the results of the analysis.

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

List/Output/Query...

Output Set:

Category:

Entity

Node

ID:

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.

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

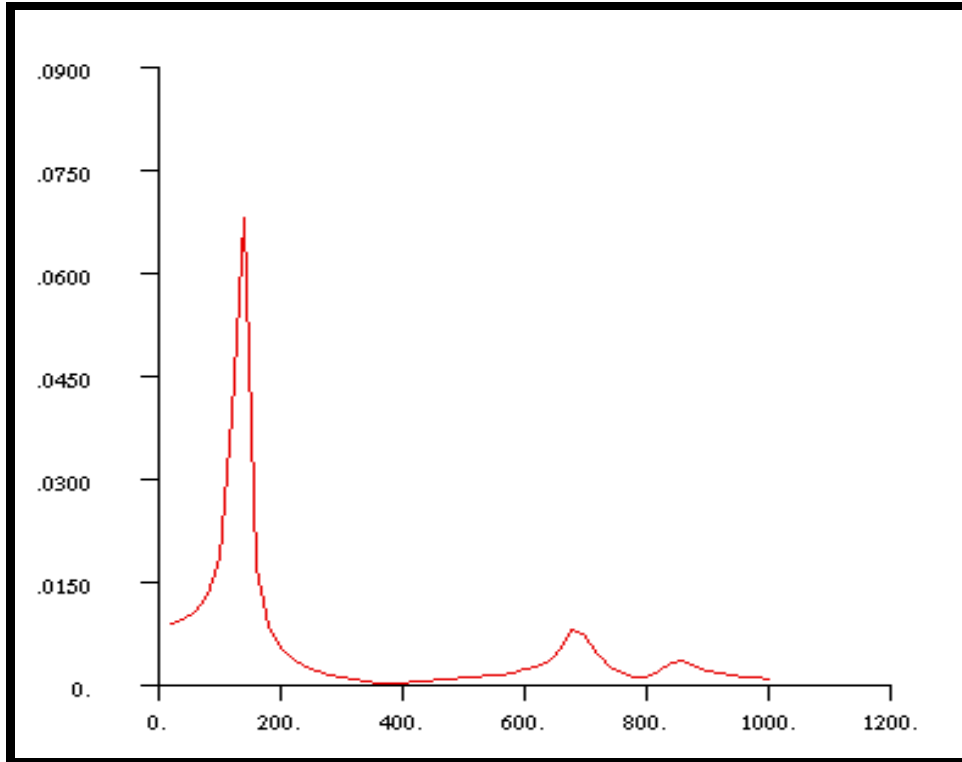


Fig 5-4: Displacement Response at Node 33

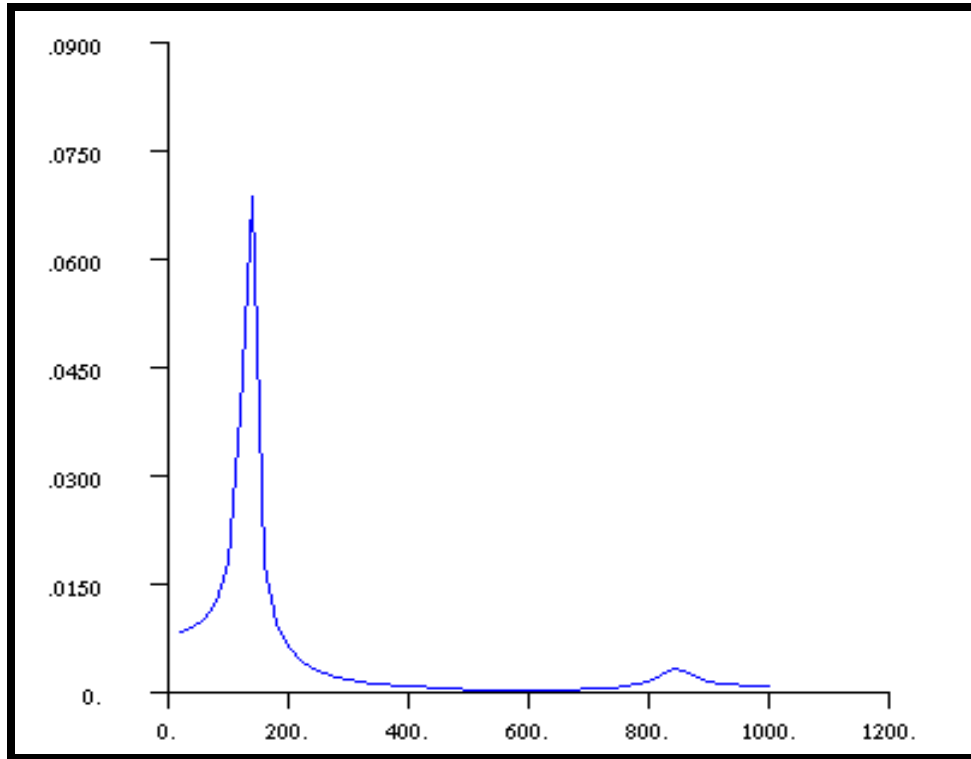
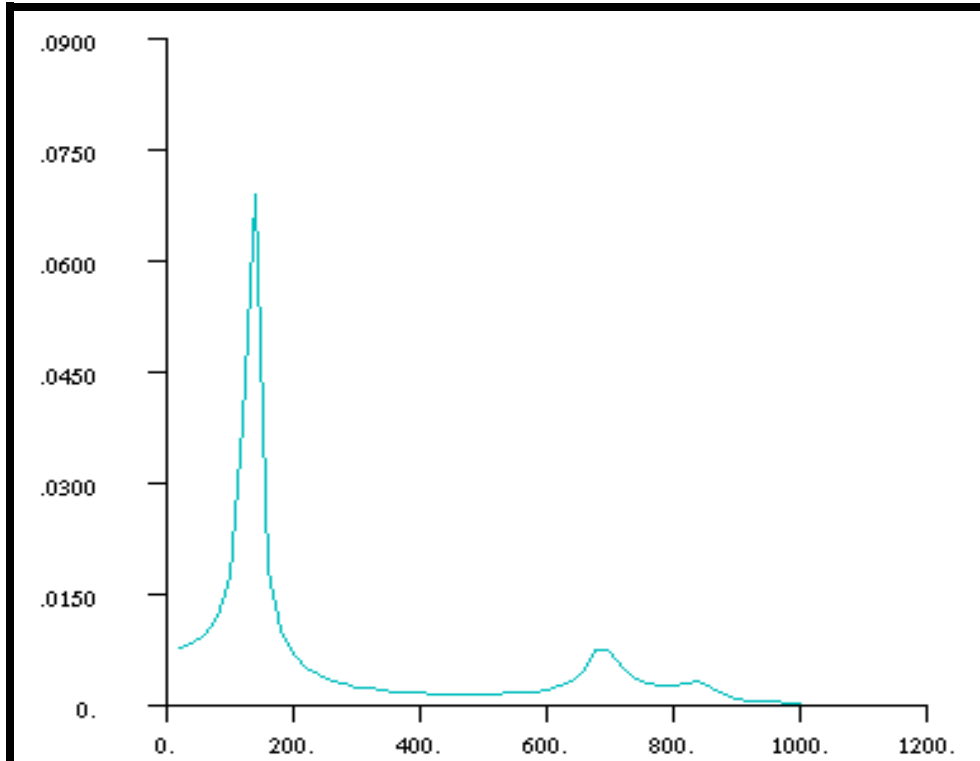


Fig 5-6: Displacement Response at Node 55



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

To remove the result plot:

View/Select...

Model Style:

Draw Model

OK

When finished, exit MSC.Nastran for Windows.

File/Exit

This concludes this exercise.

Nodal Displacement at Node 11

<i>Frequency</i>	T3
140	0.067946
380	0.00010012

Nodal Displacement at Node 33

<i>Frequency</i>	T3
140	0.068582
600	0.000049816

Nodal Displacement at Node 55

<i>Frequency</i>	T3
140	0.068966
1000	0.00021447
