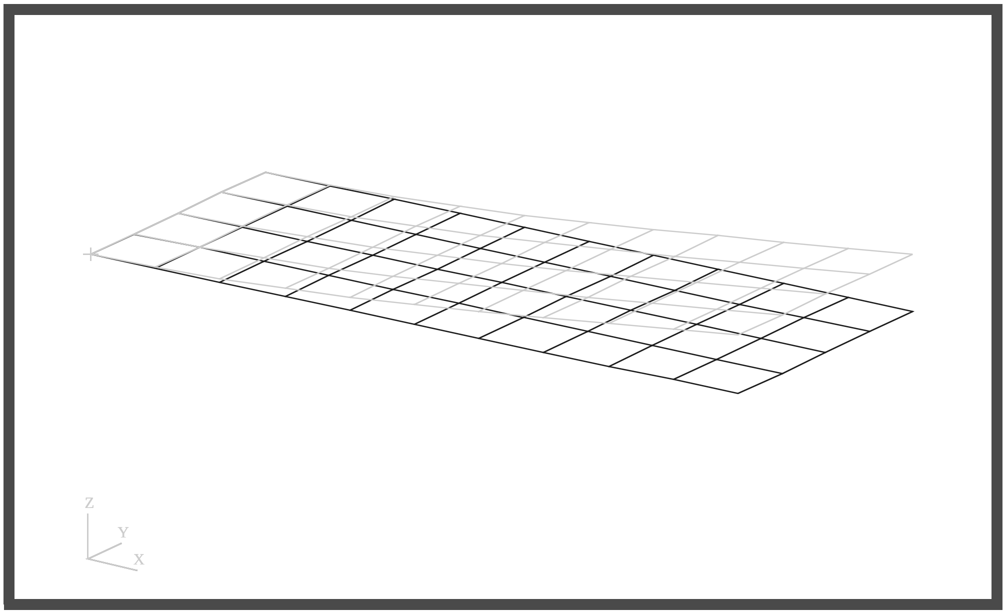
WORKSHOP PROBLEM 8

Enforced Motion with Direct Frequency Response



Objectives

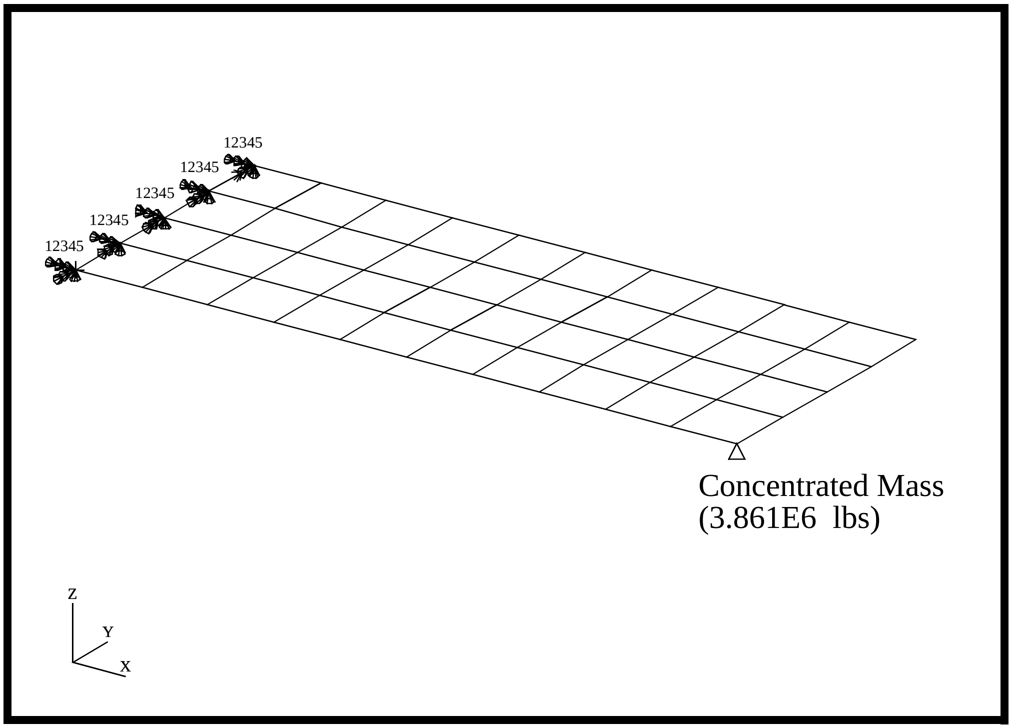
- Create a geometric representation of a flat rectangular plate.
- Use the geometry model to define an analysis model comprised of plate elements.
- Define frequency-varying tip displacement.
- Use the large mass method.
- Submit the file for analysis in MSC.Nastran for Windows.
- Compute nodal displacements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

Model Description:

Using the direct method, determine the frequency response of the flat rectangular plate, created in Workshop 1, under a 0.1 displacement at a corner of the tip. Use a frequency step of 20 Hz in the range of 20 to 1000 Hz. Use a structural damping of g = 0.06.

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

Figure 8.1 - 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. Create the frequency dependent function for the frequency response of the unit load and the list of output frequencies.

To define 0.1 inches of displacement for frequencies from 20 to 1000 the force applied to the large mass must be a function of frequency. The function is:

$$\text{acceleration} = -(2\pi f)^2 \text{ displacement}$$

Model/Function...

Title:	**SQR (2_PI_f)**
Type:	**3..vs. Frequency**
Data Entry:	● **Equation**
Delta X:	**20**
X **0**	**Y** **-1*SQR(6.28*!x)**
To X **1000**	

MORE

OK

ID	**2**

Model/Function...

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_response**

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: **1..output_frequency**

Advanced...

Mass Formulation: ● **Coupled**

OK

OK

Model/Load/Nodal...

ID **11**

OK

(highlight) ● **Force**

Method: ● **Constant**

Function Dependence: **1..SQR(2_PI_f)**

FZ ☒ **1000**

OK

Cancel

5. Now, create the concentrated mass.

First, define the property of the concentrated mass.

Model/Property...

Title: **Base_Mass**

Elem/Property Type...

● **Mass**

OK

Mass, M or Mx: **3.861E6**

OK

Cancel

Next create a element that will represent the concentrated mass.

Model/Element...

Property: **Base_Mass**

Node: **11**

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: **prob8**

Write	
	☒ **Run Analysis**
Advanced...	
Solution Type:	● **Direct**
OK	
Problem ID:	**Enforced Motion with 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:	**prob8**
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:	**1..Case 1 Freq 20**
Category:	**1..Displacement**
Entity:	● **Node**
ID:	**11**

OK

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

Displacement at Node 11

Frequency (X)		Displacement (Y)
20	=	__________
380	=	__________

Displacement at Node 33

Frequency (X)		Displacement (Y)
380	=	__________
600	=	__________

Displacement at Node 55

Frequency (X)		Displacement (Y)
380	=	__________
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.

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 20.00**

Output Vector: **4..T3 Translation**

Output Location/ Node: **11**

OK

OK

Repeat the process for Nodes 33 and 55.

Figure 8.2 - Displacement Response at Node 11

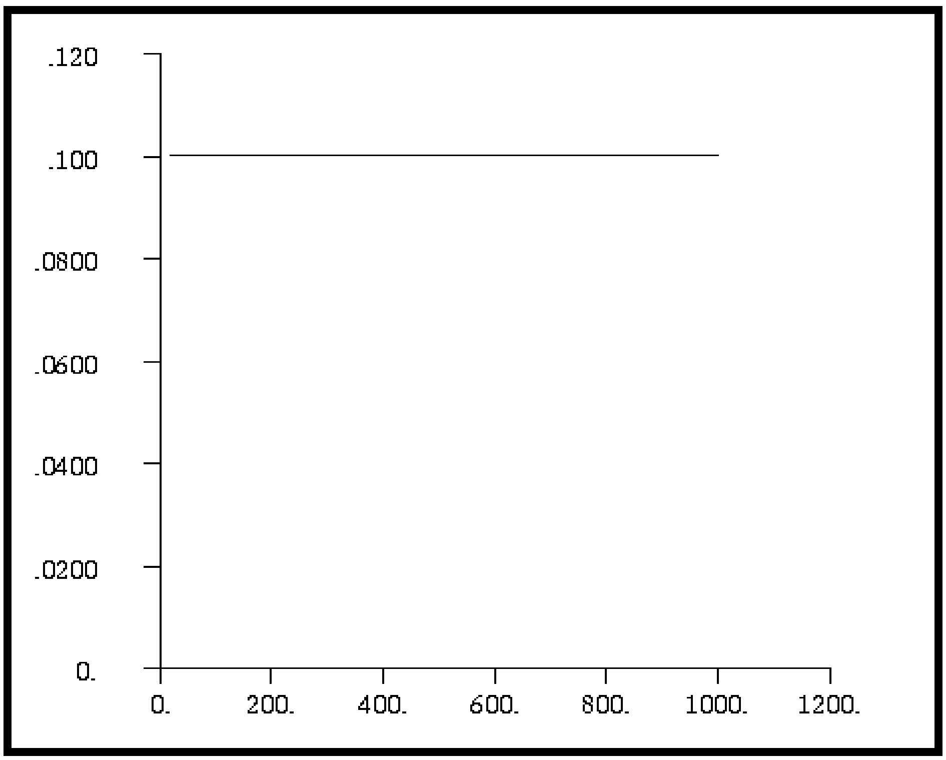


Figure 8.3 - Displacement Response at Node 33

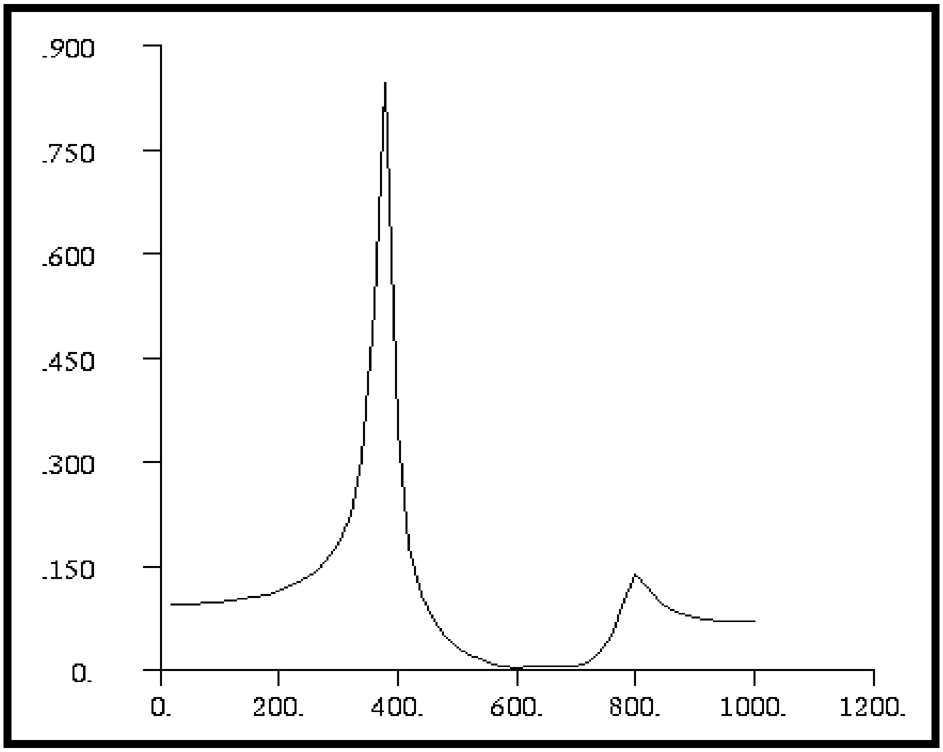
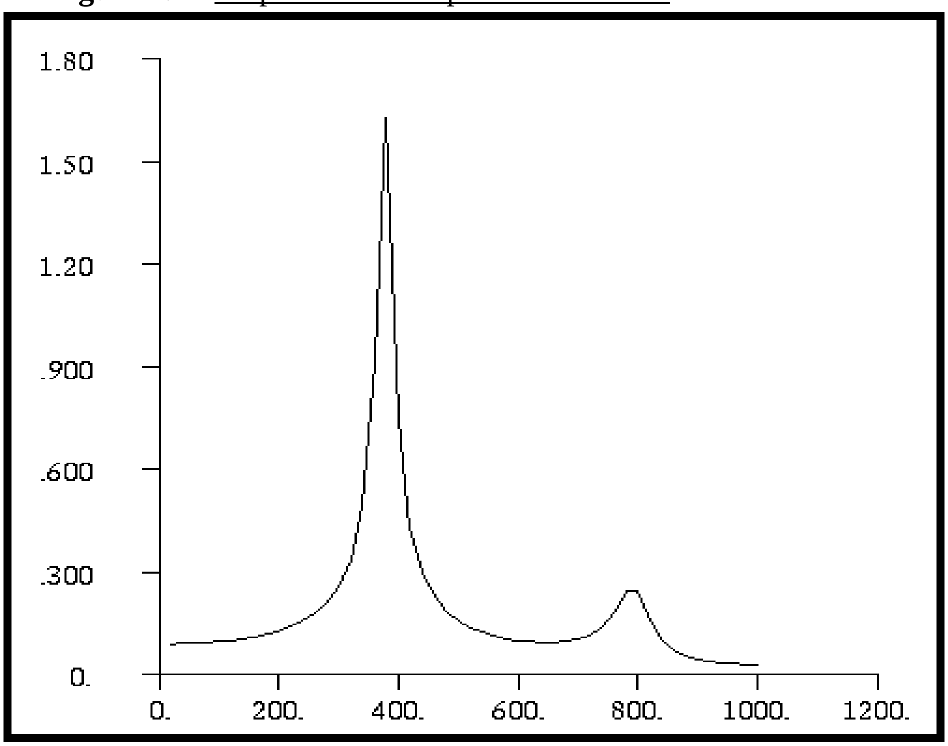


Figure 8.4 - Displacement Response at Node 55



When finished, exit MSC.Nastran for Windows.

File/Exit

This concludes this exercise.

Node 11

Freq	Displacement (T3)
20	**0.0998991**
380	**0.0999881**

Node 33

Freq	Displacement (T3)
380	**0.845199**
600	**0.00231275**

Node 55

Freq	Displacement (T3)
380	**1.625425**
1000	**0.024314**