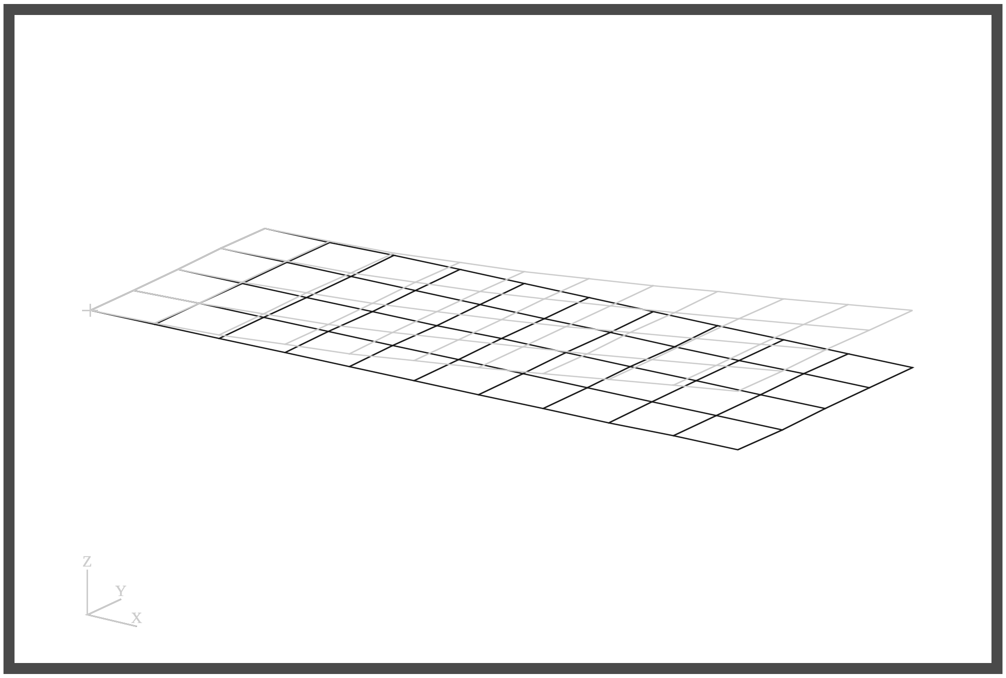
WORKSHOP PROBLEM 4

Modal Transient Response Analysis



Objectives

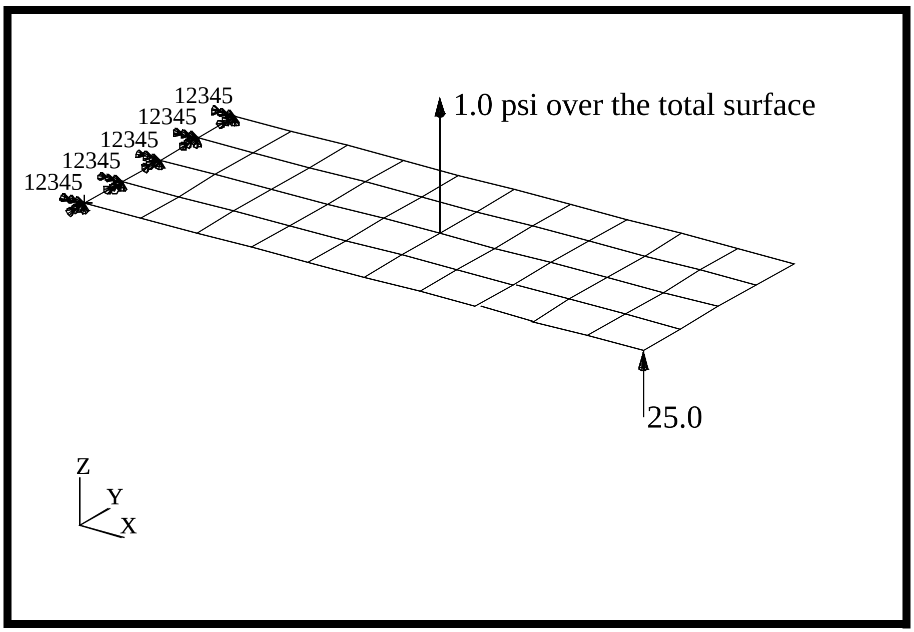
- Create a geometric representation of a flat rectangular plate.
- Use the geometry model to define an analysis model comprised of plate elements.
- Define time-varying excitation.
- Submit the file for analysis in MSC.Nastran for Windows.
- Compute nodal displacements for desired time domain.

Model Description:

Using the modal method, determine the transient response of the flat rectangular plate, created in Workshop 1, under time-varying excitation. This example structure shall be excited by a 1 psi pressure load over the total surface of the plate varying at 250Hz. In addition, a 25 lb force is applied at a corner of the tip also varying at 250Hz but starting 0.004 seconds after the pressure load begins. Both time-dependent dynamics loads are applied only for the duration of 0.008 seconds only. Use a modal damping of ξ = 0.03 for all modes. Carry out the analysis for 0.04 seconds.

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

Figure 4.1 - 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 thImport **prob1.DAT.**

View/Redraw

View/Autoscale

View/Rotate... **Dimetric**

OK

3. Create the time dependent function for the transient response of the pressure loading.

Model/Function...

Title: **time_varying_pressure**

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

Type: **1..vs. Time**

Data Entry: ● **Equation**

Delta X: **0.0004**

X **0** **Y** **sin(360*250.*!x)**

To X **0.008**

More

Data Entry: ● **Single Value**

X **0.008** **Y** **0**

More

X **0.04** **Y** **0**

More

OK

Cancel

4. Create the time-dependent function for the transient response of the nodal loading with some initial delay.

Model/Function...

ID: **2**

Title: **time_varying_nodal_force**

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

Type: **1..vs. Time**

Data Entry: ● **Single Value**

X 0 **Y** 0

More

Data Entry: ● **Equation**

Delta X: 0.0004

X 0.004 **Y** sin(360*250.*!x)

To X 0.012

More

Data Entry: ● **Single Value**

X 0.04 **Y** 0

More

OK

Cancel

5. Create the frequency dependent critical damping.

Model/Function...

ID: 3

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: **transient_loading**

OK

Now, define the dynamic analysis parameters.

Model/Load/Dynamic Analysis...

Solution Method: ● **Modal Transient**

Under *Equivalent Viscous Damping*, select the following:

Modal damping table: **3..Modal_damping**

Under *Transient Time Step Intervals*, input the following:

Number of Steps: **100**

Time per Step: **4e-4**

Output Interval: **1**

Advanced...

Mass Formulation: ● **Coupled**

OK

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 **time_varying_pressure.**

Pressure/Value: **1**

Pressure/
Function Dependence: **1..time_varying_pressure**

OK

Face: **1**

OK

Cancel

8. Next, create the time varying nodal force under the same dynamic load set previously created.

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 **time_varying_nodal_force**.

Function Dependence: **2..time_varying_nodal_force**

FZ ☒ **25**

OK

Cancel

9. Create the input file for analysis.

File/Export/Analysis Model...

Type: **3..Transient Dynamic/Time History**

OK

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

File name: **prob4**

Write

☒ **Run Analysis**

Advanced...

Solution Type: ● **Modal**

Modal Solution Method: ● **Lanczos**

Eigenvalues and Eigenvectors/ Number Desired: **5**

OK

Problem ID: **Modal Transient Response**

OK

Under *Output Requests*, unselect all except:

☒ **Displacement**

OK

Under *PARAM*, enter the following:

☒ *WTMASS* **.00259**

OK

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

Yes

File name: **prob4**

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

11. List the results of the analysis.

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

List/Output/Query...

Output Set:	**16..Case 16 Time 0.006**
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.

Nodal Displacement at Node 11

Time		T3
0.006	=	___________
0.0092	=	___________

Nodal Displacement at Node 33

Time		T3
0.0068	=	___________
0.0092	=	___________

Nodal Displacement at Node 55

Time		T3
0.0068	=	___________
0.0052	=	___________

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

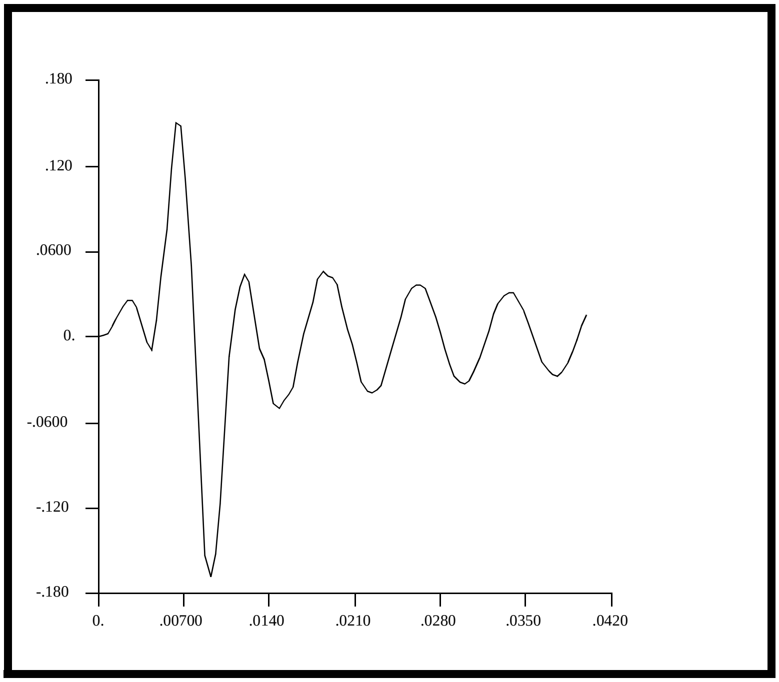
Create the XY plot.

View/Select...

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

The output should look similar to Figure 4.2.

Figure 4.2



Now repeat this process to generate the XY plots of T3 displacement at Node 33 and 55. The output of Node 33 and 55 should look similar to Figure 4.3 and 4.4.

Figure 4.3

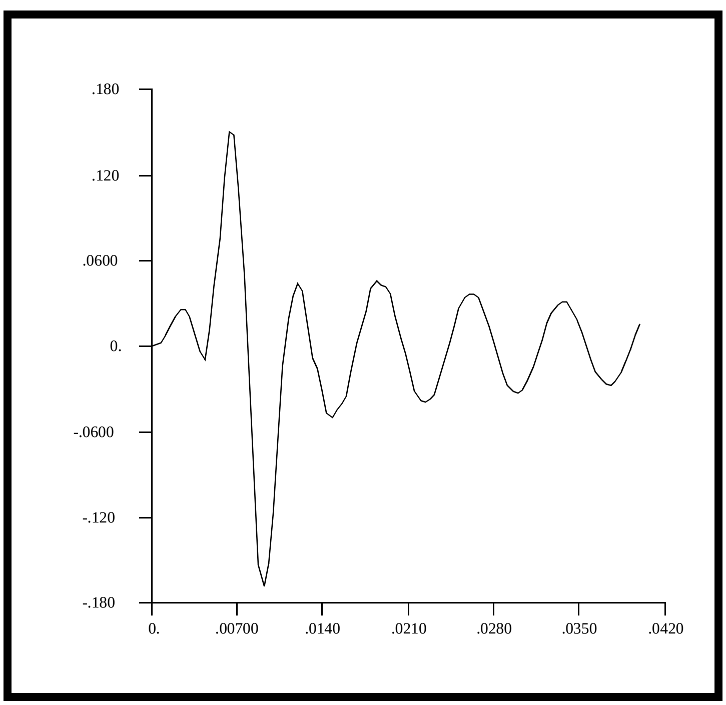
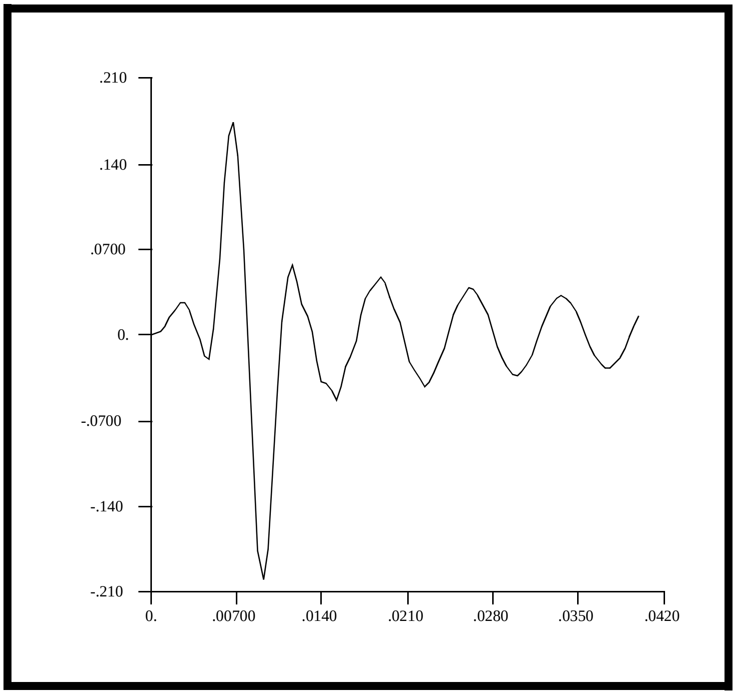


Figure 4.4



When finished, exit MSC.Nastran for Windows.

File/Exit

This concludes this exercise.

Nodal Displacement at Node 11

Time	**T3**
0.006	**0.118**
0.0092	**-0.169**

Nodal Displacement at Node 33

Time	**T3**
0.0068	**0.161**
0.0052	**0.022**

Nodal Displacement at Node 55

Time	**T3**
0.0068	**0.173**
0.0092	**-0.201**