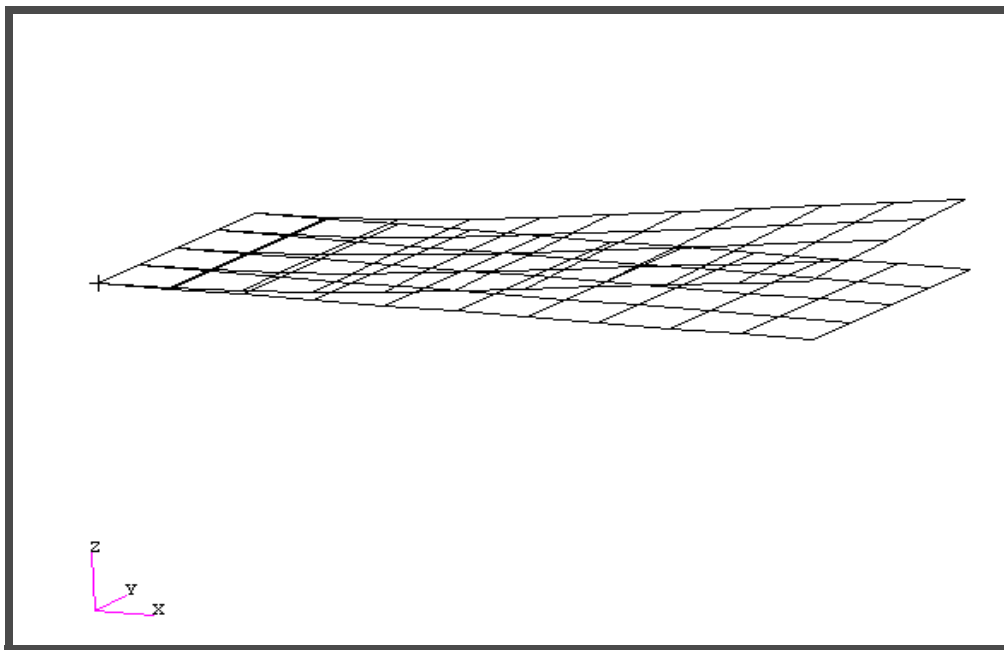

WORKSHOP PROBLEM 3

Direct 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 excitations.
- Run an MSC.Nastran for Windows direct transient response analysis.
- Visualize analysis results.

Model Description:

Use the direct method, determine the transient response of a 5x2 flat rectangular plate under time-varying excitation. This example structure shall be excited by 1 psi pressure load over the total surface of the plate varying at 250 Hz. In addition, a 50 lb force is applied at a corner of the tip also varying at 250 Hz but out-of-phase with the pressure load. Both time dependent dynamic loads are applied for the duration of 0.008 seconds only. Use structural damping of $g = 0.06$ and convert this damping to equivalent viscous damping at 250 Hz. Carry out the analysis for 0.04 seconds with a time step of 0.004 seconds.

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

Figure 3.1 - Grid Coordinates and Element Connectivity

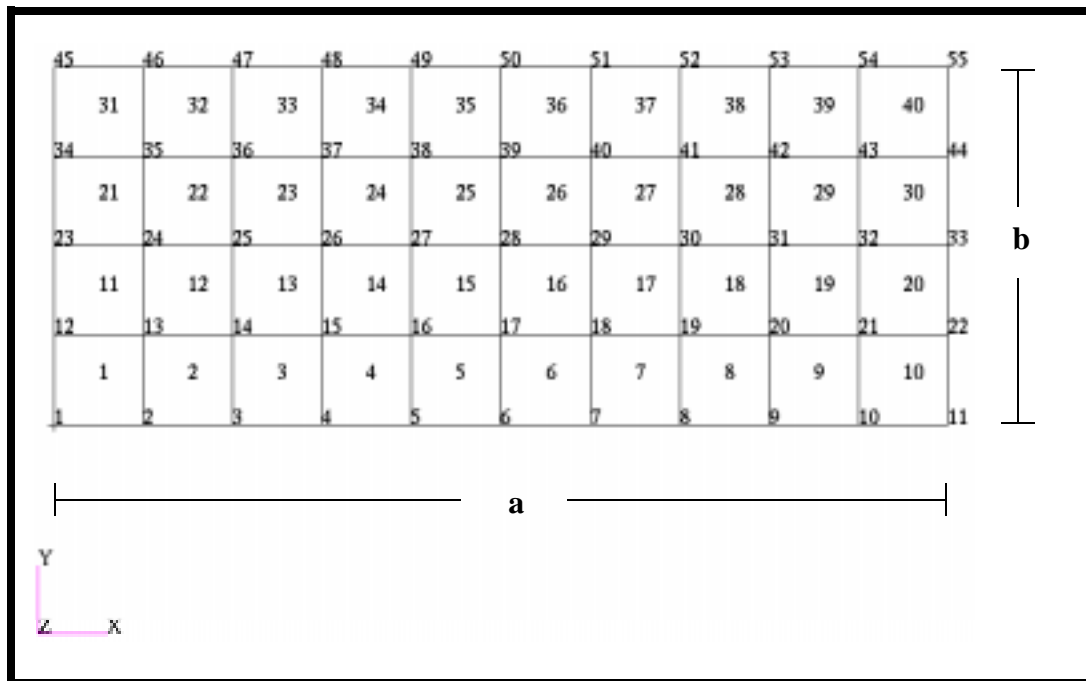


Figure 3.2 - Loads and Boundary Conditions

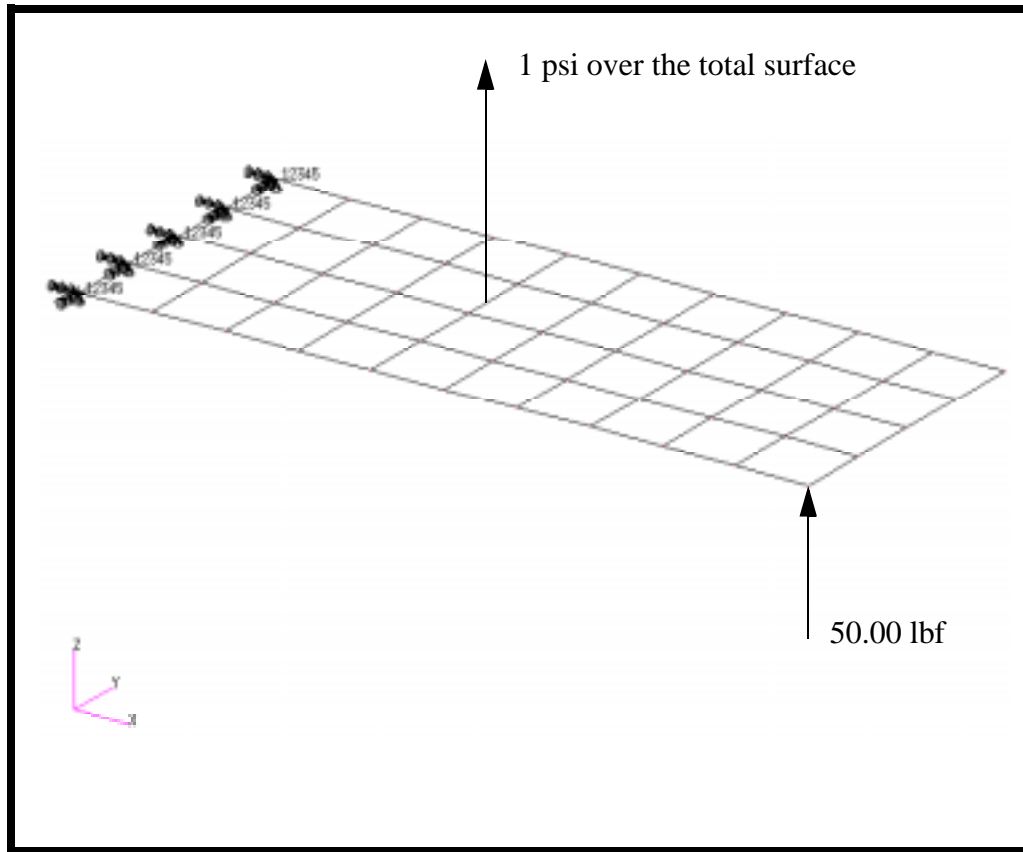


Table 3.1 - Properties

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
Youngs 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 time dependent function for the transient response of the pressure loading.

Model/Function...

Title:

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

Type:

Data Entry: Equation

Delta X:

X

Y

To X

Data Entry: Single Value

X

Y

X

Y

4. Create the time-dependent function for the transient response of the nodal loading.

Model/Function...

ID:

Title:

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

Type:

Data Entry: Equation

<i>Delta X:</i>	<input type="text" value="0.0004"/>
X	<input type="text" value="0"/>
To X	<input type="text" value="0.008"/>
<input type="button" value="More"/>	
<i>Data Entry:</i>	<input checked="" type="radio"/> Single Value
X	<input type="text" value="0.008"/>
<input type="button" value="More"/>	
X	<input type="text" value="0.04"/>
<input type="button" value="More"/>	
<input type="button" value="OK"/>	
<input type="button" value="Cancel"/>	

Y	<input type="text" value="-sin(360*250.*!x)"/>
Y	<input type="text" value="0"/>
Y	<input type="text" value="0"/>

5. Create the modal loading.

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

Model/Load/Set...

<i>Title:</i>	<input type="text" value="transient_loading"/>
<input type="button" value="OK"/>	

Now, define the dynamic analysis parameters.

Model/Load/Dynamic Analysis...

Solution Method: **Direct Transient**

Under *Equivalent Viscous Damping*, input the following:

<i>Overall Structural Damping Coeff (G):</i>	<input type="text" value="0.06"/>
--	-----------------------------------

Under *Equivalent Viscous Damping Conversion*, input the following:

*Frequency for System
Damping [W3-Hz]:*

250

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

Number of Steps:

100

Time per Step:

4e-4

Output Interval:

1

Advanced...

Mass Formulation:

Coupled

OK

OK

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

6. Now create the time varying nodal force under the same dynamic load set previously created.

Model/Load/Nodal...

Select **Node 11**.

OK

(highlight)

Force

Direction:

Components

Method:

Constant

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

FZ

50

Function Dependence:

2..time_varying_nodal_force

OK

Cancel

7. 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:

direct

Write

Run Analysis

Advanced...

Solution Type

Direct

OK

Problem ID:

Direct Transient Response

Under *Output Requests*, unselect all except:

Displacement

Under *PARAM*, enter the following:

WTMASS

8. 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.

9. 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 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
0.0024	=	_____
0.0052	=	_____
0.02	=	_____

Nodal Displacement at Node 33

Time		T3
0.0024	=	_____
0.0052	=	_____
0.02	=	_____

Nodal Displacement at Node 55

Time		T3
0.0024	=	_____
0.0052	=	_____
0.02	=	_____

- 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:

Time 0

Output Vector:

4..T3 Translation

Output Location/
Node:

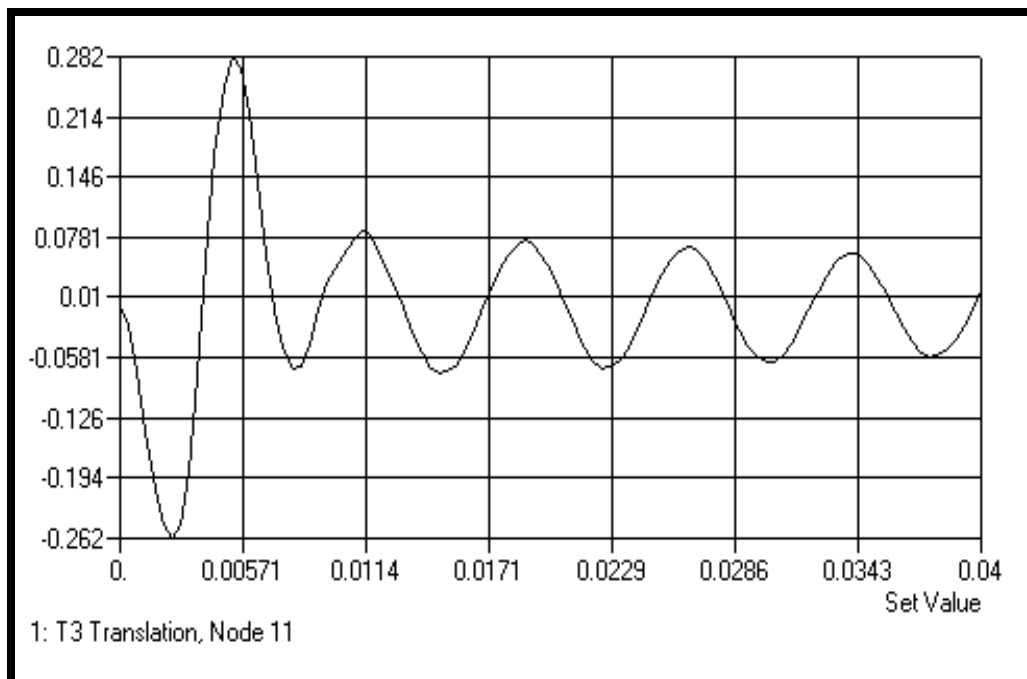
11

OK

OK

The plot should appear as follows:

Figure 3.3 - XY Plot of T3 Displacement at Node 11



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

To unpost the XY plot, do the following:

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>Time</i>	T3
0.0024	-0.26233
0.0052	0.28239
0.02	0.038671

Nodal Displacement at Node 33

<i>Time</i>	T3
0.0024	-0.28827
0.0052	0.32209
0.02	0.039833

Nodal Displacement at Node 55

<i>Time</i>	T3
0.0024	-0.3115
0.0052	0.35709
0.02	0.040889
