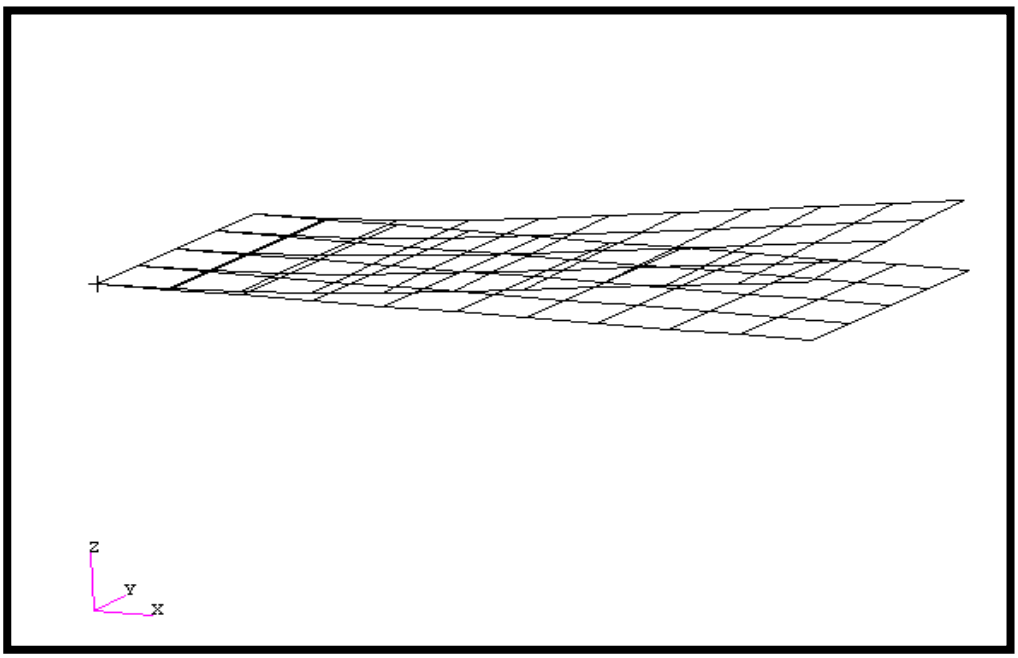
WORKSHOP PROBLEM 3

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 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 the analysis for 0.04 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

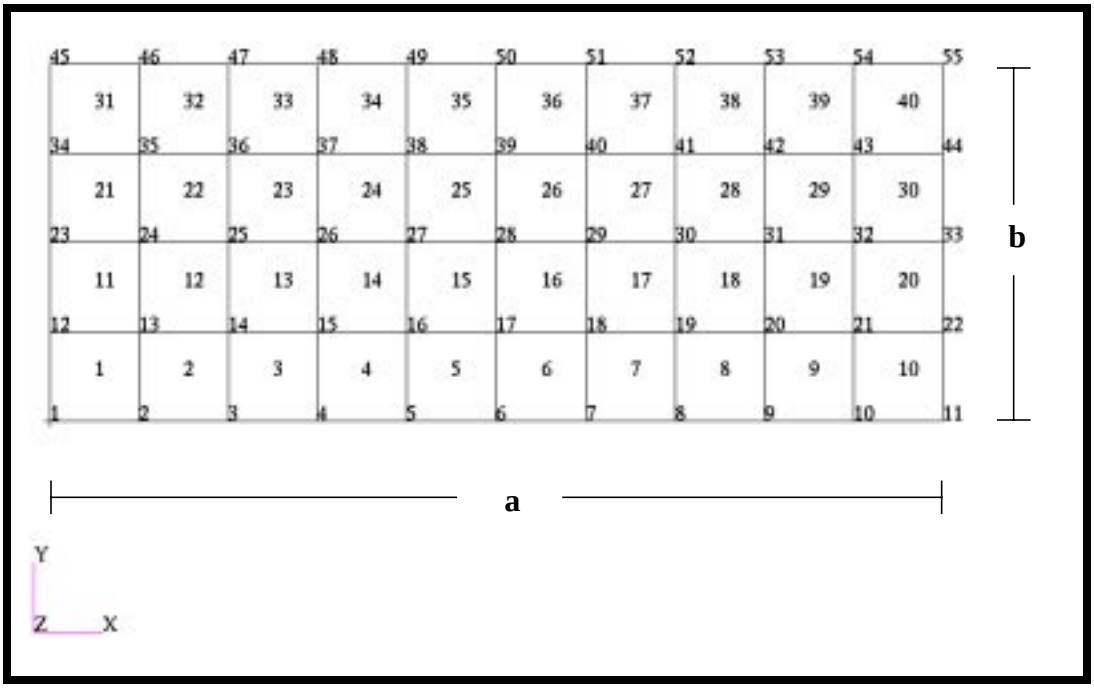


Figure 3.2 - Loads and Boundary Conditions

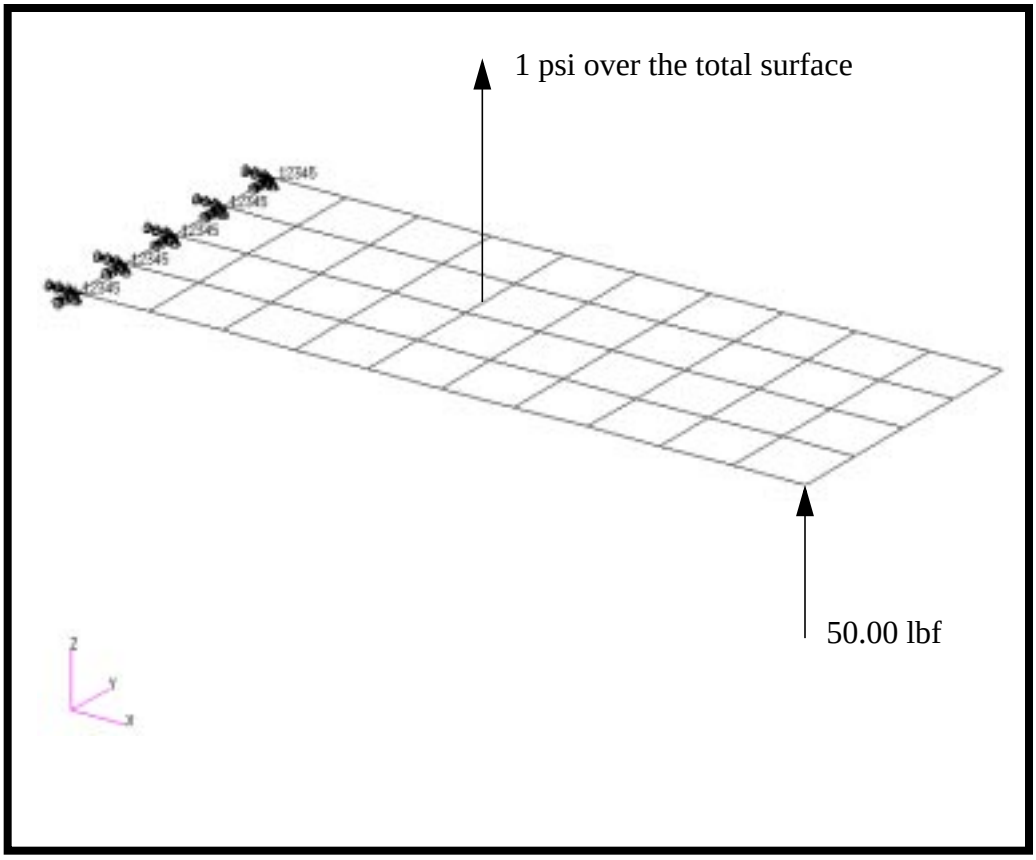


Table 3.1 - Properties

Length (a)	**5 in**
Height (b)	**2 in**
Thickness	**0.100 in**
Weight Density	**0.282 lbs/in^3**
Mass/Weight Factor	**2.59E-3 sec^2/in**
Youngs 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 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(90000.*!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.

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: ● **Equation**

Delta X: **0.0004**

X **0** **Y** **-sin(90000.*!x)**

To X **0.008**

More

Data Entry: ● **Single Value**

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

More

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

More

OK

Cancel

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...

Title: **transient_loading**

OK

Now, define the dynamic analysis parameters.

Model/Load/Dynamic Analysis...

Solution Method: ● **Direct Transient**

Under *Equivalent Viscous Damping*, input the following:

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

OK

Under *Output Requests*, unselect all except:

☒ **Displacement**

OK

Under *PARAM*, enter the following:

☒ *WTMASS* **.00259**

OK

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

Yes

File name: **direct**

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

9. 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 Time 0.0024**

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		T_3
0.0024	=	_____
0.0052	=	_____
0.02	=	_____

Nodal Displacement at Node 33

Time		T_3
0.0024	=	_____
0.0052	=	_____
0.02	=	_____

Nodal Displacement at Node 55

Time		T_3
0.0024	=	_____
0.0052	=	_____
0.02	=	_____

10. 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 - Pressure**

☐ **Load - Force**

☐ **Constraint**

Done

OK

Create the XY plot.

View/Select...

XY Style ● **XY vs Set Value**

XY Data...

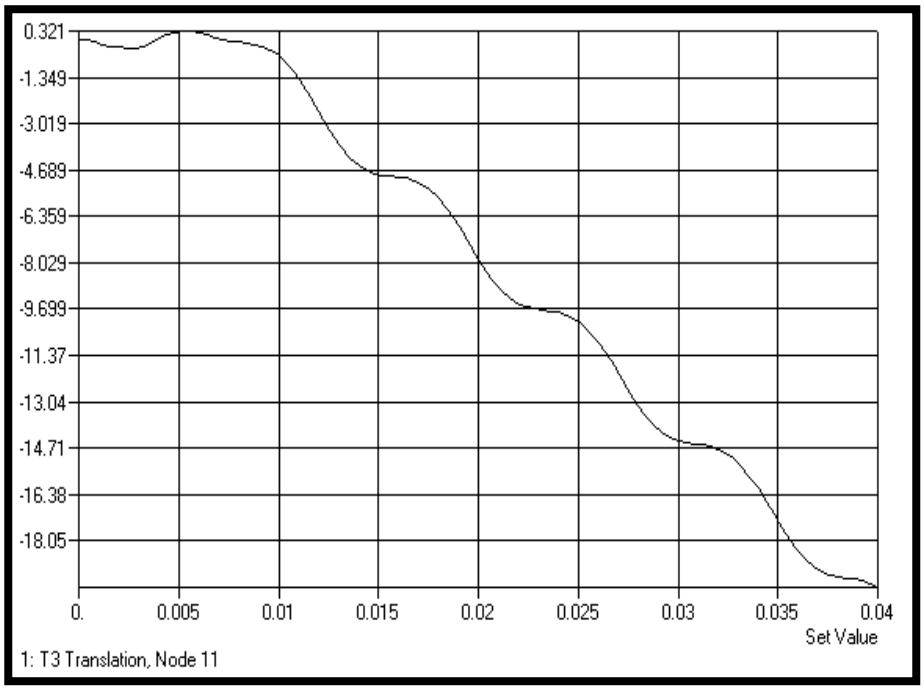
Category:	**0..Any Output**
Type:	**0..Value or Magnitude**
Output Set:	**1..Case 1 Time 0.000000**
Output Vector:	**4..T3 Translation**
Output Location/ Node:	**11**

OK

OK

The plot should appear as follows:

Figure 3.3 - XY Plot of T3 Displacement at Node 11



To unpost the XY plot, do the following:

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

Time	T3
0.0024	**-0.26233**
0.0052	**0.28239**
0.02	**0.038671**

Nodal Displacement at Node 33

Time	T3
0.0024	**-0.28827**
0.0052	**0.32209**
0.02	**0.039833**

Nodal Displacement at Node 55

Time	T3
0.0024	**-0.3115**
0.0052	**0.35709**
0.02	**0.040889**