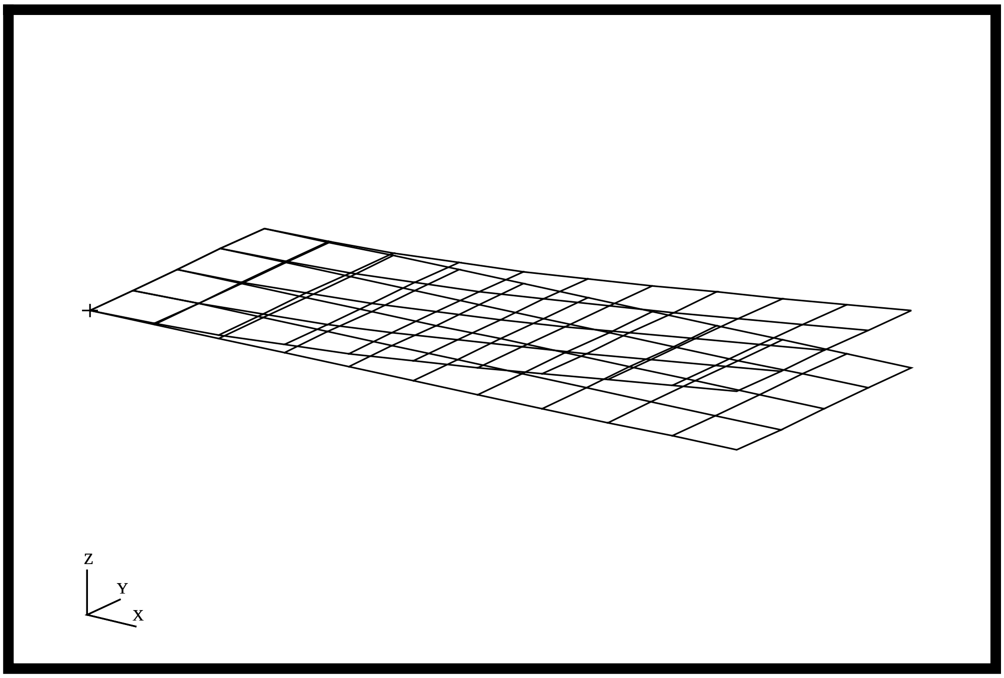
WORKSHOP PROBLEM 7

Direct Transient Response with Base Excitation



Objectives

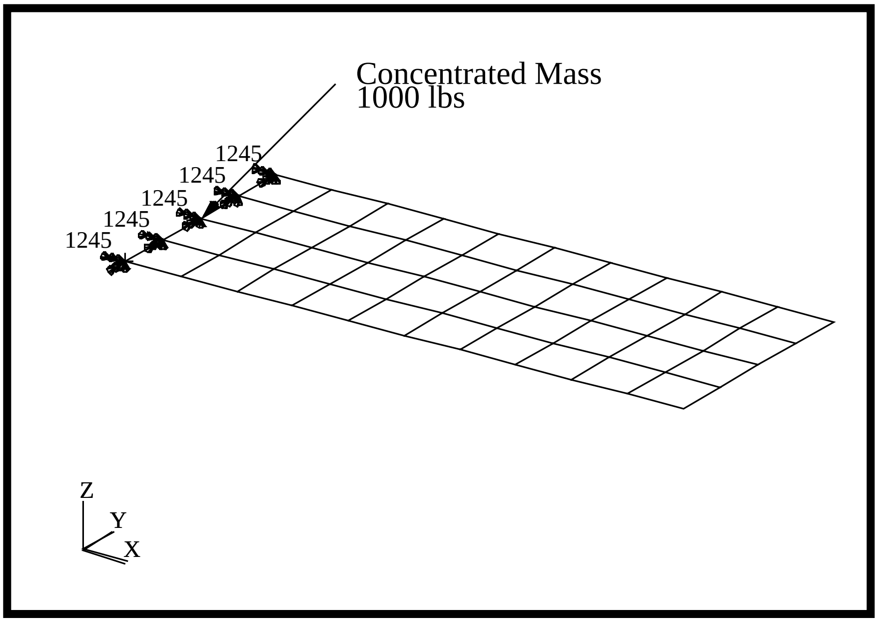
- Create a geometric representation of a flat rectangular plate.
- Use the geometry model to define an analysis model comprised of plate elements.
- Define a time-varying unit acceleration.
- Use the large mass method model.
- Submit the file for analysis in MSC/NASTRAN.
- Compute nodal displacements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

Model Description:

Using the direct method, determine the transient response to a unit acceleration sine pulse of 250 Hz applied at the base in the z-direction. A large mass of 1000 lb is applied to the base. Use a structural damping coefficient of g = 0.06 and convert this damping to equivalent viscous damping at 250 Hz.

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

Figure 7.1 - Loads and Boundary Conditions



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

OK

3. Modify the model constraints.

Modify/Edit/Constraint...

Model Brand: ● **Nodal Constraints**

OK

Select the five nodes, **Nodes 1, 12, 23, 34, and 45** along the left edge. (Hint: Use shift and left mouse button for rectangular picking.)

OK

The following step will be repeated 4 more times for the other 4 nodes.

On the *DOF* box, select these translational and rotational D.O.F.

☒ **TX** ☒ **TY** ☐ **TZ**

☒ **RX** ☒ **RY**

OK

4. Redraw and rotate the model for a better view.

View/Redraw

View/Rotate... **Dimetric**

OK

5. Create the point mass and the RBE mass.

Model/Property...

Title: **scalar_mass**

Elem/Property Type...

Other Elements: ● **Mass**

OK

In the *Property Values* box, specify the mass.

Mass, M or Mx: **1000**

OK

Cancel

To define the concentrated point mass and the RBE mass. First, assign and element to the concentrated mass.

Model/Element...

ID:	**100**
Property:	**2..scalar_mass**
Node:	**23**

OK

Next, the RBE Mass.

Type...

Other Elements: ● **Rigid**

OK

Under the *Independent* box, select the degrees of freedom.

Node:	**23**
DOF:	☐ **TX** ☐ **TY** ☒ **TZ**
	☐ **RX** ☐ **RY** ☐ **RZ**

Under the *Dependent* box, select all the nodes along the free edge.

Nodes...

Select these four nodes, **Nodes 1, 12, 34, and 45** along the left edge. Be sure that Node 23 is NOT selected.

OK

OK

Cancel

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

Model/Function...

ID:	1
Title:	time_depen_acceleration

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:		0.0002	
X	0	**Y**	sin(90000.*!x)
To X	0.004		

More

Data Entry:		● Single Value	
X	0.004	**Y**	0

More

X	0.04	**Y**	0

More

OK

Cancel

7. Create the loading conditions.

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

Model/Load/Set...

Title:	transient_response

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: **200**

Time per Step: **2e-4**

Output Interval: **1**

Advanced...

Mass Formulation: ● **Coupled**

OK

OK

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

Model/Load/Nodal...

Select **Node 23**.

OK

(highlight) **Force**

Method: ● **Constant**

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

Function Dependence: **1..time_depen_acceleration**

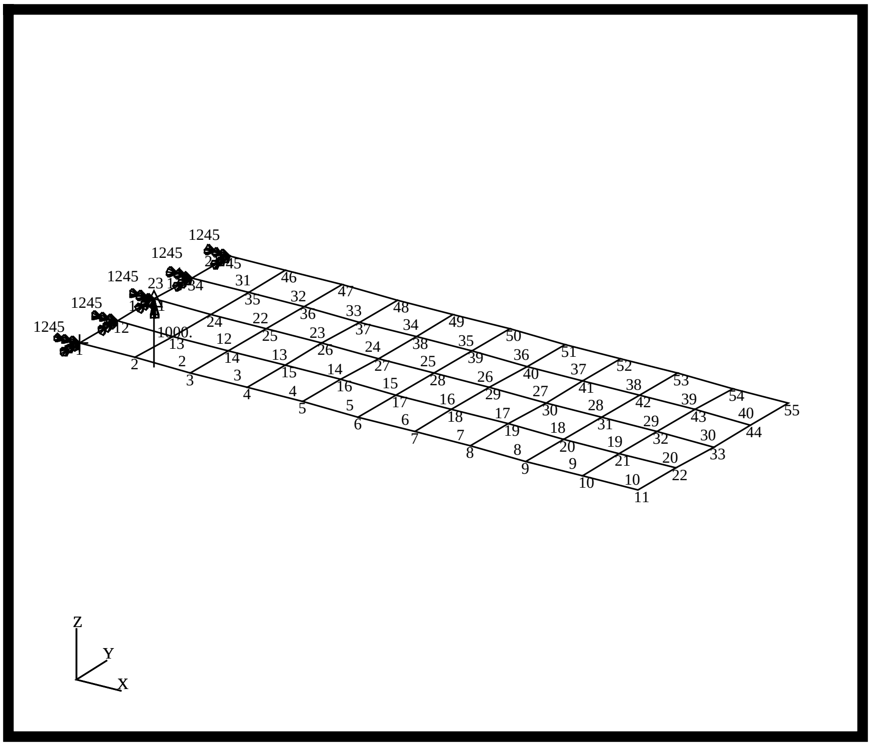
FZ ☒ **2.588**

OK

Cancel

The resulting model is shown below.

Figure 7.2



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: **prob7**

Write

☒ **Run Analysis**

Advanced...

Solution Type: ● **Direct**

OK

Problem ID: **Direct Transient Response**

OK

Under *Output Requests*, unselect all except:

☒ **Displacement**

☒ **Velocity**

☒ **Acceleration**

OK

Under *PARAM*, enter the following:

☒ *WTMASS* **.00259**

OK

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

Yes

File name: **prob7**

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:	**1..MSC/NASTRAN Case 1**
Category:	**0..Any Output**
Entity:	● **Node**
ID:	**23**

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

Nodal Displacement at Node 23

Time T3

.0 = __________

.02 = __________

.04 = __________

Nodal Displacement at Node 33

Time T3

.0 = __________

.02 = __________

.04 = __________

Velocity

Nodal Velocity at Node 23

Time T3

.0 = ___________

.02 = ___________

.04 = ___________

Nodal Velocity at Node 33

Time T3

.0 = ___________

.02 = ___________

.04 = ___________

Acceleration

Nodal Acceleration at gNode 23

Time T3

.0 = ___________

.02 = ___________

.04 = ___________

Nodal Acceleration at Node 33

Time T3

.0 = ___________

.02 = ___________

.04 = ___________

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

View/Options...

Quick Options...

Labels Off

Deselect the following:

☐ **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..MSC/NASTRAN Case 1**
Output Vector:	**4..T3 Translation**
Output Location/ Node:	**23**

OK

OK

To unpost the XY plot.

View/Select...

Model Style: ● **Draw Model**

OK

Now repeat this process to generate the XY plots by altering the *Output Vector* to **T3 displacement, velocity,** or **acceleration** at Node 23 and 33.

The results are shown below.

Figure 7.3 - Displacement at Node 23

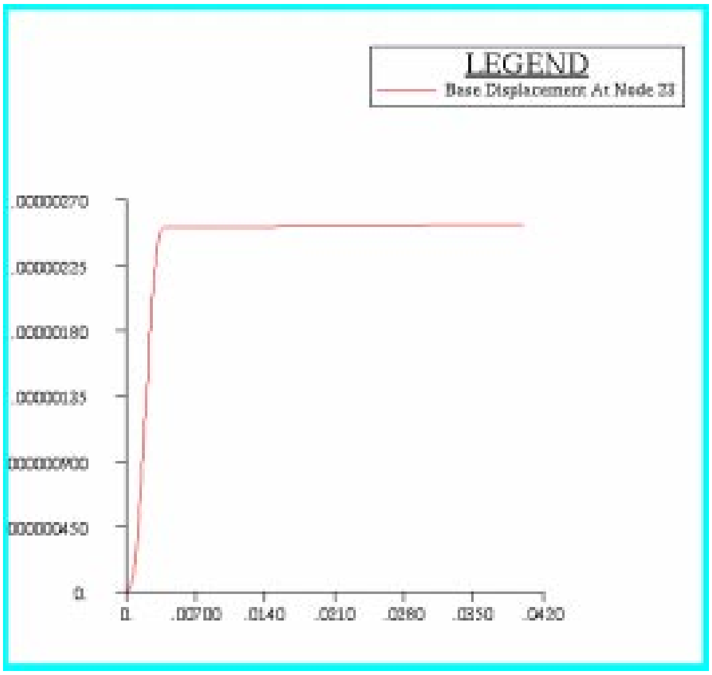


Figure 7.4 - Displacement at Node 33

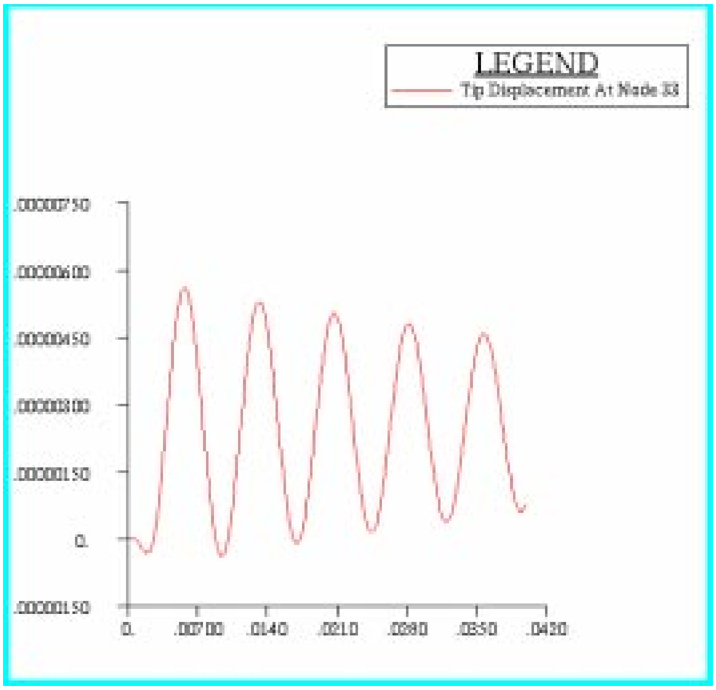


Figure 7.5 - Velocity at Node 23

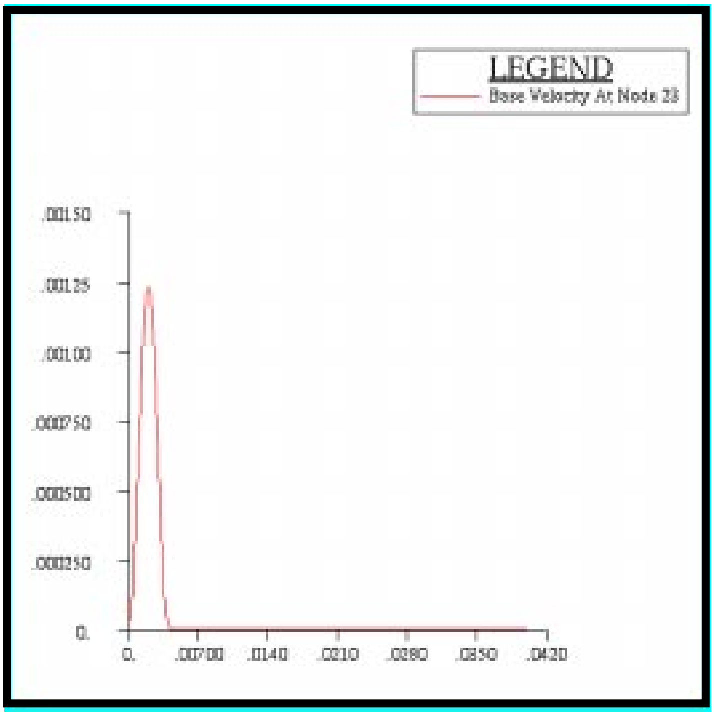


Figure 7.6 - Velocity at Node 33

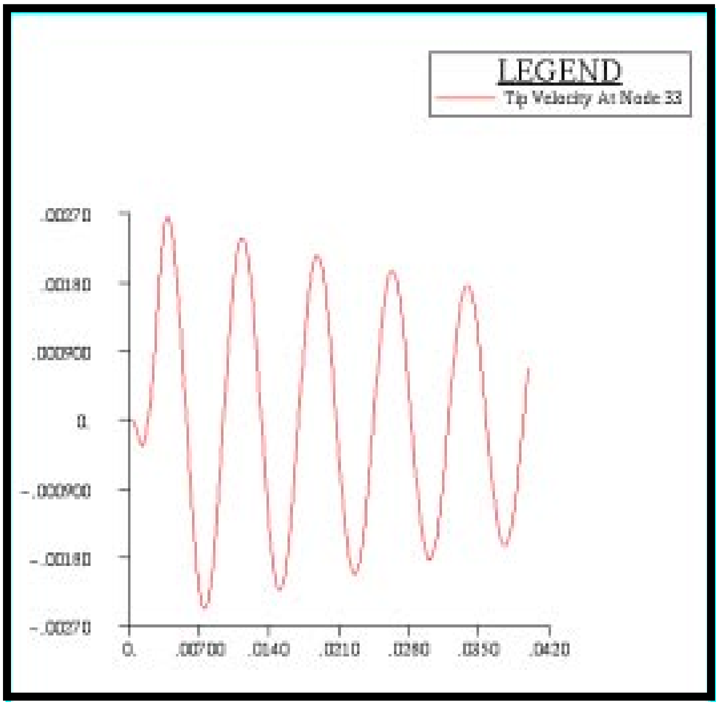


Figure 7.7 - Acceleration at Node 23

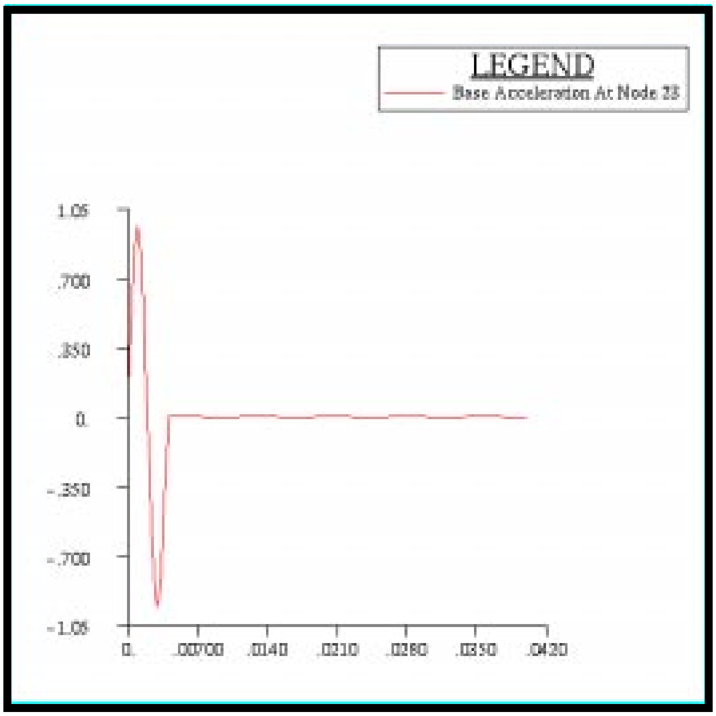
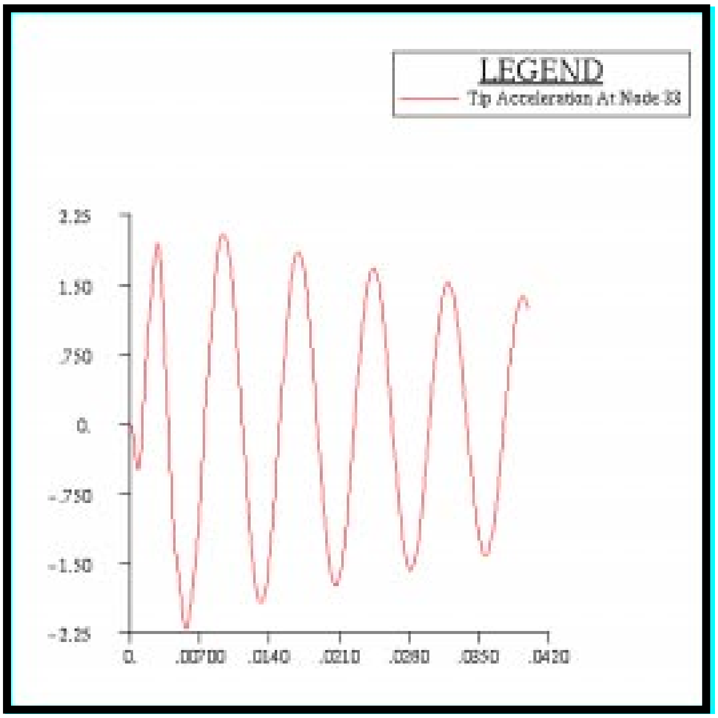


Figure 7.8 - Acceleration at Node 33



When finished, exit MSC/NASTRAN for Windows.

File/Exit

This concludes this exercise.

Displacement

Time	Node 23	Node 33
0	**0**	**0**
0.02	**2.523 E-6**	**4.588 E-6**
0.04	**2.523 E-6**	**7.213 E-7**

Velocity

Time	Node 23	Node 33
0	**1.030 E-5**	**-4.376 E-7**
0.02	**-1.358 E-7**	**1.230 E-3**
0.04	**-7.353 E-8**	**6.631 E-4**

Acceleration

Time	Node 23	Node 33
0	**0.1029**	**-4.376 E-3**
0.02	**1.624 E-4**	**-1.477**
0.04	**-1.365 E-4**	**1.242**