MSC.Software Corporation
815 Colorado Boulevard
Los Angeles, California 90041-1777
Tel: (323) 258-9111
Fax: (323) 259-3838

United States
MSC.Patran Support
Tel: 1-800-732-7284
Fax: 714-9792990

Tokyo, Japan
Tel: 81-3-3505-0266
Fax: 81-3-3505-0914

Munich, Germany
Tel: (+49)-89-43 19 87 0
Fax: (+49)-89-43 61 716

MSC.Nastran Dynamic Analysis

NAS102 EXERCISE WORKBOOK
MSC.Nastran Version 70.7
MSC.Patran Version 9.0

NA*70.7*Z*Z*Z*SM–NAS102–WBK

November 1999

If you would like to order more copies of this document, please contact MSC.Software Contracts Processing at (800) 400-4672. U.S.A. orders: All orders must be accompanied by a check or purchase order. Your order will be sent prepaid via UPS or fourth class mail and the shipping charges will be added to the invoice. F.O.B. will be the shipping point. Terms are net amount due within 30 days. Outside U.S.A. orders: Please contact your local MSC.Software office for a quotation.

DISCLAIMER

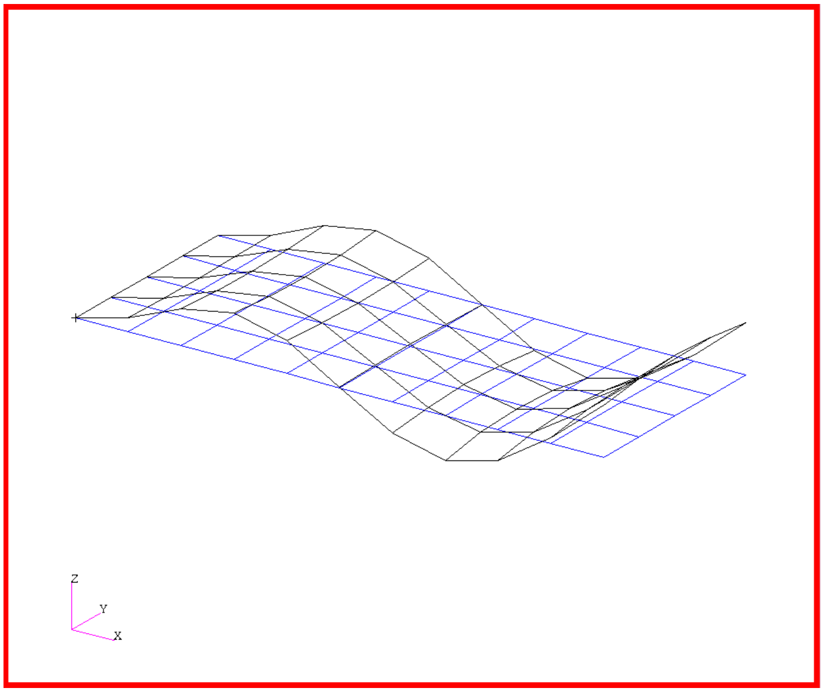
The concepts, methods, and examples presented in this text are for educational purposes only and are not intended to be exhaustive or to apply to any particular engineering problem or design. The MSC.Software Corporation assumes no liability or responsibility to any person or company for direct or indirect damages resulting from the use of any information contained herein.

Printed in U.S.A.

TABLE OF CONTENTS

WORKSHOP 1

Modal Analysis of a Flat Plate



Objectives

- Produce a MSC.Nastran input file.
- Submit the file for analysis in MSC.Nastran.
- Find the first five natural frequencies and mode shapes of the flat plate.

Model Description:

For this example, use Lanczos method to find the first five natural frequencies and mode shapes of a flat rectangular plate. One of the edges is fixed, (See Figure 1.2.). Below is a finite element representation of the rectangular plate. It also contains the geometric dimensions and the loads and boundary constraints. Table 1.1 contains the necessary parameters to construct the input file.

Figure 1.1-Grid Coordinates and Element Connectivities

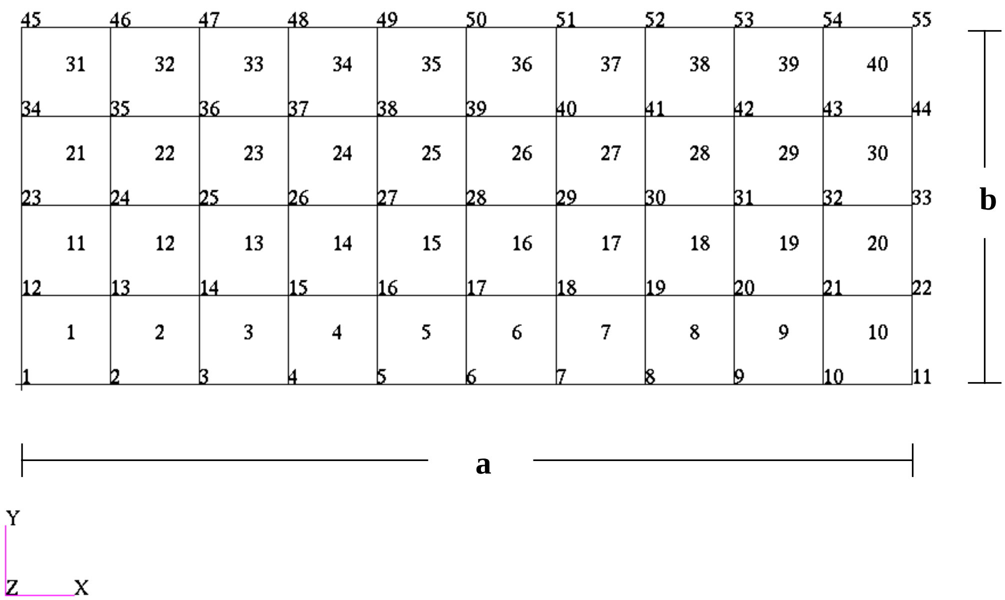


Figure 1.2-<u>Loads and Boundary Conditions</u>

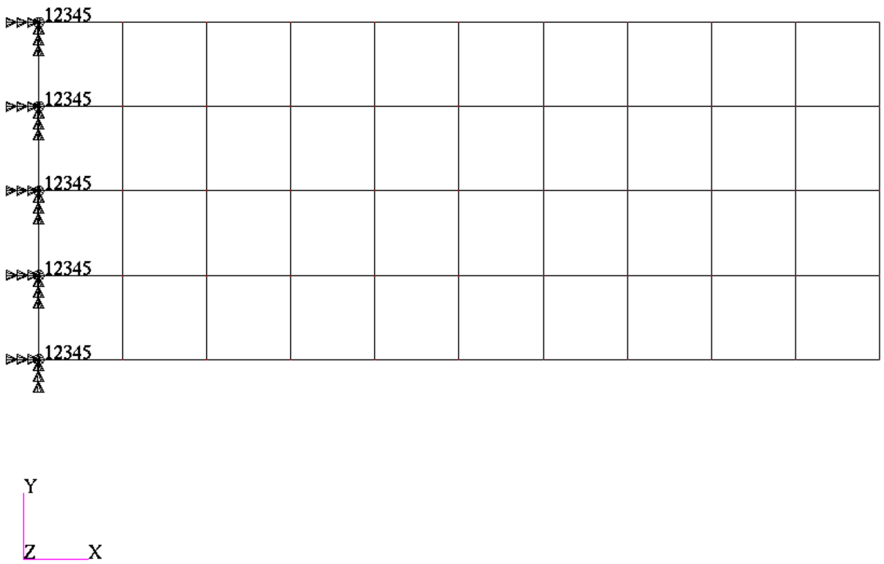


Table 1.1

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**
Elastic Modulus	**30.0E6 lbs/in^2**
Poisson’s Ratio	**0.3**

Modal Analysis of a Flat Plate

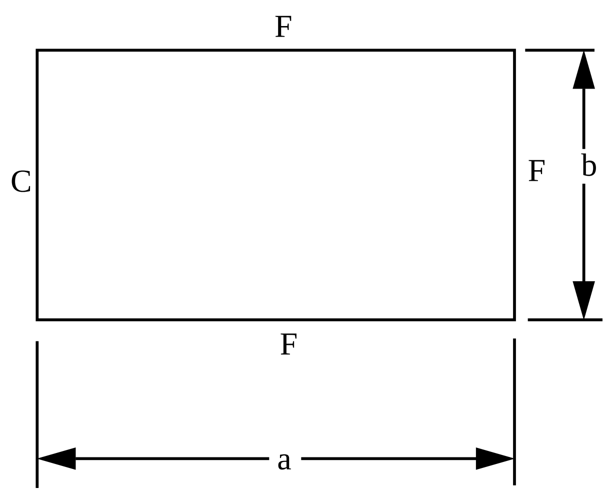
Natural Frequency: Hertz

$$f_{ij} = \frac{\lambda_{ij}^{2}}{2\pi a^{2}}\left[\frac{Eh^{3}}{12\gamma(1-\nu^{2})}\right]^{1/2}$$

where i= 1,2,3, ...

j= 1,2,3, ...

Description: Clamped-Free-Free-Free



a = length of plate

b = width of plate

h = thickness of plate

i = number of half-waves in mode shape along horizontal axis

j = number of half-waves in mode shape along vertical axis

C = clamped edge

E = modulus of elasticity

F = free edge

S = simply supported edge

γ = mass per unit area of plate (μ h for a plate material with density μ)

ν = Poisson ratio

λ_{ij}^2 and (ij)

	Mode Sequence					
a/b	**1**	**2**	**3**	**4**	**5**	**6**
0.40	3.511	4.786	8.115	13.88	21.64	23.73
	(11)	(12)	(13)	(14)	(21)	(22)
2/3	3.502	6.406	14.54	22.04	26.07	31.62
	(11)	(12)	(13)	(21)	(22)	(14)
1.0	3.492	8.525	21.43	27.33	31.11	54.44
	(11)	(12)	(21)	(13)	(22)	(23)
1.5	3.477	11.68	21.62	39.49	53.88	61.99
	(11)	(12)	(21)	(22)	(13)	(31)
2.5	3.456	17.99	21.56	57.46	60.58	106.5
	(11)	(12)	(21)	(22)	(31)	(32)

$\nu = 0.3$

Suggested Exercise Steps

- Explicitly generate a finite element representation of the plate structure. (i.e., The grids (GRID) and element connectivities (CQUAD4) should be defined manually.)
- Define material (MAT1) and element (PSHELL) properties.
- Apply the fixed boundary constraints (SPC1).
- Prepare the model for a normal modes analysis (SOL 103 and PARAMs).
  - PARAM, WTMASS, 0.00259
  - PARAM, COUPMASS, 1
- Generate an input file and submit it to the MSC.Nastran solver for normal modes analysis.
- Review the results, specifically the eigenvalues.

```
ID SEMINAR,PROB1
```

```
CEND
```

```
BEGIN BULK
```

1	2	3	4	5	6	7	8	9	10

1	2	3	4	5	6	7	8	9	10

ENDDATA

Exercise Procedure:

1. Users who are not utilizing MSC.Patran for generating an input file should go to Step 11, otherwise, proceed to step 2.

2. Create a new database named **prob1.db**.

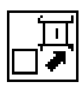
   File/New

New Database Name	**prob1**
OK	

   In the *New Model Preferences* form, set the following:

Tolerance	◆ **Default**
Analysis Code:	**MSC/NASTRAN**
OK	

3. Activate the entity labels by selecting the Show Labels icon on the tool-bar.

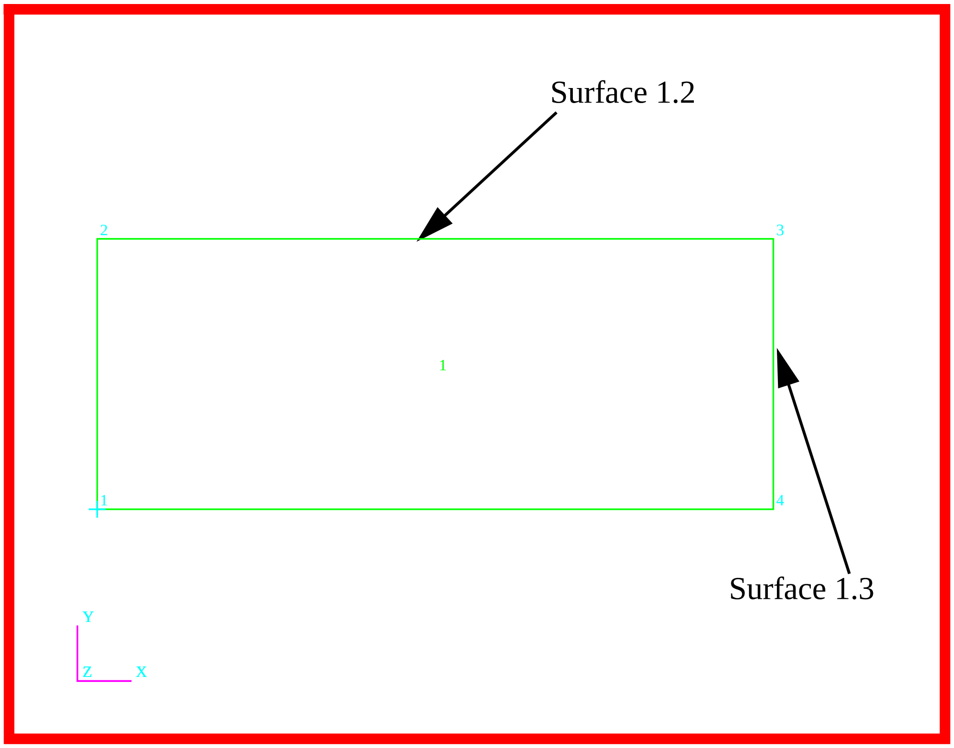
   Show Labels

4. Create a surface.

   ◆ **Geometry**

Action:	**Create**
Object:	**Surface**
Method	**XYZ**
Vector Coordinates List	**<5, 2, 0>**
Origin Coordinates List	**[0, 0, 0]**
Apply	

Figure 1.3-The surface should resemble the output below.



5. Create the finite element model and mesh the surface.

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh Seed**
Type:	**Uniform**

◆ **Number of Elements**

Number =	**10**
Curve List *(see Figure 1.3)*	**Surface 1.2**

Apply

5a. Change the number of mesh seeds to 4 and select the right edge.

Number =	**4**
Curve List *(see Figure 1.3)*	**Surface 1.3**

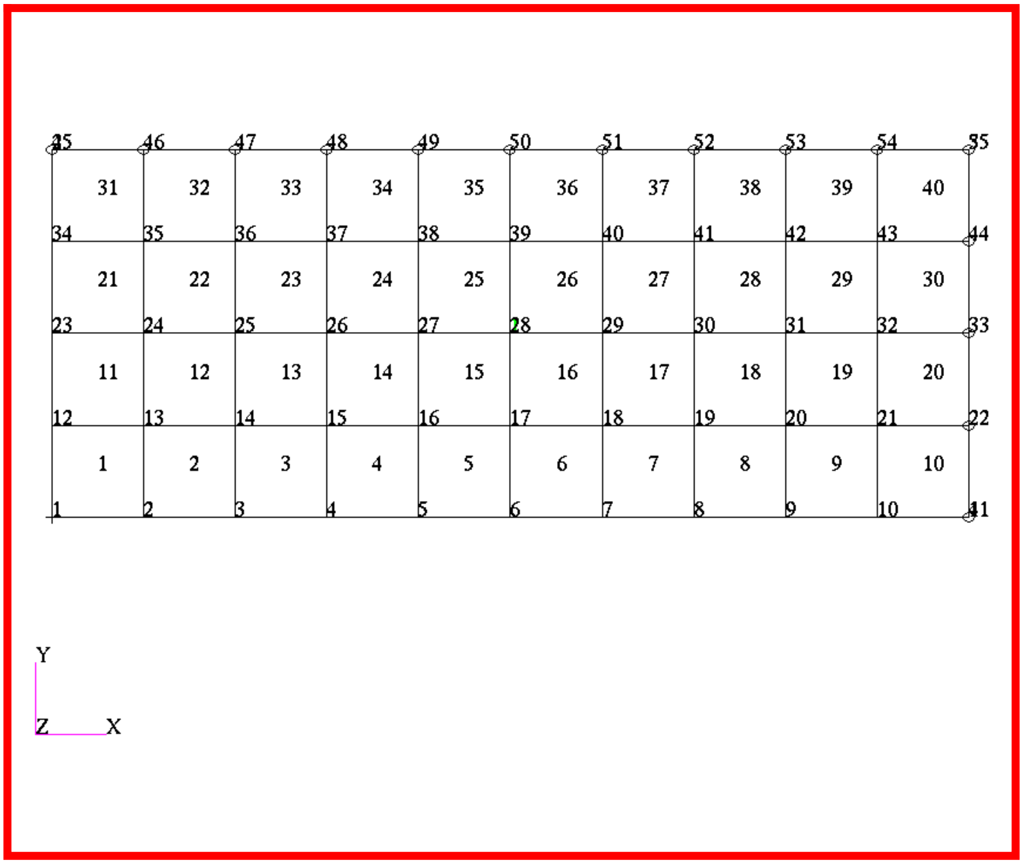
Apply

5b. Mesh the surface.

Action:	**Create**
Object:	**Mesh**
Type:	**Surface**
Surface List	**Surface 1**

Apply

Figure 1.4-The model should appear as below.



6. Create a set of material properties for the plate.

◆ **Materials**

Action:	**Create**

Object:	**Isotropic**
Method:	**Manual Input**
Material Name	**mat_1**
Input Properties...	
Elastic Modulus =	**30.0E6**
Poisson Ratio =	**.3**
Density =	**.282**
OK	
Apply	

7. Define the plate thickness.

◆ **Properties**

Action:	**Create**
Dimension:	**2D**
Type:	**Shell**
Property Set Name	**plate**
Input Properties...	
Material Name *(Select from* ***Material Property Sets*** *box.)*	**m:mat_1**
Thickness	**0.100**
OK	
Select Members	**Surface 1**
Add	
Apply	

8. Apply constraints to the model.

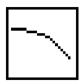
8a. Constrain the left edge from moving through all degrees of freedom.

◆ **Load/BC's**

Action:	**Create**

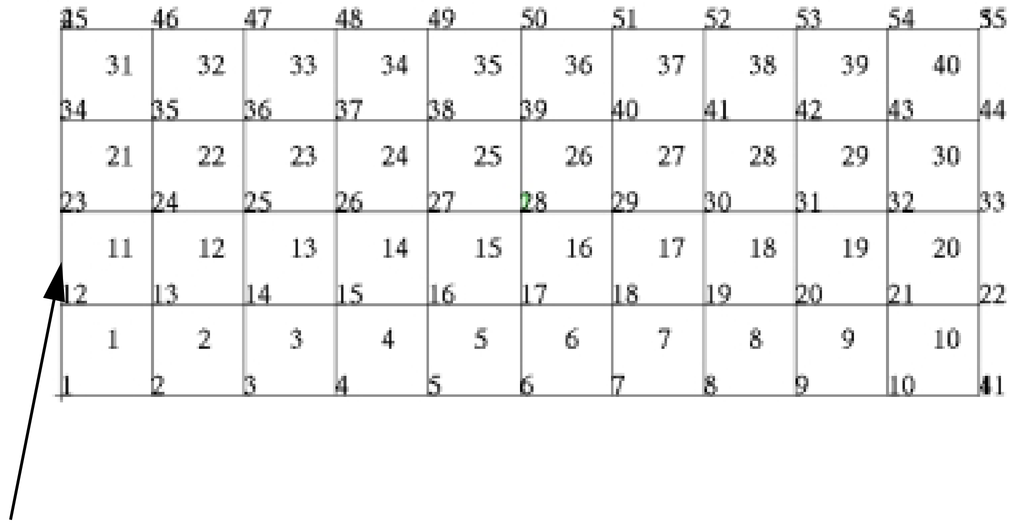
Object:	**Displacement**
Type:	**Nodal**
New Set Name	**fixed**
Input Data...	
Translations <T1 T2 T3>	**<0, 0, 0>**
Rotations <R1 R2 R3>	**<0, 0, >**
Analysis Coordinate Frame	**Coord 0**
OK	
Select Application Region...	

Select the curve or edge icon.

 **Curve or edge**

Select Geometry Entities *(see Figure 1.5)*	**Surface 1.1**
Add	
OK	
Apply	

Figure 1.5



Surface 1.1

9. Run the analysis.

Before the complete input deck is generated for this analysis, a file that contains only the model data needs to be created. This file is to be used in later workshops.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method	**Model Only**
Job Name	plate

Apply

10. Now, you will generate the input file for analysis.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method	**Analysis Deck**

Job Name	**prob1**
Translation Parameters...	
Data Output:	**XDB and Print**
OK	
Solution Type...	
Solution Type:	**◆ NORMAL MODES**
Solution Parameters...	
Mass Calculation:	**Coupled**
Data Deck Echo:	**Unsorted**
Wt. -Mass Conversion =	**.00259**
OK	
OK	
Subcase Create...	
Available Subcases	**Default**
Subcase Parameters...	
Number of Desired Roots =	**5**
OK	
Output Requests...	

Under ***Output Requests***, highlight:

SPCFORCES(SORT1,Real)=All FEM

Delete

OK

Apply

Cancel

Apply

An MSC.Nastran input file called **prob1.bdf** will be generated. The process of translating your model into an input file is called Forward Translation. The Forward Translation is complete when the Heartbeat turns green. MSC.Patran Users should proceed to step 12.

Generating an input file for MSC.Nastran Users:

MSC.Nastran users can generate an input file using the data from Table 1.1. The result should be similar to the output below.

11. MSC.Nastran Input File: **prob1.dat**

```
ID SEMINAR, PROB1
SOL 103
TIME 600
CEND
TITLE = NORMAL MODES EXAMPLE
ECHO = UNSORTED
SUBCASE 1
   SUBTITLE= USING LANCZOS
   METHOD = 1
   SPC = 1
   VECTOR=ALL
BEGIN BULK
PARAM   COUPMASS 1
PARAM    WTMASS  .00259
EIGRL    1                       5
PSHELL   1       1       .1      1               1
CQUAD4   1       1       1       2       13      12
=,*1,=,*1,*1,*1,*1
=8
CQUAD4   11      1       12      13      24      23
=,*1,=,*1,*1,*1,*1
=8
CQUAD4   21      1       23      24      35      34
=,*1,=,*1,*1,*1,*1
=8
CQUAD4   31      1       34      35      46      45
=,*1,=,*1,*1,*1,*1
=8
MAT1     1       3.+7            .3      .282
GRID     1               0.      0.      0.
=,*1,=,*0.5,==
=9
GRID     12              0.      .5      0.
=,*1,=,*0.5,==
=9
GRID     23              0.      1.      0.
=,*1,=,*0.5,==
=9
```

```
GRID     34              0.      1.5     0.
=,*1,=,*0.5,==
=9
GRID     45              0.      2.      0.
=,*1,=,*0.5,==
=9
SPC1     1       12345  1       12      23      34      45
ENDDATA
```

11a. We will also create an input file **plate.bdf**, which contains all the relevant model data. This file is to be used in later workshops.

```
GRID    1               0.      0.      0.
=,*1,=,*0.5,==
=9
GRID    12              0.      .5      0.
=,*1,=,*0.5,==
=9
GRID    23              0.      1.      0.
=,*1,=,*0.5,==
=9
GRID    34              0.      1.5     0.
=,*1,=,*0.5,==
=9
GRID    45              0.      2.      0.
=,*1,=,*0.5,==
=9
PSHELL  1       1       .1      1               1
CQUAD4  1       1       1       2       13      12
=,*1,=,*1,*1,*1,*1
=8
CQUAD4  11      1       12      13      24      23
=,*1,=,*1,*1,*1,*1
=8
CQUAD4  21      1       23      24      35      34
=,*1,=,*1,*1,*1,*1
=8
CQUAD4  31      1       34      35      46      45
=,*1,=,*1,*1,*1,*1
=8
MAT1    1       3.+7            .3      .282
SPC1    1       12345   1       12      23      34      45
```

Submitting the input file for analysis:

12. Submit the input file to MSC.Nastran for analysis.

    12a. To submit the MSC.Patran **.bdf** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran prob1.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

    12b. To submit the MSC.Nastran **.dat** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran prob1 scr=yes**. Monitor the run using the UNIX **ps** command.

13. When the run is completed, edit the **prob1.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing WARNING messages indicate modeling errors.

14. While still editing **prob1.f06**, search for the word:

R E A L (spaces are necessary)

1st = __________Hz

2nd = __________Hz

3rd = __________Hz

4th = __________Hz

5th = __________Hz

Comparison of Results

15. Compare the results obtained in the **.f06** file with the follwoing results:

R E A L   E I G E N V A L U E S

MODE NO.	EXTRACTION ORDER	EIGENVALUE	RADIANS	CYCLES	GENERALIZED MASS	GENERALIZED STIFFNESS
1	1	7.056994E+05	8.400591E+02	1.336996E+02	1.000000E+00	7.056994E+05
2	2	1.878432E+07	4.334088E+03	6.897916E+02	1.000000E+00	1.878432E+07
3	3	2.811467E+07	5.302327E+03	8.438915E+02	1.000000E+00	2.811467E+07
4	4	1.931709E+08	1.389859E+04	2.212030E+03	1.000000E+00	1.931709E+08
5	5	2.234434E+08	1.494802E+04	2.379052E+03	1.000000E+00	2.234434E+08

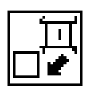
16. MSC.Nastran Users have finished this exercise. MSC.Patran Users should proceed to the next step.

17. Proceed with the Reverse Translation process, that is attaching the **prob1.xdb** results file into MSC.Patran. To do this, return to the Analysis form and proceed as follows:

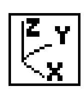
◆ **Analysis**

Action:	**Attach XDB**
Object:	**Result Entities**
Method	**Local**
Select Results File...	
Select Results File	**prob1.xdb**
OK	
Apply	

To simplify the view, turn off the entity labels using the toolbar.


Hide Labels

In addition, switch to a 3 view isometric view point.



Iso 3 View

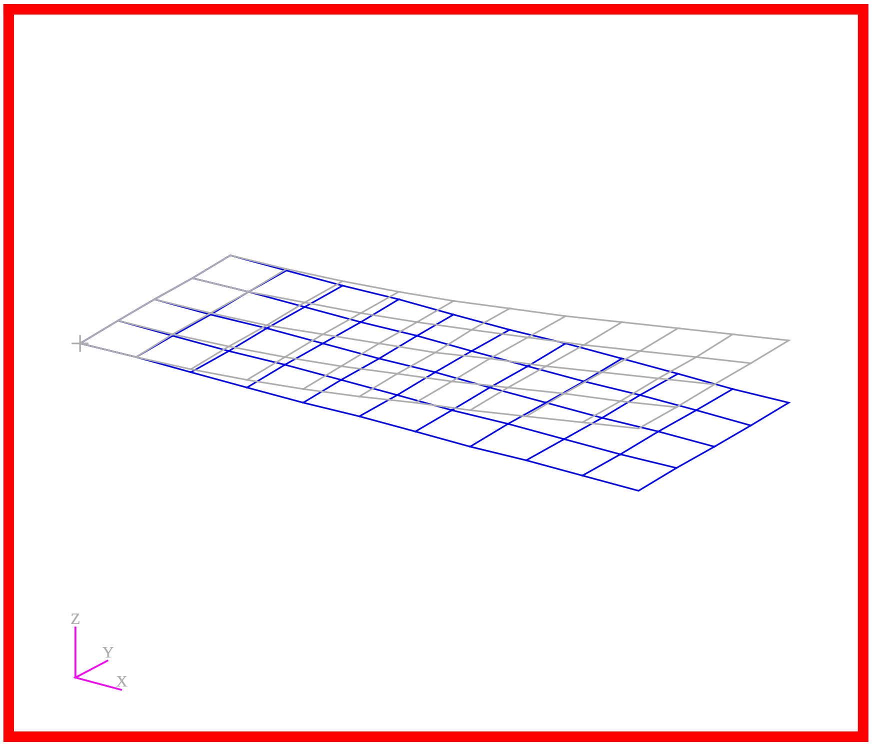
When the translation is complete bring up the ***Results*** form.

◆ **Results**

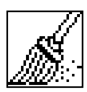
Action:	**Create**
Object:	**Deformation**
Select Results Cases	**Default, A1:Mode 1 : Freq. = 133.7**
Select Deformation Result	**Eigenvectors, Translational**
Apply	

The results should resemble Figure 1.6.

Figure 1.6



To reset the graphics, click on this icon:

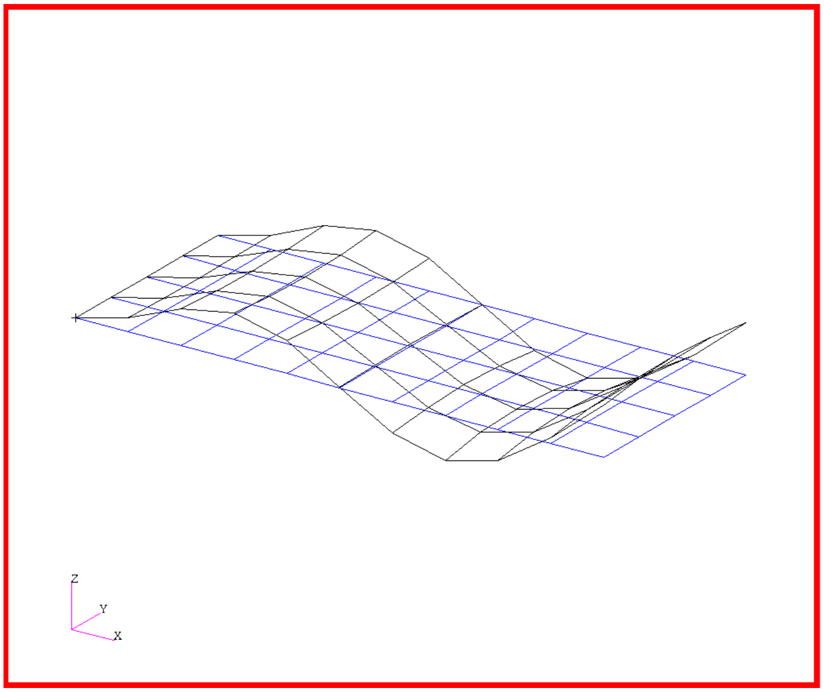
 **Reset Graphics**

Repeat the procedure to view the other mode shapes.

Quit MSC.Patran when you are finished with this exercise.

WORKSHOP 2

Modal Analysis of A Flat Plate using Static Reduction



Objectives

- Reduce the dynamic math model, created in Workshop 1, to one with fewer degrees of freedom.
- Produce a MSC.Nastran input file.
- Submit the file for analysis in MSC.Nastran.
- Find the first five natural frequencies and mode shapes of the flat plate.

Model Description:

For this example, reduce the dynamic math model created in Workshop 1, using static reduction. Then find the first five natural frequencies and mode shapes using the Automatic Givens method. Use the points indicated in Figure 2.2 for the A-set.

Figure 2.1-Grid Coordinates and Element Connectivities

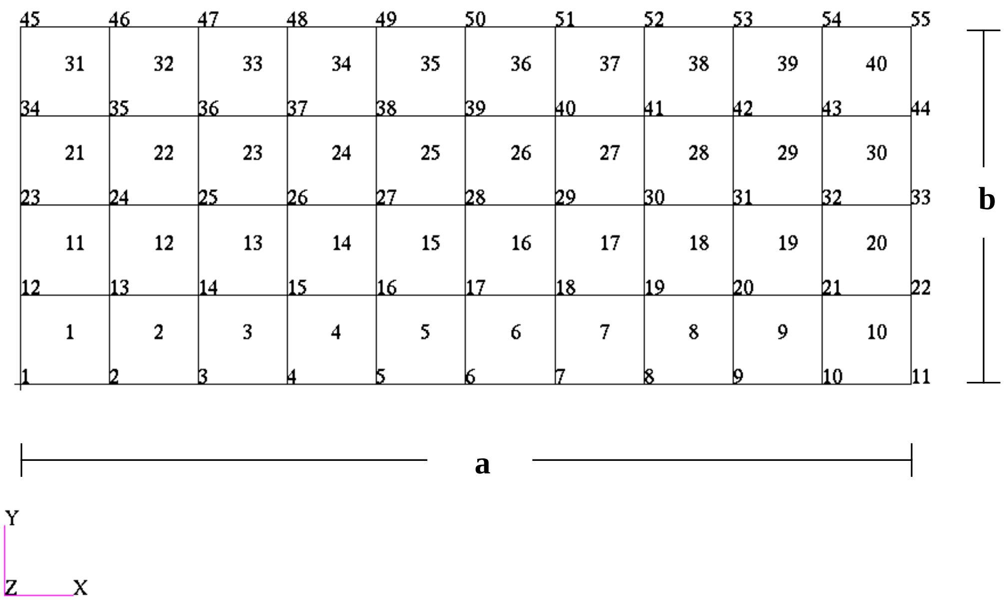


Figure 2.2-Loads and Boundary Conditions

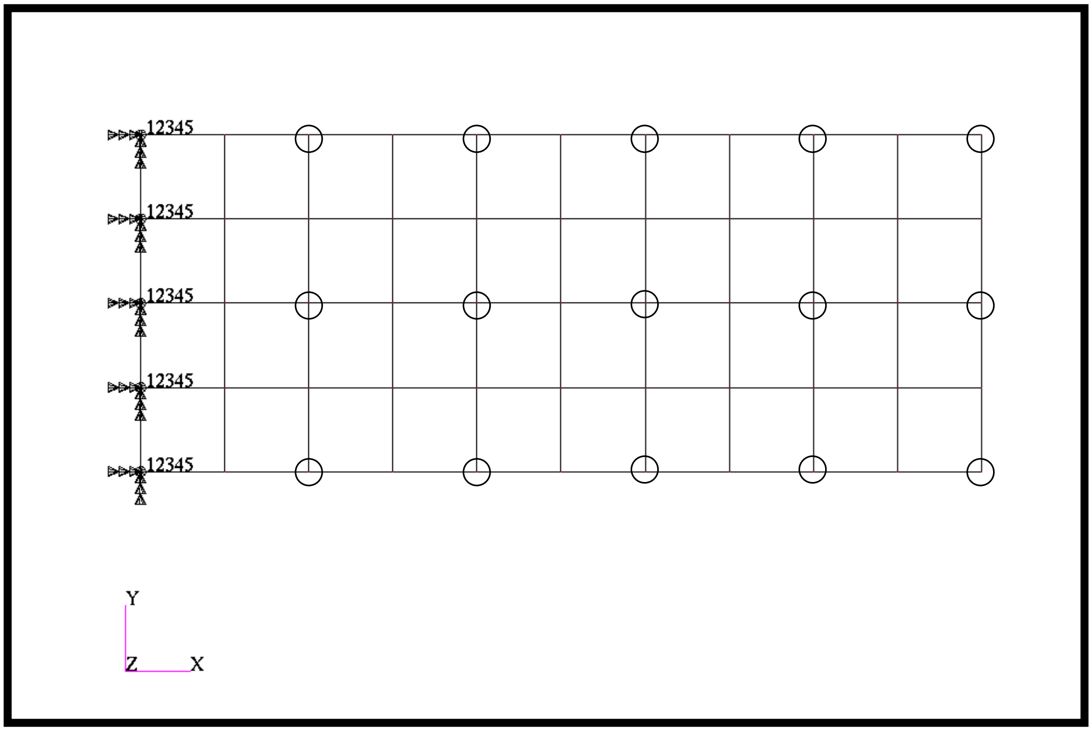


Table 2.1

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**
Elastic Modulus	**30.0E6 lbs/in^2**
Poisson's Ratio	**0.3**

Suggested Exercise Steps

- Reference a previously created dynamic math model, **plate.bdf**, by using the INCLUDE statement.
- Prepare the model for a normal modes analysis (SOL 103 and PARAMs).
  - PARAM, WTMASS, 0.00259
  - PARAM, COUPMASS, 1
- Define degrees of freedom in the analysis set (ASET) for grids indicated in Figure 2.2.
- Generate an input file and submit it to the MSC.Nastran solver for normal modes analysis.
- Review the results, specifically the eigenvalues.

```
ID SEMINAR,PROB2
CEND
BEGIN BULK
```

1	2	3	4	5	6	7	8	9	10

1	2	3	4	5	6	7	8	9	10

ENDDATA

Exercise Procedure:

1. Users who are not utilizing MSC.Patran for generating an input file should go to Step 7, otherwise, proceed to step 2.

2. Create a new database named **prob2.db**.

   File/New Database

   New Database Name **prob2**

   OK

   In the *New Model Preference* form set the following:

   Tolerance ◆ **Default**

   Analysis Code: **MSC/NASTRAN**

   OK

3. Create the model by importing an existing MSC.Nastran input file, (**plate.bdf**).

   ◆ **Analysis**

   Action: **Read Input File**

   Object: **Model Data**

   Method **Translate**

   Select Input File...

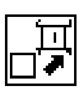
   Select File **plate.bdf**

   OK

   Apply

   OK

4. Activate the entity labels by selecting the Show Labels icon on the tool-bar.

    **Show Labels**

5. Add the pre-defined constraints into the **Default** load case.

◆ **Load Cases**

Action:	**Modify**
Select Load Case to Modify *(Highlight the following:)*	**Default**
Select Individual Load/BCs *(Highlight the following:)*	**Displ_spc1.1**

OK

Apply

6. Create the new analysis deck.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method	**Analysis Deck**
Jobname:	**prob2**

Translation Parameters...

Data Output:	**XDB and Print**

OK

Solution Type...

Solution Type:	◆ **NORMAL MODES**

Solution Parameters...

Mass Calculation:	**Coupled**
Data Deck Echo:	**Unsorted**
Wt. -Mass Conversion =	**.00259**

OK

OK

Direct Text Input...

In the ***Bulk Data Section,*** type in the following:

```
ASET1, 345, 3, 5, 7, 9, 11
```

```
ASET1, 345, 25, 27, 29, 31, 33
ASET1, 345, 47, 49, 51, 53, 55
```

OK

Subcase Create...

Available Subcases **Default**

Subcase Parameters...

Extraction Method: **Automatic Givens**

Number of Desired Roots = **5**

OK

Output Requests...

Under ***Output Requests***, highlight:

SPCFORCES(SORT1,Real)=All FEM

Delete

OK

Apply

Cancel

Apply

An MSC.Nastran input file called **prob2.bdf** will be generated. This process of translating your model into an input file is called the Forward Translation. The Forward Translation is complete when the Heartbeat turns green. MSC.Patran Users should proceed to step 8.

Generating an input file for MSC.Nastran Users:

MSC.Nastran users can generate an input file using the data from Table 2.1. The result should be similar to the output below.

7. MSC.Nastran input file: **prob2.dat**

```
ID SEMINAR, PROB2
SOL 103
TIME 10
CEND
TITLE = REDUCTION PROCEDURES, NORMAL MODES EXAMPLE
SUBTITLE = USING STATIC REDUCTION
ECHO = UNSORTED
SUBCASE 1
   SUBTITLE=USING LANCZOS
   METHOD = 1
   SPC = 1
   VECTOR=ALL
BEGIN BULK
EIGR,1,AGIV,,,,5
PARAM, COUPMASS, 1
PARAM, WTMASS, 0.00259
INCLUDE ’plate.bdf’
$
$ SELECT A-SET, STATIC REDUCTION IS DONE AUTOMATICALLY
$
ASET1,345,3,5,7,9,11
ASET1,345,25,27,29,31,33
ASET1,345,47,49,51,53,55
ENDDATA
```

Submitting the input file for analysis:

8. Submit the input file to MSC.Nastran for analysis.

    8a. To submit the MSC.Patran **.bdf** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran prob2.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

    8b. To submit the MSC.Nastran **.dat** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran prob2 scr=yes**. Monitor the run using the UNIX **ps** command.

9. When the run is completed, edit the **prob2.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing WARNING messages indicate modeling errors.

10. While still editing **prob2.f06**, search for the word:

R E A L (spaces are necessary)

1st = ___________Hz

2nd = ___________Hz

3rd = ___________Hz

4th = ___________Hz

5th = ___________Hz

Comparison of Results

11. Compare the results obtained in the **.f06** file with the following results:

MODE NO.	EXTRACTION ORDER	EIGENVALUE	RADIANS	CYCLES	GENERALIZED MASS	GENERALIZED STIFFNESS
1	43	7.057452E+05	8.400864E+02	1.337039E+02	1.000000E+00	7.057452E+05
2	45	1.880877E+07	4.336908E+03	6.902404E+02	1.000000E+00	1.880877E+07
3	44	2.818009E+07	5.308492E+03	8.448727E+02	1.000000E+00	2.818009E+07
4	42	1.956108E+08	1.398609E+04	2.225956E+03	1.000000E+00	1.956108E+08
5	41	2.367820E+08	1.538772E+04	2.449032E+03	1.000000E+00	2.367820E+08

12. **MSC.Nastran Users have finished this exercise. MSC.Patran Users should proceed to the next step.**

13. Proceed with the Reverse Translation process, that is attaching the **prob2.xdb** results file into MSC.Patran. To do this, return to the Analysis form and proceed as follows:

◆ **Analysis**

Action: **Attach XDB**

Object: **Result Entities**

Method **Local**

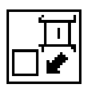
Select Results File...

Select Results File **prob2.xdb**

OK

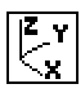
Apply

To simplify the view, turn off the entity labels using the toolbar.



Hide Labels

In addition, switch to a 3 view isometric view point.



Iso 3 View

When the translation is complete bring up the ***Results*** form.

◆ **Results**

Action: **Create**

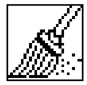
Object: **Deformation**

Select Results Case **Default, A1:Mode1: Freq. =133.7**

Select Deformation Result **Eigenvectors, Translational**

Apply

Reset the graphics by clicking on this icon:

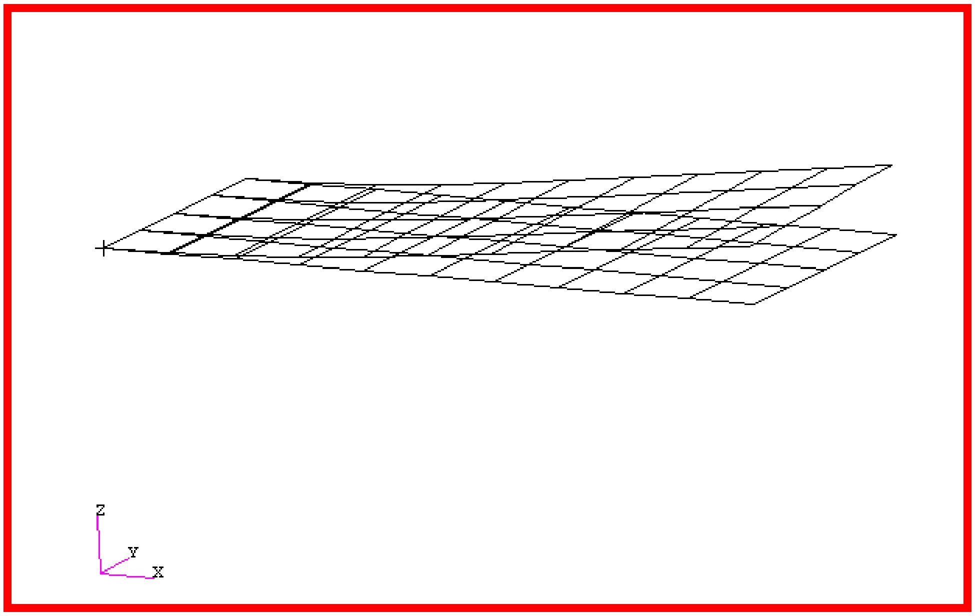
 **Reset Graphics**

You can go back and select any *Results Case*, *Fringe Results or Deformation Results* you are interested in.

Quit MSC.Patran when you are finished with this exercise.

WORKSHOP 3

Direct Transient Response Analysis



Objectives

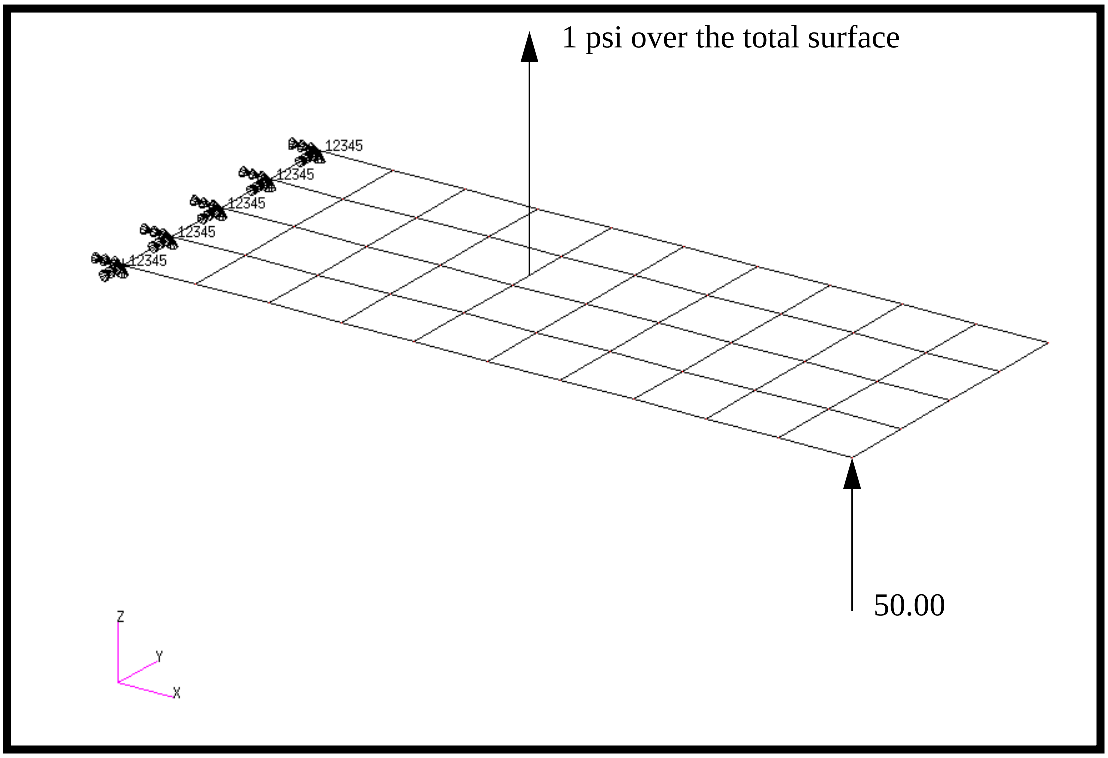
- Define time-varying excitation.
- Produce a MSC.Nastran input file from dynamic math model created in Workshop 1.
- Submit the file for analysis in MSC.Nastran.
- Compute nodal displacements for desired time domain.

Model Description:

Using the direct method, determine the transient response of the flat rectangular plate, created in Workshop 1, under time-varying excitation. This example structure shall be excited by 1 psi pressure load over the total surface of the plate varying at 250Hz. In addition, a 50 lb force is applied at a corner of the tip also varying at 250Hz 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 250Hz. Carry 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 3.1-Loads and Boundary Conditions



Suggested Exercise Steps

- Reference previously created dynamic math model, **plate.bdf**, by using the INCLUDE statement.
- Define the time-varying pressure loading (PLOAD2, LSEQ and TLOAD2). (Hint, be certain to specify phase angle since the applied loads are out-of-phase).
- Define the time-varying tip load (DAREA and TLOAD2). (Again, be certain to specify the phase angle).
- Combine the time-varying loads (DLOAD).
- Specify integration time steps (TSTEP).
- Prepare the model for a direct transient analysis (SOL 109).
- Specify the structural damping and convert this damping to equivalent viscous damping.
  - PARAM, G, 0.06
  - PARAM, W3, 1571.0
- Request response in terms of nodal displacement at grid points 11, 33 and 55.
- Generate an input file and submit it to the MSC.Nastran solver for direct transient analysis.
- Review the results, specifically the nodal displacements and xy-plot output.

```
ID SEMINAR,PROB3
```

```
CEND
```

```
BEGIN BULK
```

1	2	3	4	5	6	7	8	9	10

1	2	3	4	5	6	7	8	9	10

ENDDATA

Exercise Procedure:

1. Users who are not utilizing MSC.Patran for generating an input file should go to Step 13, otherwise, proceed to step 2.

2. Open a new database named **prob3.db**.

   File/New

   New Database Name **prob3**

   OK

   In the *New Model Preferences* form, set the following:

   Tolerance ◆ **Default**

   Analysis Code: **MSC/NASTRAN**

   OK

3. Create the model by importing an existing MSC.Nastran input file, **(plate.bdf)**.

   ◆ **Analysis**

   Action: **Read Input File**

   Object: **Model Data**

   Method **Translate**

   Select Input File

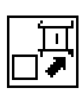
   Select File **plate.bdf**

   OK

   Apply

   OK

4. Activate the entity labels by selecting the Show Labels icon on the tool-bar.

    **Show Labels**

5. Add the pre-defined constraints into the default load case.

◆ **Load Cases**

Action:	**Create**
Load Case Name	**transient_response**
Load Case Type:	**Time Dependent**
Assign/Prioritize Loads/BCs	
Select Individual Load/BCs *(Select from menu.)*	**Displ_spc1.1**
OK	
Apply	

6. Create a time-dependent field for the transient response of the pressure loading.

◆ **Fields**

Action:	**Create**
Object:	**Non Spatial**
Method	**Tabular Input**
Field Name	**time_dependent_pressure**
[Options ...]	
Maximum Number of t	**21**
OK	
Input Data ...	
Map Function to Table...	
PCL Expression f'(t):	**sind(360.*250.*'t)**
Start Time	**0.0**
End Time	**0.008**
Number of Points	**20**
Apply	

Cancel

In the ***Time/Frequency Scalar Table Data*** window, add the following to Row 21:

	Time(t)	Value
21	**0.04**	**0.0**

OK

Apply

7. Create another time-dependent field for the transient response of the nodal force.

◆ **Fields**

Action:	**Create**
Object:	**Non Spatial**
Method	**Tabular Input**
Field Name	**time_dependent_force**

[Options ...]

Maximum Number of t	**21**

OK

Input Data ...

Map Function to Table...

PCL Expression f'(t)	**-sind(360*250*'t)**
Start Time	**0.0**
End Time	**0.008**
Number of Points	**20**

Apply

Cancel

In the ***Time/Frequency Scalar Table Data*** window, add the following to Row 21:

	Time(t)	Value
21	0.04	0.0

OK

Apply

8. Create the time dependent pressure.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Pressure**
Type:	**Element Uniform**
New Set Name	**pressure**
Target Element Type:	**2D**

Input Data...

Top Surf Pressure	**-1**

Note: The default direction of pressure in MSC.Patran is opposite from default MSC.Nastran assumption.

** Time/Freq. Dependence:* *(Select from the* ***Time Dependent Fields*** *box)*	**f:time_dependent_pressure**

OK

Select Application Region...

◆ **FEM**

Select 2D Elements or Edge *(Select all elements)*	**Elem 1:40**

Add

OK

Apply

9. Create the time-dependent nodal force.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Force**
Type:	**Nodal**
New Set Name	**force**

Input Data...

Spatial Dependence *Force <F1 F2 F3>*	**<0 0 50>**
** Time/Freq. Dependence:* *(Select from the **Time Dependent Fields** box)*	**f:time_dependent_force**

OK

Select Application Region...

◆ **FEM**

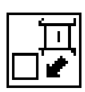
Select Nodes	**Node 11**

Add

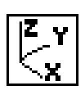
OK

Apply

To simplify the view, turn off the entity labels using the toolbar.


Hide Labels

In addition, switch to a 3 view isometric view point.



Iso 3 View

Action:	**Plot Markers**

Under ***Assigned Load/BC Sets***, highlight:

Displ_spc1.1

Force_force

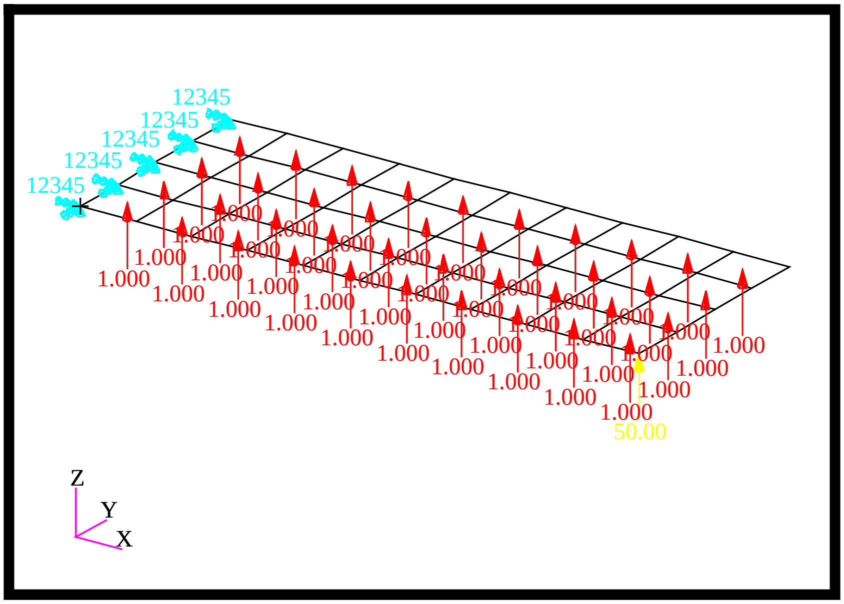
Press_pressure

Under ***Select Groups***, highlight:

default_group

The result should be similar to **Figure 3.2**.

Figure 3.2-The model with loads and boundary conditions applied.



10. Create the analysis.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Analysis Deck**
Job Name	**prob3**
Solution Type...	
Translation Parameters...	
Data Output:	**XDB and Print**
OK	
Solution Type:	◆ **TRANSIENT RESPONSE**

Formulation:	**Direct**
Solution Parameters...	
Mass Calculation:	**Coupled**
Wt.-Mass Conversion =	**.00259**
Struct. Damping Coeff. =	**0.06**
W3, Damping Factor =	**1571**
OK	
OK	
Subcase Create...	
Available Subcases *(Select from menu.)*	**transient_response**
Subcase Parameters...	
Time Recovery Points	**DEFINE TIME STEPS ...**
Number of Time Steps =	**100**
Delta - T *(Hit Return to Input Data.*	**.0004**
OK	
OK	
Output Requests...	
Form Type:	**Advanced**

Under ***Output Requests***, highlight:

SPCFORCES(SORT1,Real)=All FEM

Delete	
Output Requests:	select **DISPLACEMENT(...**
Options/Sorting:	**By Freq/Time**
Modify	
OK	

Apply

Cancel

Subcase Select...

Subcases Selected: **Default**
(Click to de-select.)

Subcases for Solution Sequence: 109 **transient_response**
(Click to select.)

OK

Apply

An MSC.Nastran input file called **prob3.bdf** will be generated. This process of translating your model into an input file is called the Forward Translation. The Forward Translation is complete when the Heartbeat turns green. MSC.Patran Users should proceed to step 14.

Generating an input file for MSC.Nastran Users:

MSC.Nastran users can generate an input file using the data previously stated. The result should be similar to the output below.

11. MSC.Nastran input file: **prob3.dat**

```
ID SEMINAR, PROB3
SOL 109
TIME 30
CEND
TITLE= TRANSIENT RESPONSE WITH TIME DEPENDENT PRESSURE AND POINT LOADS
SUBTITLE= USE THE DIRECT METHOD
ECHO= PUNCH
SPC= 1
SET 1= 11, 33, 55
DISPLACEMENT= 1
SUBCASE 1
DLOAD= 700 $ SELECT TEMPORAL COMPONENT OF TRANSIENT LOADING
LOADSET= 100 $ SELECT SPACIAL DISTRIBUTION OF TRANSIENT LOADING
TSTEP= 100 $ SELECT INTEGRATION TIME STEPS
$
OUTPUT (XYPLOT)
XGRID=YES
YGRID=YES
XTITLE= TIME (SEC)
YTITLE= DISPLACEMENT RESPONSE AT LOADED CORNER
XYPLOT DISP RESPONSE / 11 (T3)
YTITLE= DISPLACEMENT RESPONSE AT CENTER TIP
XYPLOT DISP RESPONSE / 33 (T3)
YTITLE= DISPLACEMENT RESPONSE AT OPPOSITE CORNER
XYPLOT DISP RESPONSE / 55 (T3)
$
BEGIN BULK
PARAM, COUPMASS, 1
PARAM, WTMASS, 0.00259
$
$ PLATE MODEL DESCRIBED IN NORMAL MODES EXAMPLE
$
INCLUDE 'plate.bdf'
$
$ SPECIFY STRUCTURAL DAMPING
$ 3 PERCENT AT 250 HZ. = 1571 RAD/SEC.
$
PARAM, G, 0.06
```

```
PARAM, W3, 1571.
$
$ APPLY UNIT PRESSURE LOAD TO PLATE
$
LSEQ, 100, 300, 400
$
PLOAD2, 400, 1., 1, THRU, 40
$
$ VARY PRESSURE LOAD (250 HZ)
$
TLOAD2, 200, 300, , 0, 0., 8.E-3, 250., -90.
$
$ APPLY POINT LOAD OUT OF PHASE WITH PRESSURE LOAD
$
TLOAD2, 500, 600, , 0, 0., 8.E-3, 250., 90.
$
DAREA, 600, 11, 3, 1.
$
$ COMBINE LOADS
$
DLOAD, 700, 1., 1., 200, 50., 500
$
$ SPECIFY INTERGRATION TIME STEPS
$
TSTEP, 100, 100, 4.0E-4, 1
$
ENDDATA
```

Submitting the input file for analysis:

12. Submit the input file to MSC.Nastran for analysis.

    12a. To submit the MSC.Patran **.bdf** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran prob3.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

    12b. To submit the MSC.Nastran **.dat** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran prob3 scr=yes**. Monitor the run using the UNIX **ps** command.

13. When the run is completed, use **plotps** utility to create a postscript file, **prob3.ps**, from the binary plot file **prob3.plt**. The displacement response plots for Grids 11, 33 and 55 are shown in figures 3.2, 3.3 and 3.4**.**

14. Edit the **prob3.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing WARNING messages indicate modeling errors.

15. While still editing **prob3.f06**, search for the word:

D I S P L (spaces are necessary)

Displacement at Grid 11

Time T3

.0024 = __________

.0052 = __________

.02 = __________

Displacement at Grid 33

Time T3

.0024 = __________

.0052 = __________

.02 = __________

Displacement at Grid 55

Time		T3
.0024	=	___________
.0052	=	___________
.02	=	___________

Comparison of Results

16. Compare the results obtained in the **.f06** file with the following results:

```
   POINT-ID =         11
                                     D I S P L A C E M E N T   V E C T O R

   TIME       TYPE          T1             T2             T3             R1             R2             R3
0.0              G      0.0            0.0            0.0            0.0            0.0            0.0
4.000000E-04     G      0.0            0.0           -2.173625E-02   1.104167E-02   1.050818E-02   0.0
8.000000E-04     G      0.0            0.0           -7.204904E-02   2.847414E-02   2.852519E-02   0.0
1.200000E-03     G      0.0            0.0           -1.433462E-01   4.082027E-02   4.915178E-02   0.0
     .
     .
     .
3.879996E-02     G      0.0            0.0           -3.726422E-02  -6.629907E-05   1.039267E-02   0.0
3.919996E-02     G      0.0            0.0           -2.122380E-02  -1.431050E-05   5.916678E-03   0.0
3.959996E-02     G      0.0            0.0           -2.998187E-03  -7.089762E-06   8.371174E-04   0.0
3.999996E-02     G      0.0            0.0            1.535974E-02   5.380207E-06  -4.281030E-03   0.0

   POINT-ID =         33
                                     D I S P L A C E M E N T   V E C T O R

   TIME       TYPE          T1             T2             T3             R1             R2             R3
0.0              G      0.0            0.0            0.0            0.0            0.0            0.0
4.000000E-04     G      0.0            0.0           -1.122398E-02   9.220218E-03   6.138594E-03   0.0
8.000000E-04     G      0.0            0.0           -4.424753E-02   2.576699E-02   2.014980E-02   0.0
1.200000E-03     G      0.0            0.0           -1.030773E-01   3.819036E-02   3.922388E-02   0.0
     .
     .
     .
3.879996E-02     G      0.0            0.0           -3.729695E-02   1.898676E-05   1.037927E-02   0.0
3.919996E-02     G      0.0            0.0           -2.121863E-02   3.488550E-05   5.907703E-03   0.0
3.959996E-02     G      0.0            0.0           -3.002583E-03  -2.228106E-07   8.361273E-04   0.0
3.999996E-02     G      0.0            0.0            1.535096E-02  -3.032754E-05  -4.274252E-03   0.0

   POINT-ID =         55
                                     D I S P L A C E M E N T   V E C T O R

   TIME       TYPE          T1             T2             T3             R1             R2             R3
0.0              G      0.0            0.0            0.0            0.0            0.0            0.0
4.000000E-04     G      0.0            0.0           -2.849185E-03   7.791447E-03   4.611430E-03   0.0
8.000000E-04     G      0.0            0.0           -1.992890E-02   2.322436E-02   1.681028E-02   0.0
1.200000E-03     G      0.0            0.0           -6.643156E-02   3.540079E-02   3.501805E-02   0.0
     .
     .
     .
3.879996E-02     G      0.0            0.0           -3.722652E-02   1.035188E-04   1.039059E-02   0.0
3.919996E-02     G      0.0            0.0           -2.115454E-02   8.268487E-05   5.912832E-03   0.0
3.959996E-02     G      0.0            0.0           -2.998628E-03   6.654292E-06   8.371378E-04   0.0
3.999996E-02     G      0.0            0.0            1.529953E-02  -6.482315E-05  -4.277684E-03   0.0
```

17. MSC.Nastran Users have finished this exercise. MSC.Patran Users should proceed to the next step.

18. Proceed with the Reverse Translation process, that is attaching the **prob3.xdb** results file into MSC.Patran. To do this, return to the Analysis form and proceed as follows:

◆ **Analysis**

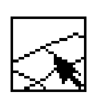
Action:	**Attach XDB**
Object:	**Result Entities**
Method	**Local**
Select Results File...	
Select File	**prob3.xdb**
OK	
Apply	

When the translation is complete bring up the **Results** form.

◆ **Results**

Action:	**Create**
Object:	**Graph**
Select Results Cases	**Transient_response, 0 of 101 subcases**
Filter Method	**All**
Filter	
Apply	
Close	
y:	**Result**
Select y Result:	**Displacement, Translational**
Quanity:	**Z Component**
x:	**Global Variable**
Variable:	**Time**

Select the **target entities** form by clicking on this Icon



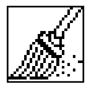
Target Entities

Target Entities

Select Nodes: **Node 11**

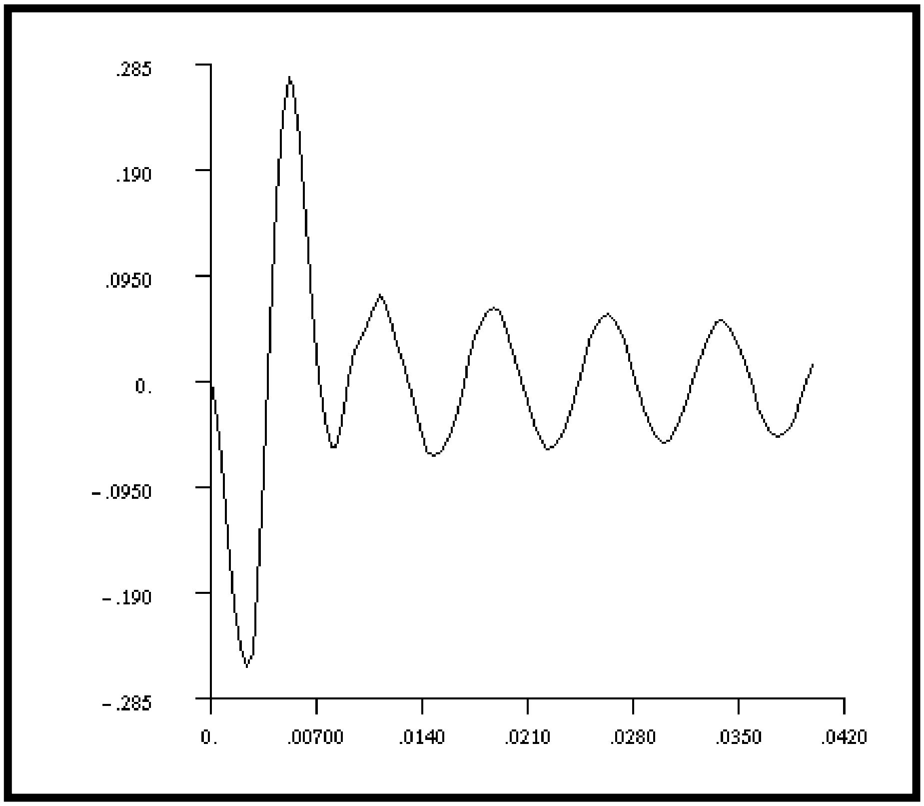
Apply

You may reset the graphics by clicking on this icon :



Reset Graphics

Figure 3.3-Displacement Response at Node 11

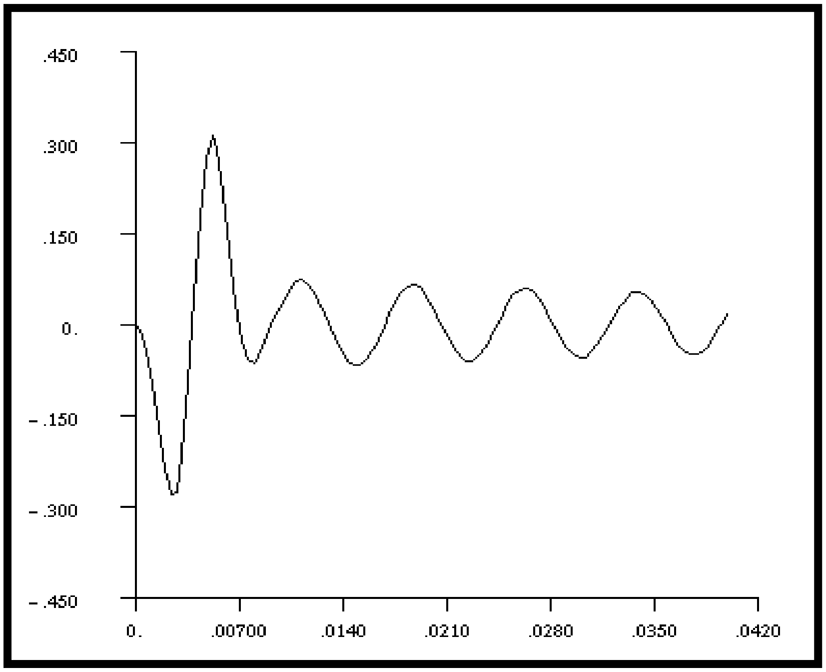


To Plot Node 33 and 55, simply select them..

Select Nodes: **Node 33**

Apply

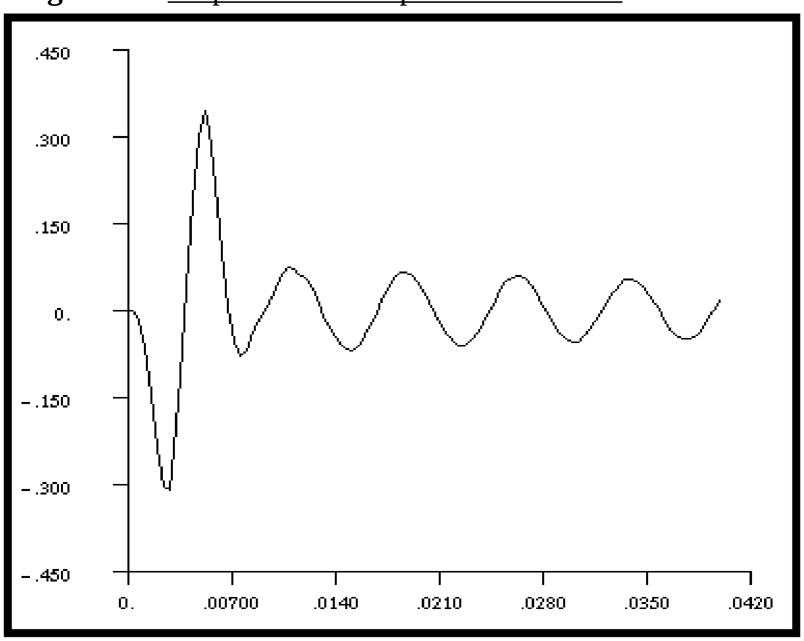
Figure 3.4-Displacement Response at Node 33



Select Nodes: **Node 55**

Apply

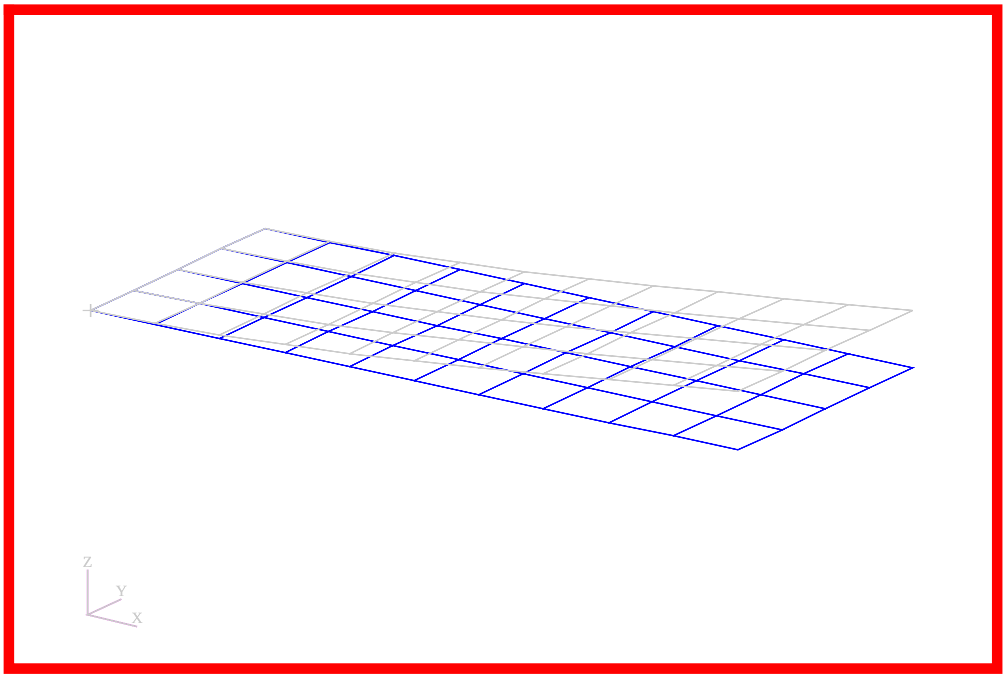
Figure 3.5-Displacement Response at Node 55



Quit MSC.Patran when you are finished with this exercise.

WORKSHOP 4

Modal Transient Response Analysis



Objectives

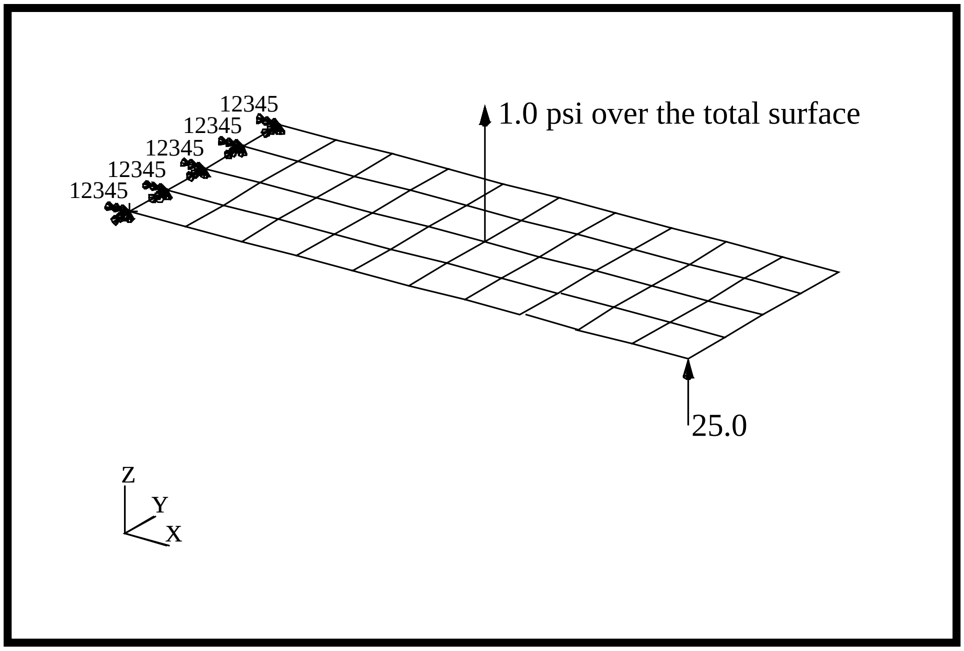
- Define time-varying excitation.
- Produce a MSC.Nastran input file from a dynamic math model, created in Workshop 1.
- Submit the file for analysis in MSC.Nastran.
- Compute nodal displacements for desired time domain.

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 $\zeta = 0.03$ for all nodes. 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



Suggested Exercise Steps

- Reference previously created dynamic math model, **plate.bdf**, by using the INCLUDE statement.
- Specify modal damping as a tabular function of natural frequency (TABDMP1).
- Define the time-varying pressure loading (PLOAD2, LSEQ and TLOAD2).
- Define the time-varying tip load (DAREA and TLOAD2).
- Define the time delay term in the equations of the dynamic loading function (DELAY).
- Combine the time-varying loads (DLOAD).
- Specify integration time steps (TSTEP).
- Prepare the model for a modal transient analysis (SOL 112).
- Request response in terms of nodal displacement at grid 11, 33, and 55.
- Generate an input file and submit it to the MSC.Nastran solver for normal modes analysis.
- Review the results, specifically the nodal displacements.

```
ID SEMINAR,PROB4
```

```
CEND
```

```
BEGIN BULK
```

1	2	3	4	5	6	7	8	9	10

1	2	3	4	5	6	7	8	9	10

ENDDATA

Exercise Procedure:

1. Users who are not utilizing MSC.Patran for generating an input file should go to Step 11, otherwise, proceed to step 2.

2. Create a new database and named **prob4.db**.

File/New

New Database Name **prob4**

OK

In the *New Model Preferences* form, set the following:

Tolerance ◆ **Default**

Analysis code: **MSC/NASTRAN**

OK

3. Create the model by importing an existing MSC.Nastran input file, **(plate.bdf)**.

◆ **Analysis**

Action: **Read Input File**

Object: **Model Data**

Method: **Translate**

Select Input File

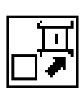
Select Input File **plate.bdf**

OK

Apply

OK

4. Activate the entity labels by selecting the Show Labels icon on the tool-bar.

 **Show Labels**

5. Add the pre-defined constraints into a newly defined load case.

◆ **Load Cases**

Action:	**Create**
Load Case Name:	**transient_response**
Load Case Type:	**Time Dependent**
Assign/Prioritize Loads/BCs	
Select Individual Load/BCs *(Select from menu.)*	**Displ_spc1.1**
OK	
Apply	

6. Create a time-dependent field for the pressure loading.

◆ **Fields**

Action:	**Create**
Object:	**Non Spatial**
Method:	**Tabular Input**
Field Name:	**time_dependent_pressure**
Options ...	
Maximum Number of t:	**21**
OK	
Input Data ...	
Map Function to Table...	
PCL Expression f'(t):	**sind(360.*250.*'t)**
Start time:	**0.0**
End time:	**0.008**
Number of Points:	**20**
Apply	
Cancel	

Go back to the ***Time/Frequency Scalar Table Data*** window, go down to row 21, and add the following:

	Time(t)	Value
21	0.04	0.0

OK

Apply

7. Create a time-dependent field for the nodal force.

◆ **Fields**

Action:	**Create**
Object:	**Non Spatial**
Method:	**Tabular Input**
Field Name:	**time_dependent_force**

Options...

Maximum Number of t:	**32**

OK

Input Data...

Map Function to Table...

PCL Expression f'(t):	**sind(360.*250.*('t-0.004))**
Start time:	**0.004**
End time:	**0.012**
Number of Points:	**31**

Apply

Cancel

Go back to the *Time/Frequency Scalar Table Data* window, go down to row 32, and add the following:

	Time(t)	Value
32	0.04	0.0

OK

Apply

8. Create the time-dependent pressure.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Pressure**
Type:	**Element Uniform**
New Set Name:	pressure
Target Element Type:	**2D**

Input Data...

Top Surf Pressure	-1
** Time/Freq. Dependence* (Select from the ***Time Dependent Fields*** box.)	f:time_dependent_pressure

OK

Select Application Region ...

◆ **FEM**

Select 2D Elements or Edge (Click and mouse drag to select all elements.)	Elm 1:40

Add

OK

Apply

9. Create the time-dependent nodal force.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Force**
Type:	**Nodal**
New Set Name:	force

Input Data ...

Force <F1 F2 F3> | **<0 0 25>**

** Time/Freq. Dependence* | **f:time_dependent_force**
*(Select from the **Time Dependent Fields** box.)*

OK

Select Application Region ...

◆ **FEM**

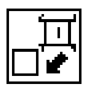
Select Nodes | **Node 11**

Add

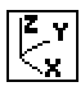
OK

Apply

To simplify the view, turn off the entity labels using the toolbar.

 **Hide Labels**

In addition, switch to a 3 view isometric view point.

 **Iso 3 View**

Action: | **Plot Markers**

Under ***Assigned Load/BC Sets***, highlight:

Displ_spc1.1

Force_force

Press_pressure

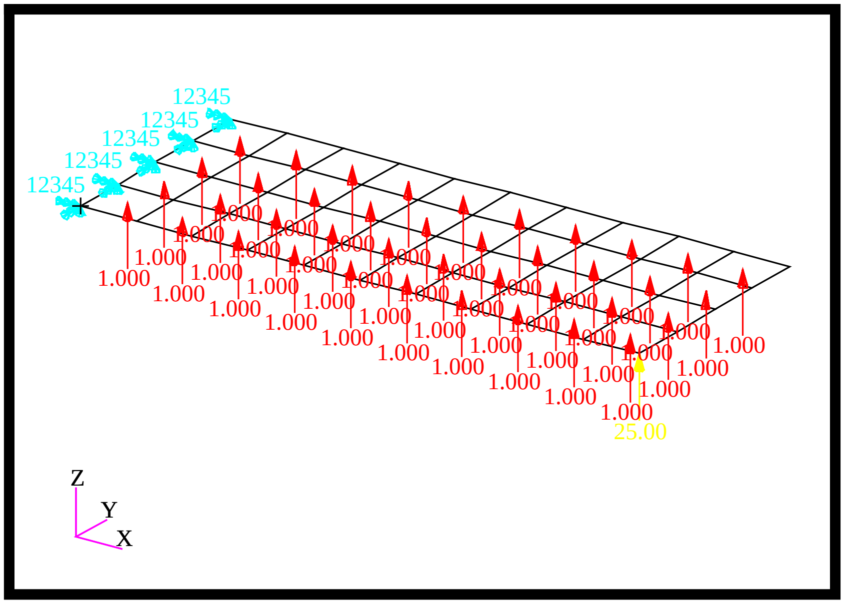
Under ***Select Groups***, highlight:

default_group

Apply

The result should be similar to Figure 4.2.

Figure 4.2



10. Generate the input file.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Analysis Deck**
Jobname:	**prob4**
Translation Parameters...	
Data Output:	**XDB and Print**
OK	
Solution Type...	
Solution Type:	◆ **TRANSIENT RESPONSE**
Formulation	**Modal**
Solution Parameters ...	
Mass Calculation	**Coupled**
Wt.-Mass Conversion	**.00259**
Eigenvalue Extraction...	

Number of Desired Roots	**5**

OK

OK

OK

Subcase Create...

Available Subcases *(Select from menu.)*	**transient_response**

Subcase Parameters...

Time Recovery Points	**DEFINE TIME STEPS ...**
Number of Time Steps =	**100**
Delta - T *(Hit Return to Input Data.)*	**.0004**

OK

Modal Damping	**Crit. Damp. (CRIT)**

DEFINE MODAL DAMPING...

	Frequency	Value
1	0	.03
2	10	.03

OK

OK

Apply

Output Requests...

Form Type:	**Advanced**

under *Output Requests*, highlight:

SPCFORCES(SORT1,Real)=ALL FEM

Delete

Output Requests:	select **DISPLACEMENT(...**
Sorting:	**By Freq/Time**

Modify

OK

Apply

Cancel

Subcase Select ...

Subcases Selected: *(Click to deselect.)*	**default**
Subcases for Solution Sequence: 112 *(Click to select.)*	**transient_response**

OK

Apply

An MSC.Nastran input file called **prob4.bdf** will be generated. This process of translating your model into an input file is called the Forward Translation. The Forward Translation is complete when the Heartbeat turns green. MSC.Patran Users should proceed to step 12.

Generating an input file for MSC.Nastran Users:

MSC.Nastran users can generate an input file using the data previously stated. The result should be similar to the output below.

11. MSC.Nastran input File: **prob4.dat**

```
ID SEMINAR, PROB4
SOL 112
TIME 30
CEND
TITLE = TRANSIENT RESPONSE WITH TIME DEPENDENT PRESSURE AND POINT LOADS
SUBTITLE = USE THE MODAL METHOD
ECHO = UNSORTED
SPC = 1
SET 111 = 11, 33, 55
DISPLACEMENT(SORT2) = 111
SDAMPING = 100
SUBCASE 1
METHOD = 100
DLOAD = 700
LOADSET = 100
TSTEP = 100
$
OUTPUT (XYPLOT)
XGRID=YES
YGRID=YES
XTITLE= TIME (SEC)
YTITLE= DISPLACEMENT RESPONSE AT LOADED CORNER
XYPLOT DISP RESPONSE / 11 (T3)
YTITLE= DISPLACEMENT RESPONSE AT TIP CENTER
XYPLOT DISP RESPONSE / 33 (T3)
YTITLE= DISPLACEMENT RESPONSE AT OPPOSITE CORNER
XYPLOT DISP RESPONSE / 55 (T3)
$
BEGIN BULK
PARAM, COUPMASS, 1
PARAM, WTMASS, 0.00259
$
$ PLATE MODEL DESCRIBED IN NORMAL MODES EXAMPLE PROBLEM
$
INCLUDE 'plate.bdf'
$
$ EIGENVALUE EXTRACTION PARAMETERS
$
```

```
EIGRL, 100, , ,5
$
$ SPECIFY MODAL DAMPING
$
TABDMP1, 100, CRIT,
+, 0., .03, 10., .03, ENDT
$
$ APPLY UNIT PRESSURE LOAD TO PLATE
$
LSEQ, 100, 300, 400
$
PLOAD2, 400, 1., 1, THRU, 40
$
$ VARY PRESSURE LOAD (250 HZ)
$
TLOAD2, 200, 300, , 0, 0., 8.E-3, 250., -90.
$
$ APPLY POINT LOAD (250 HZ)
$
TLOAD2, 500, 600,610, 0, 0.0, 8.E-3, 250., -90.
$
DAREA, 600, 11, 3, 1.
DELAY, 610, 11, 3, 0.004
$
$ COMBINE LOADS
$
DLOAD, 700, 1., 1., 200, 25., 500
$
$ SPECIFY INTERGRATION TIME STEPS
$
TSTEP, 100, 100, 4.0E-4, 1
$
ENDDATA
```

Submitting the input file for analysis:

12. Submit the input file to MSC.Nastran for analysis.

12a. To submit the MSC.Patran **.bdf** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran prob4.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

12b. To submit the MSC.Nastran **.dat** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran prob4 scr=yes**. Monitor the run using the UNIX **ps** command.

13. When the run is completed, use **plotps** utility to create a postscript file, **prob4.ps**, from the binary plot file **prob4.plt**. The displacement response plots for Grids 11, 33 and 55 are shown in figures 4.3, 4.4, and 4.5**.**

14. When the run is completed, edit the **prob4.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing WARNING messages indicate modeling errors.

15. While still editing **prob4.f06**, search for the word:

D I S P L (spaces are necessary)

Displacement at Grid 11.

Time T3

.0064= __________

.0092= __________

.02 = __________

Displacement at Grid 33.

Time T3

.0068= __________

.0092= __________

.02 = __________

Displacement at Grid 55.

Time T3

.0068= ___________

.0092= ___________

.02 = ___________

Comparison of Results

16. Compare the results obtained in the **.f06** file with the following results:

POINT-ID = 11

D I S P L A C E M E N T V E C T O R

TIME	TYPE	T1	T2	T3	R1	R2	R3
.0	G	.0	.0	.0	.0	.0	.0
4.000000E-04	G	3.138503E-15	5.333171E-15	1.873720E-04	-6.340404E-06	5.161942E-05	.0
5.999999E-03	G	9.901832E-13	2.054362E-12	1.177721E-01	3.009433E-03	-3.418436E-02	.0
6.399998E-03	G	1.301660E-12	2.610037E-12	1.495051E-01	6.088505E-03	-4.068905E-02	.0
6.799998E-03	G	1.361787E-12	2.701773E-12	1.469920E-01	1.272909E-02	-3.955902E-02	.0
8.799998E-03	G	-1.399555E-12	-2.778663E-12	-1.534481E-01	-1.183270E-02	4.150834E-02	.0
9.199998E-03	G	-1.565439E-12	-3.143368E-12	-1.692225E-01	-1.568289E-02	4.710494E-02	.0
9.599999E-03	G	-1.396290E-12	-2.830759E-12	-1.533197E-01	-1.109114E-02	4.241555E-02	.0
2.000000E-02	G	1.762308E-13	3.609815E-13	2.043042E-02	1.705799E-04	-5.402198E-03	.0
3.959996E-02	G	5.328810E-14	1.099402E-13	6.485358E-03	-1.067494E-05	-1.817145E-03	.0

POINT-ID = 33

D I S P L A C E M E N T V E C T O R

TIME	TYPE	T1	T2	T3	R1	R2	R3
.0	G	.0	.0	.0	.0	.0	.0
4.000000E-04	G	-1.482405E-15	5.013036E-15	1.835858E-04	-3.682543E-14	5.086755E-05	.0
5.999999E-03	G	-7.504724E-13	1.795108E-12	1.207052E-01	2.766146E-03	-3.436569E-02	.0
6.399998E-03	G	-9.796515E-13	2.269019E-12	1.556774E-01	6.135463E-03	-4.140196E-02	.0
6.799998E-03	G	-1.045427E-12	2.321332E-12	1.599741E-01	1.308315E-02	-4.083382E-02	.0
7.199998E-03	G	-8.759517E-13	1.829195E-12	1.278863E-01	1.804037E-02	-3.306541E-02	.0
8.799998E-03	G	1.073612E-12	-2.391129E-12	-1.655026E-01	-1.212454E-02	4.270243E-02	.0
9.199998E-03	G	1.217997E-12	-2.696051E-12	-1.851369E-01	-1.596580E-02	4.844051E-02	.0
2.000000E-02	G	-1.280910E-13	3.184533E-13	2.059413E-02	1.496853E-04	-5.411018E-03	.0
3.959996E-02	G	-3.946867E-14	9.686225E-14	6.468208E-03	-2.682333E-05	-1.813416E-03	.0

POINT-ID = 55

D I S P L A C E M E N T V E C T O R

TIME	TYPE	T1	T2	T3	R1	R2	R3
.0	G	.0	.0	.0	.0	.0	.0
4.000000E-04	G	-4.974573E-15	5.435887E-15	1.873720E-04	6.340404E-06	5.161942E-05	.0
6.399998E-03	G	-2.151553E-12	2.189475E-12	1.615714E-01	5.634375E-03	-4.218743E-02	.0
6.799998E-03	G	-2.229502E-12	2.212659E-12	1.728057E-01	1.248924E-02	-4.215589E-02	.0
7.199998E-03	G	-1.757434E-12	1.697970E-12	1.456460E-01	1.736620E-02	-3.459810E-02	.0
8.799998E-03	G	2.297826E-12	-2.283533E-12	-1.773652E-01	-1.152392E-02	4.395273E-02	.0
9.199998E-03	G	2.561482E-12	-2.555566E-12	-2.007833E-01	-1.524836E-02	4.984966E-02	.0
9.599999E-03	G	2.290908E-12	-2.331874E-12	-1.754856E-01	-1.067918E-02	4.413952E-02	.0
2.000000E-02	G	-2.917673E-13	3.108817E-13	2.072625E-02	1.192453E-04	-5.428383E-03	.0
3.959996E-02	G	-8.886282E-14	9.431532E-14	6.432103E-03	-4.190110E-05	-1.814187E-03	.0

17. MSC.Nastran Users have finished this exercise. MSC.Patran Users should proceed to the next step.

18. Proceed with the Reverse Translation process, that is attaching the **prob4.xdb** results file into MSC.Patran. To do this, return to the Analysis form and proceed as follows:

◆ **Analysis**

Action:	**Attach XDB**
Object:	**Result Entities**
Method:	**Local**
Select Results File...	
Select Results File	**prob4.xdb**
OK	
Apply	

When the translation is complete bring up the **Results** form.

◆ **Results**

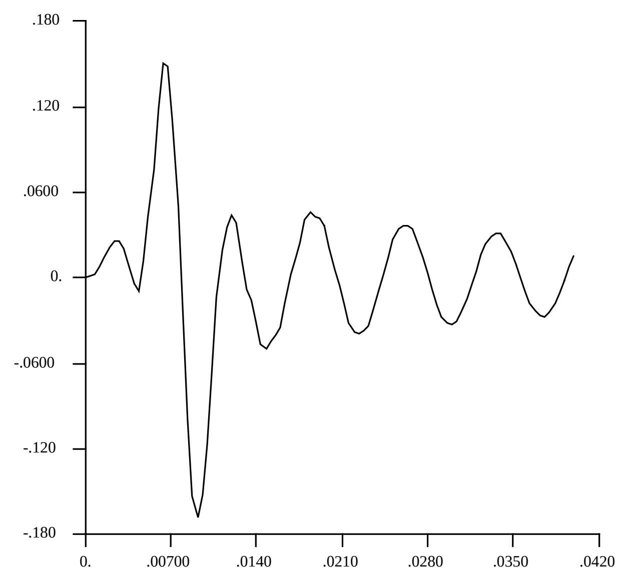
Action:	**Create**
Object:	**Graph**
Select Results Cases	**Transient_response, 0 of 101 subcases**
Filter Method	**All**
Filter	
Apply	
Close	
y:	**Result**
Select y Result:	**Displacement, Translational**
Quanity:	**Z Component**
x:	**Global Variable**
Variable:	**Time**
Target Entities	

Select Nodes: **Node 11**

Apply

The output should look similar to Figure 4.3.

Figure 4.3-<u>Displacement Response at Loaded Corner</u>



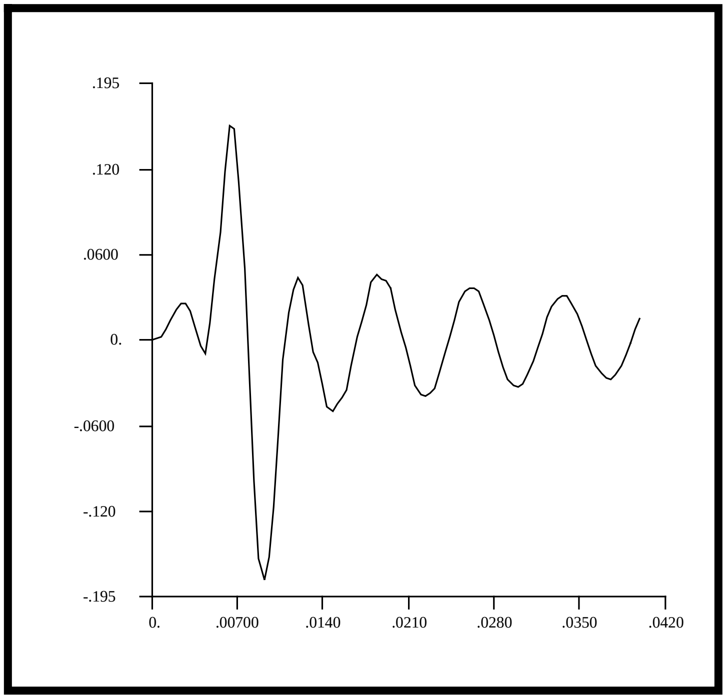
19. Repeat the procedure to find the nodal displacement for Node 33.

Select Nodes: **Node 33**

Apply

The output should look similar to Figure 4.4.

Figure 4.4-Displacement Response at Tip Center



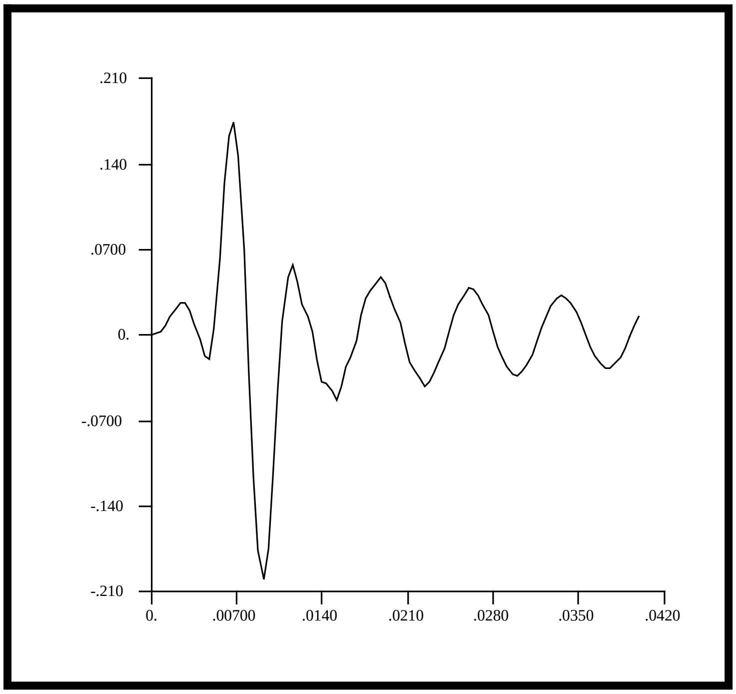
20. Repeat the procedure to find the nodal displacement for Node 55.

Select Nodes: **Node 55**

Apply

The output should look similar to Figure 4.5.

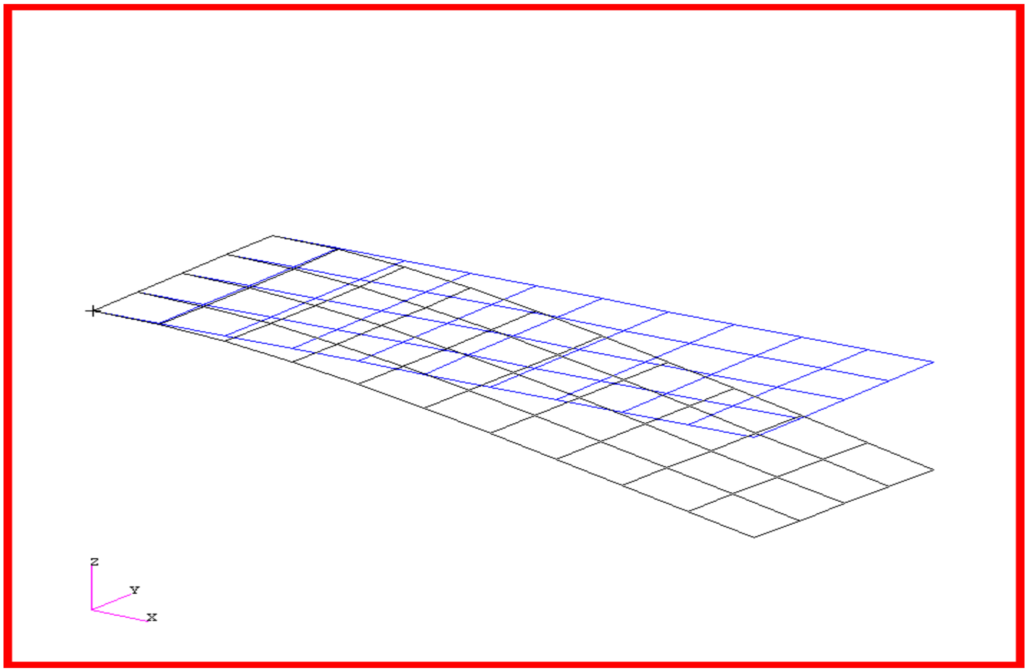
Figure 4.5-Displacement Response at Opposite Corner



Quit MSC.Patran when you are finished with this exercise.

WORKSHOP 5

Direct Frequency Response Analysis



Objectives:

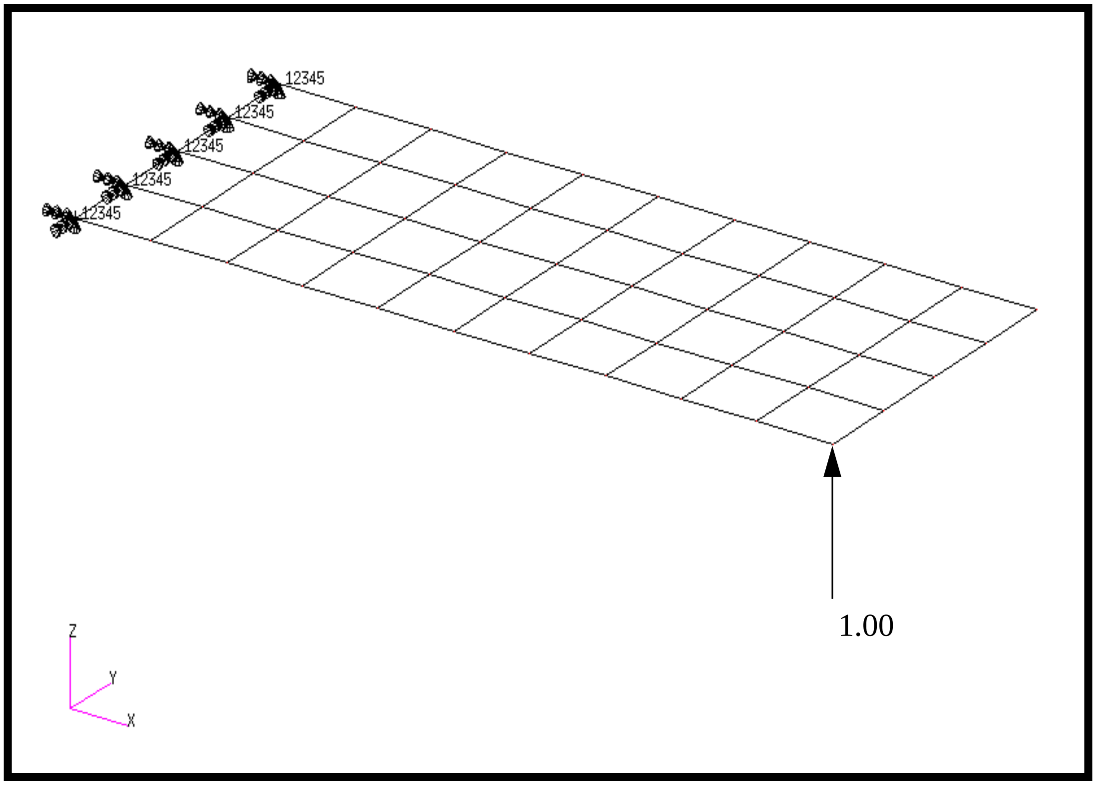
- Define frequency-varying excitation.
- Produce a MSC.Nastran input file from dynamic math model created in Workshop 1.
- Submit the file for analysis in MSC.Nastran.
- Compute nodal displacements for desired frequency domain.

Model Description:

Using the direct method, determine the frequency response of the flat rectangular plate, created in Workshop 1, under frequency-varying excitation. This example structure shall be excited by a unit load at a corner of the tip. Use a frequency step of 20 Hz between a range of 20 and 1000 Hz. Use structural damping of g=0.06.

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

Figure 5.1-Loads and Boundary Conditions



Suggested Exercise Steps:

- Reference previously created dynamic math model, **plate.bdf**, by using the INCLUDE statement
- Define the frequency-varying tip load (DAREA and RLOAD2).
- Define a set of frequencies to be used in the solution (FREQ1).
- Prepare the model for a direct frequency response analysis (SOL 108).
- Specify the structural damping.
  - PARAM, G, 0.06
- Request response in terms of nodal displacement at Grids 11, 33 and 55.
- Generate an input file and submit it to the MSC.Nastran solver for direct transient analysis.
- Review the results, specifically the nodal displacements and phase angles.

```
ID SEMINAR,PROB5
```

```
CEND
```

```
BEGIN BULK
```

1	2	3	4	5	6	7	8	9	10

1	2	3	4	5	6	7	8	9	10

ENDDATA

Exercise Procedure:

1. Users who are not utilizing MSC.Patran for generating an input file should go to Step 9, otherwise, proceed to step 2.

2. Create a new database called **prob5.db**.

File/New

New Database Name: prob5

OK

In the *New Model Preferences* form, set the following:

Tolerance: ◆ **Default**

Analysis Code: **MSC/NASTRAN**

Analysis Type: **Structural**

OK

3. Create the model by importing an existing MSC.Nastran input file, **(plate.bdf)**.

◆ **Analysis**

Action: **Read Input file**

Object: **Model Data**

Method: **Translate**

Select Input File ...

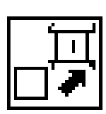
plate.bdf

OK

Apply

OK

4. Activate the entity labels by selecting the **Show Labels** icon on the toolbar.



Show Labels

5. Create a frequency dependent load case for the frequency response.

◆ **Load Cases**

Action:	**Create**
Load Case Name:	**frequency_response**
Load Case Type:	**Time Dependent**
Assign/Prioritize Loads/BCs *(Highlight the following:)*	**Displ_spc1.1**
OK	
Apply	

6. Create a frequency dependent field for the frequency dependent load.

◆ **Fields**

Action:	**Create**
Object:	**Non Spatial**
Method:	**Tabular Input**
Field Name	**frequency_dependent_load**
Active Independent Variables	■ **Frequency (f)**
[Options ...]	
Maximum Number of f:	**2**
OK	
Input Data...	

Using the data in the table below, enter the values describing the frequency dependent force into the ***Time/Frequency Scalar Table Data*** form.

	Freq (f)	Value
1	0	1.0

2	1000	1.0

OK

Apply

7. Create the frequency dependent unit force.

◆ **Load/BCs**

Action:	**Create**
Object:	**Force**
Type:	**Nodal**
New Set Name:	unit_force

Input Data...

Spatial Dependence/Force:	<0 0 1>
* *Time/Freq. Dependence:* (*Select from the* ***Time Dependent Fields*** *box*)	f:frequency_dependent_load

OK

Select Application Region...

■ **FEM**

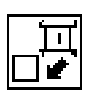
Select Nodes:	Node 11

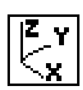
Add

OK

Apply

To better visualize the model, hide the entity labels and switch to an isometric view using the icons below:


Hide Labels


Iso 3 View

Action:	**Plot Markers**

Under ***Assigned Load/BC Sets***, highlight:

Displ_spc1.1

Force_unit_force

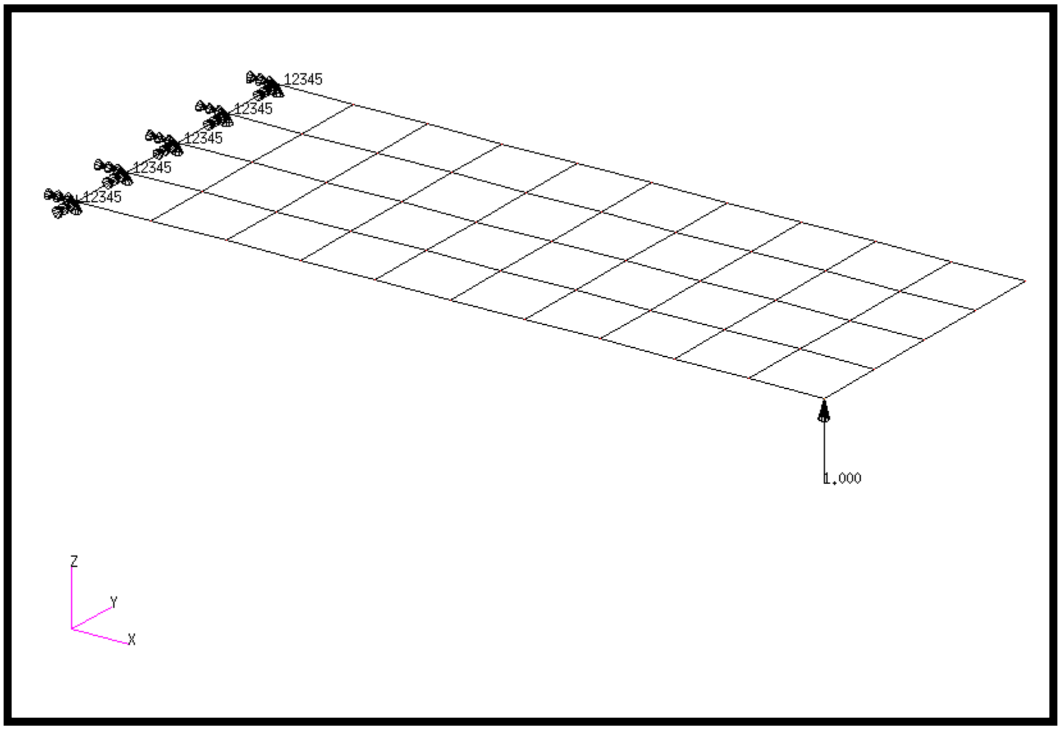
Under ***Select Groups***, highlight:

default_group

Apply

The model should be similar to Figure 5.2.

Figure 5.2



8. Now you are ready to generate an input file for analysis.

   Click on the **Analysis** radio button on the Top Menu Bar and complete the entries as shown here.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Analysis Deck**

Job Name	**prob5**
Translation Parameters...	
Data Output:	**XDB and Print**
OK	
Solution Type...	
Solution Type:	**◆ FREQUENCY RESPONSE**
Formulation:	**Direct**
Solution Parameters...	
Mass Calculation:	**Coupled**
Wt.-Mass Conversion=	**0.00259**
Struct. Damping Coeff. =	**0.06**
OK	
OK	
Subcase Create ...	
Available Subcases	**frequency_response**
Subcase Parameters...	
DEFINE FREQUENCIES...	
Starting Frequency = *(Hit Return to Input Data.)*	**20**
Ending Frequency = *(Hit Return to Input Data.)*	**1000**
# of Freq. Increments = *(Hit Return to Input Data.)*	**49**
OK	
OK	
Output Requests...	
Form Type:	**Advanced**

under *Output Request* highlight: **SPCFORCES(SORT1,Real)=All FEM**

Delete

Output Requests:	select **DISPLACEMENT...**
Sorting:	**By Freq/Time**

Modify

OK

Apply

Cancel

Subcase Select ...

Subcases Selected: *(Click to de-select.)*	**Default**
Subcases for Solution Sequence: 108 *(Click to select.)*	**frequency_response**

OK

Apply

An input file called **prob5.bdf** will be generated. This process of translating your model into an input file is called the Forward Translation. The Forward Translation is complete when the Heartbeat turns green. MSC.Patran users should now proceed to Step 10.

Generating an input file for MSC.Nastran Users:

MSC.Nastran users can generate an input file using the data from page 5-3 (Model Description). The result should be similar to the output below.

9. MSC.Nastran input file: **prob5.dat**

```
ID SEMINAR, PROB5
SOL 108
TIME 30
CEND
TITLE = FREQUENCY RESPONSE DUE TO UNIT FORCE AT TIP
ECHO = UNSORTED
SPC = 1
SET 111 = 11, 33, 55
DISPLACEMENT(SORT2, PHASE) = 111
SUBCASE 1
DLOAD = 500
FREQUENCY = 100
$
OUTPUT (XYPLOT)
$
XTGRID= YES
YTGRID= YES
XBGRID= YES
YBGRID= YES
YTLOG= YES
YBLOG= NO
XTITLE= FREQUENCY (HZ)
YTTITLE= DISPLACEMENT RESPONSE AT LOADED CORNER, MAGNITUDE
YBTITLE= DISPLACEMENT RESPONSE AT LOADED CORNER, PHASE
XYPLOT DISP RESPONSE / 11 (T3RM, T3IP)
YTTITLE= DISPLACEMENT RESPONSE AT TIP CENTER, MAGNITUDE
YBTITLE= DISPLACEMENT RESPONSE AT TIP CENTER, PHASE
XYPLOT DISP RESPONSE / 33 (T3RM, T3IP)
YTTITLE= DISPLACEMENT RESPONSE AT OPPOSITE CORNER, MAGNITUDE
YBTITLE= DISPLACEMENT RESPONSE AT OPPOSITE CORNER, PHASE
XYPLOT DISP RESPONSE / 55 (T3RM, T3IP)
$
BEGIN BULK
PARAM, COUPMASS, 1
PARAM, WTMASS, 0.00259
$
$ PLATE MODEL DESCRIBED IN NORMAL MODES EXAMPLE
$
INCLUDE ’plate.bdf’
```

```
$
$ SPECIFY STRUCTURAL DAMPING
$
PARAM, G, 0.06
$
$ APPLY UNIT FORCE AT TIP POINT
$
RLOAD2, 500, 600, , ,310
$
DAREA, 600, 11, 3, 1.0
$
TABLED1, 310,
, 0., 1., 1000., 1., ENDT
$
$ SPECIFY FREQUENCY STEPS
$
FREQ1, 100, 20., 20., 49
$
ENDDATA
```

Submitting the input file for analysis:

10. Submit the input file to MSC.Nastran for analysis.

    10a. To submit the MSC.Patran **.bdf** file, find an available UNIX shell window. At the command prompt enter **nastran prob5.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

    10b. To submit the MSC.Nastran **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran prob5 scr=yes**. Monitor the run using the UNIX **ps** command.

11. When the run is completed, use **plotps** utility to create a postscript file, **prob5.ps**, from the binary plot file, **prob5.plt**. The displacement response plots for Grids 11, 33 and 55 are shown in figures 5-2 to 5-7.

12. When the run is completed, edit the **prob5.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing WARNING messages indicate modeling errors.

For MSC.Nastran users only. MSC.Patran users should skip to step 16.

13. While still editing **prob5.f06**, search for the word:

    X Y - O U T P U T S U M M A R Y (spaces are necessary).

    Displacement at Grid 11

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

    Displacement at Grid 33

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

    Displacement at Grid 55

Frequency (X)		Displacement (Y)
140	=	__________
1000	=	__________

14. Compare the results obtained in the **.f06** file with the following results:

POINT-ID = 11

C O M P L E X   D I S P L A C E M E N T   V E C T O R
(MAGNITUDE/PHASE)

FREQUENCY	TYPE	T1	T2	T3	R1	R2	R3
2.000000E+01	G	0.0	0.0	8.817999E-03	6.435859E-04	2.632016E-03	0.0
		0.0	0.0	356.4954	176.5664	176.5000	0.0
4.000000E+01	G	0.0	0.0	9.404316E-03	6.434991E-04	2.795561E-03	0.0
		0.0	0.0	356.2596	176.5677	176.2785	0.0
9.799999E+02	G	0.0	0.0	9.965085E-04	2.691742E-04	4.097779E-04	0.0
		0.0	0.0	187.6832	7.8008	15.1581	0.0
1.000000E+03	G	0.0	0.0	8.803170E-04	2.354656E-04	3.317750E-04	0.0
		0.0	0.0	186.9299	8.2146	14.6645	0.0

POINT-ID = 33

C O M P L E X   D I S P L A C E M E N T   V E C T O R
(MAGNITUDE/PHASE)

FREQUENCY	TYPE	T1	T2	T3	R1	R2	R3
2.000000E+01	G	0.0	0.0	8.183126E-03	5.993295E-04	2.443290E-03	0.0
		0.0	0.0	356.4899	176.5639	176.4950	0.0
4.000000E+01	G	0.0	0.0	8.768992E-03	6.006200E-04	2.606561E-03	0.0
		0.0	0.0	356.2376	176.5565	176.2581	0.0
9.799999E+02	G	0.0	0.0	6.867234E-04	3.836353E-04	5.393046E-04	0.0
		0.0	0.0	188.0180	5.5597	10.0794	0.0
1.000000E+03	G	0.0	0.0	6.062436E-04	3.454144E-04	4.648783E-04	0.0
		0.0	0.0	186.8358	5.4959	8.8514	0.0

POINT-ID = 55

C O M P L E X   D I S P L A C E M E N T   V E C T O R
(MAGNITUDE/PHASE)

FREQUENCY	TYPE	T1	T2	T3	R1	R2	R3
2.000000E+01	G	0.0	0.0	7.606255E-03	5.587703E-04	2.371172E-03	0.0
		0.0	0.0	356.4844	176.5612	176.4928	0.0
4.000000E+01	G	0.0	0.0	8.190030E-03	5.613805E-04	2.534562E-03	0.0
		0.0	0.0	356.2155	176.5442	176.2492	0.0
9.799999E+02	G	0.0	0.0	2.558788E-04	4.612964E-04	5.702980E-04	0.0
		0.0	0.0	193.1958	4.6290	9.0143	0.0
1.000000E+03	G	0.0	0.0	2.144666E-04	4.204372E-04	4.981144E-04	0.0
		0.0	0.0	190.6200	4.3746	7.6762	0.0

15. MSC.NastranMSC.Patran Users have finished this exercise. MSC.Patran Users should proceed to the next step.

16. Proceed with the Reverse Translation process, that is attaching the **prob5.xdb** results file into MSC.Patran. To do this, return to the Analysis form and proceed as follows.

◆ **Analysis**

Action:	**Attach XDB**
Object:	**Result Entities**
Method:	**Local**
Select Results File...	
Select Available Files	**prob5.xdb**
OK	
Apply	

17. Plot the results in XY plots.

The first plot is to make the Displacement versus Frequency plot at Node 11.

◆ **Results**

Action:	**Create**
Object:	**Graph**
Method:	**y vs x**
Select Results Cases	**Frequency_response, 0 of 50 subcases**
Filter Method	**All**
Filter	
Apply	
Close	
Select Y result:	**Displacement, Translational**
Quanity:	**Z Component**

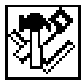
x: **Global Variable**

Variable: **Frequency**

Target Entities

Select Nodes: **Node 11**

Select the Plot Options by clicking on the icon:



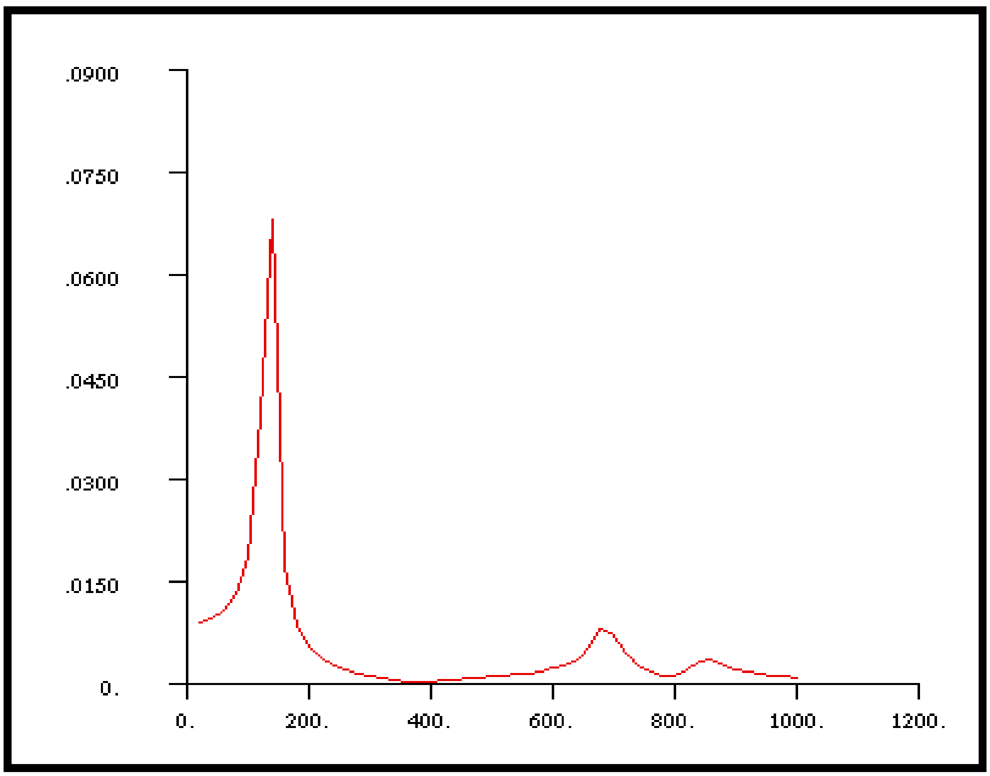
Plot Options

Plot Options

Complex No as: **Magnitude**

Apply

Figure 5.3-Displacement Response at Node 11



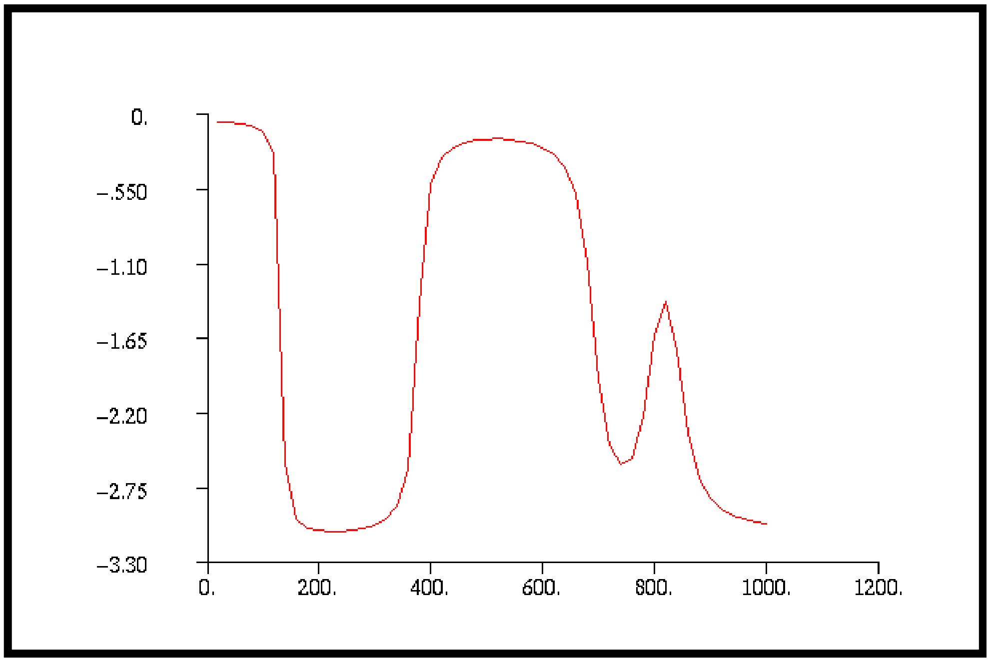
The next step is to make the plot of Phase versus Frequency.

Plot Options

Complex No as: **Phase:**

Apply

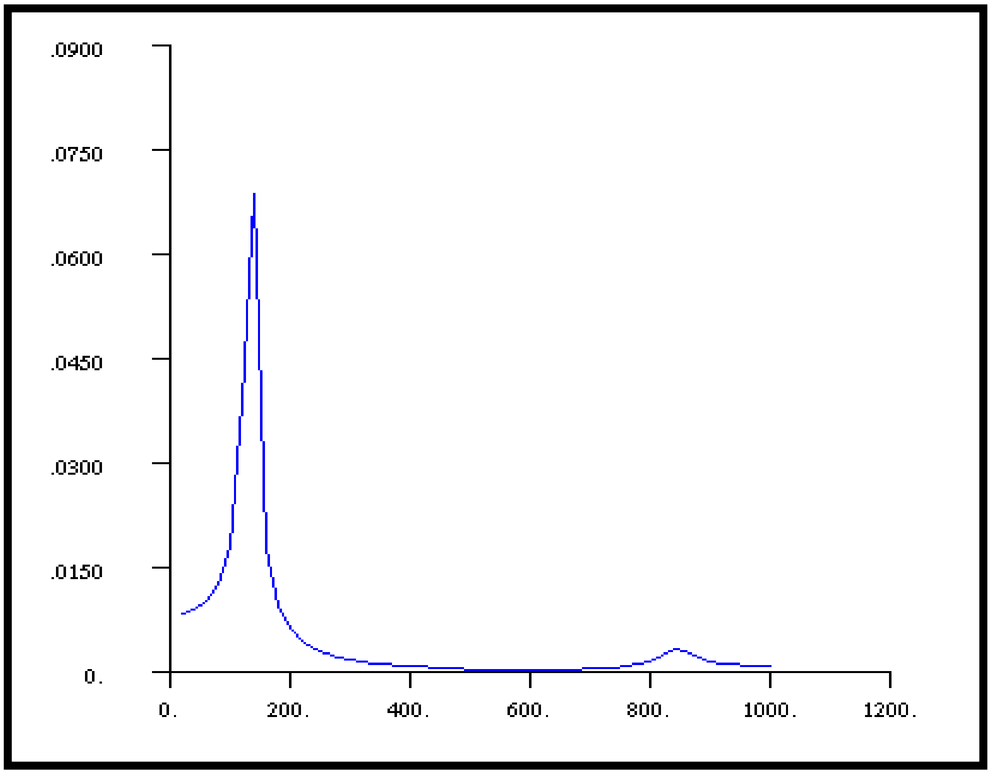
Figure 5.4-Phase Angle at Node 11



Repeat the above steps of plotting the XY plots of Node 11 for Node 33 and 55. Once again, push **Cancel** to remove any miscellaneous forms until the ***Results Display*** form.

Plot Options	
Complex No as:	**Magnitude**
Target Entities:	
Select Nodes:	**Node 33**
Apply	

Figure 5.5-Displacement Response at Node 33

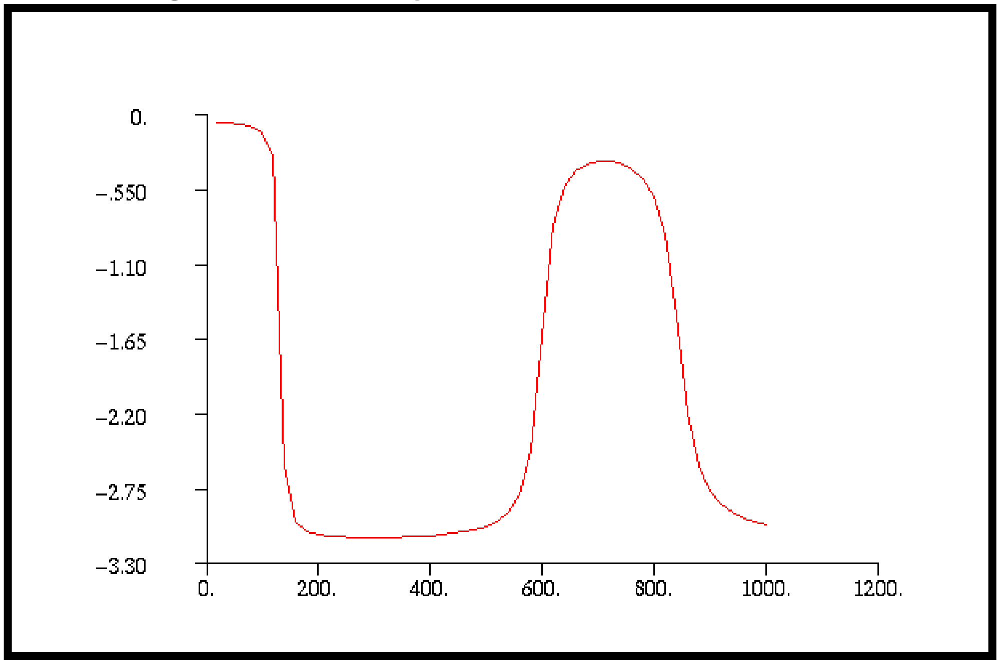


Plot Options

Complex No as: **Phase**

Apply

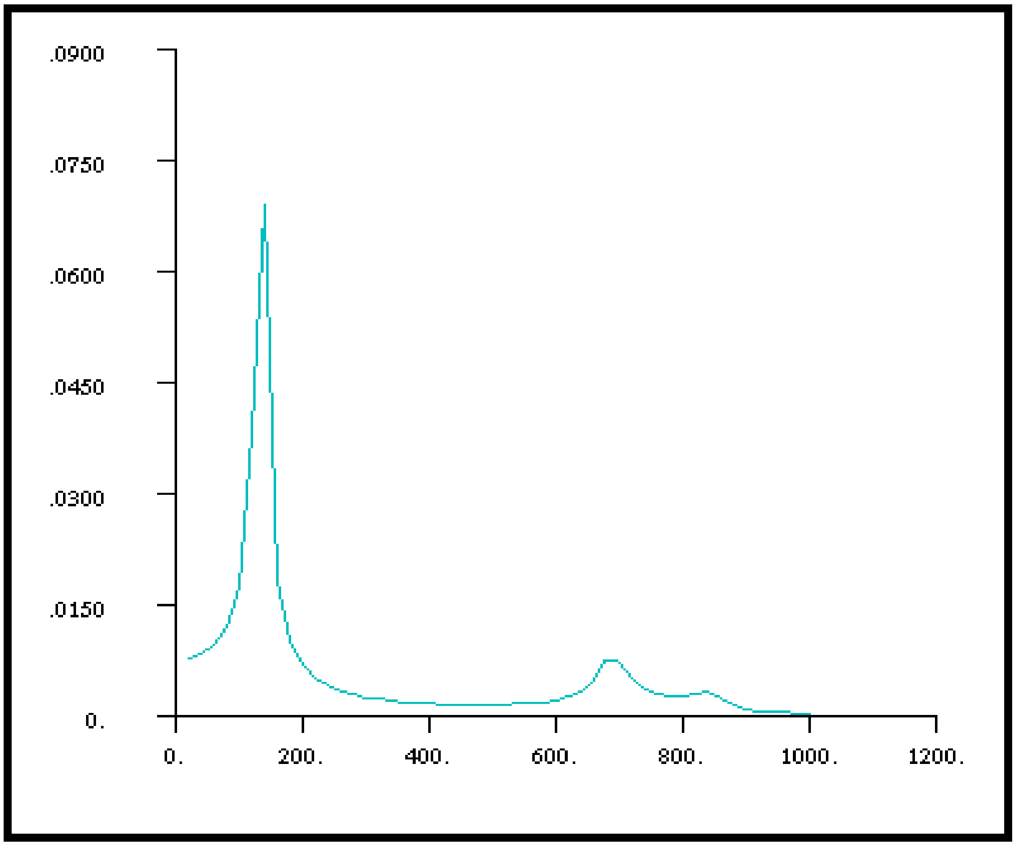
Figure 5.6-Phase Angle at Node 33



Plot Options

Complex No as	**Phase**
Target Entities:	
Select Nodes:	**Node 55**
Apply	

Figure 5.7-Displacement Response at Node 55

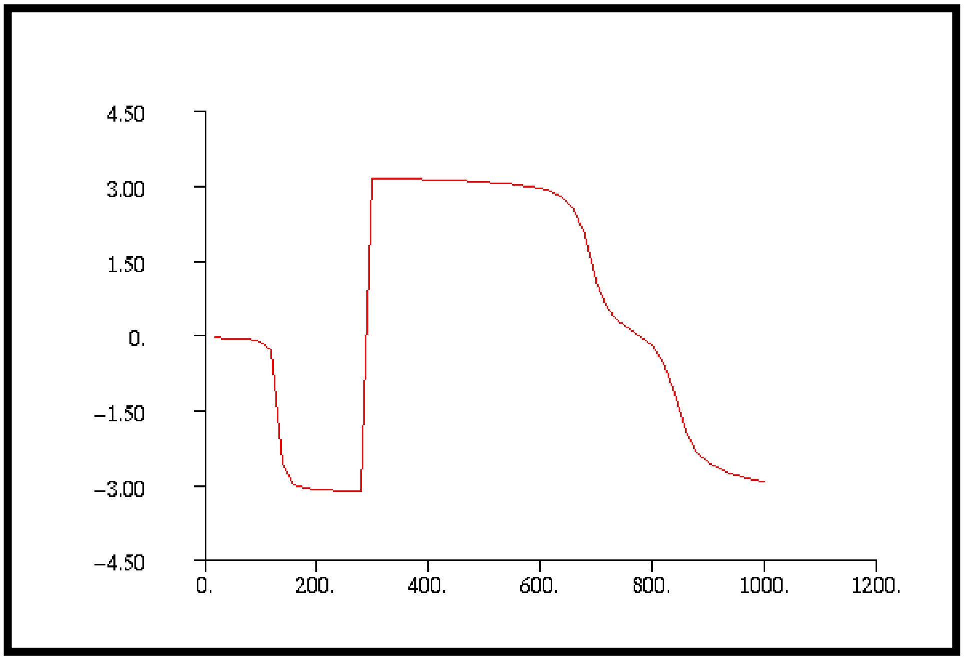


Plot Options

Complex No as: **Phase**

Apply

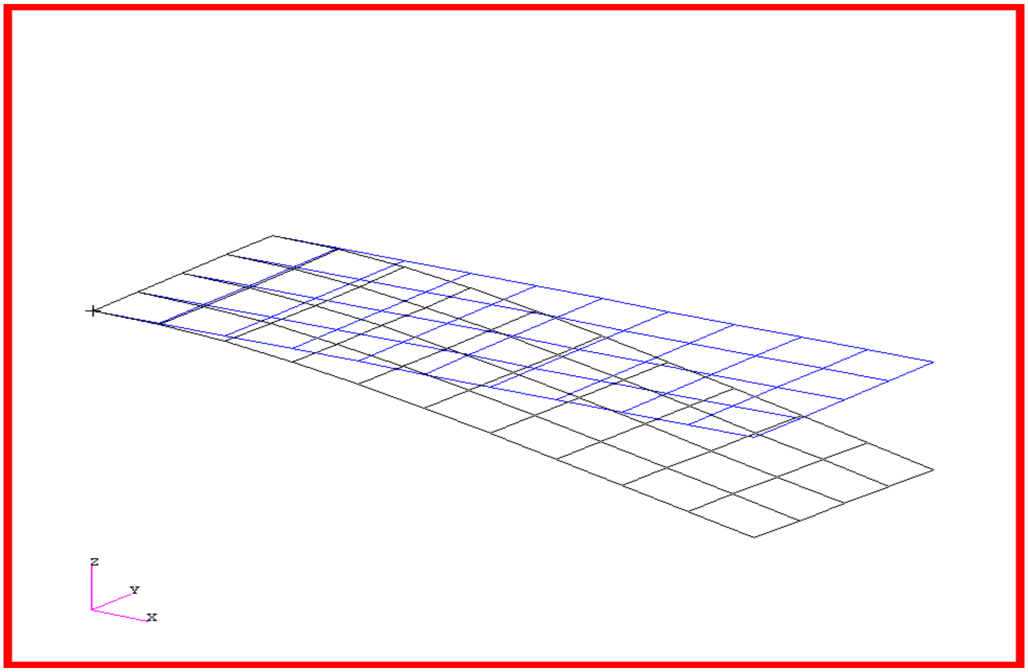
Figure 5.8-Phase Angle at Node 55



Quit MSC.Patran when you have completed this exercise.

WORKSHOP 6

Modal Frequency Response Analysis



Objectives:

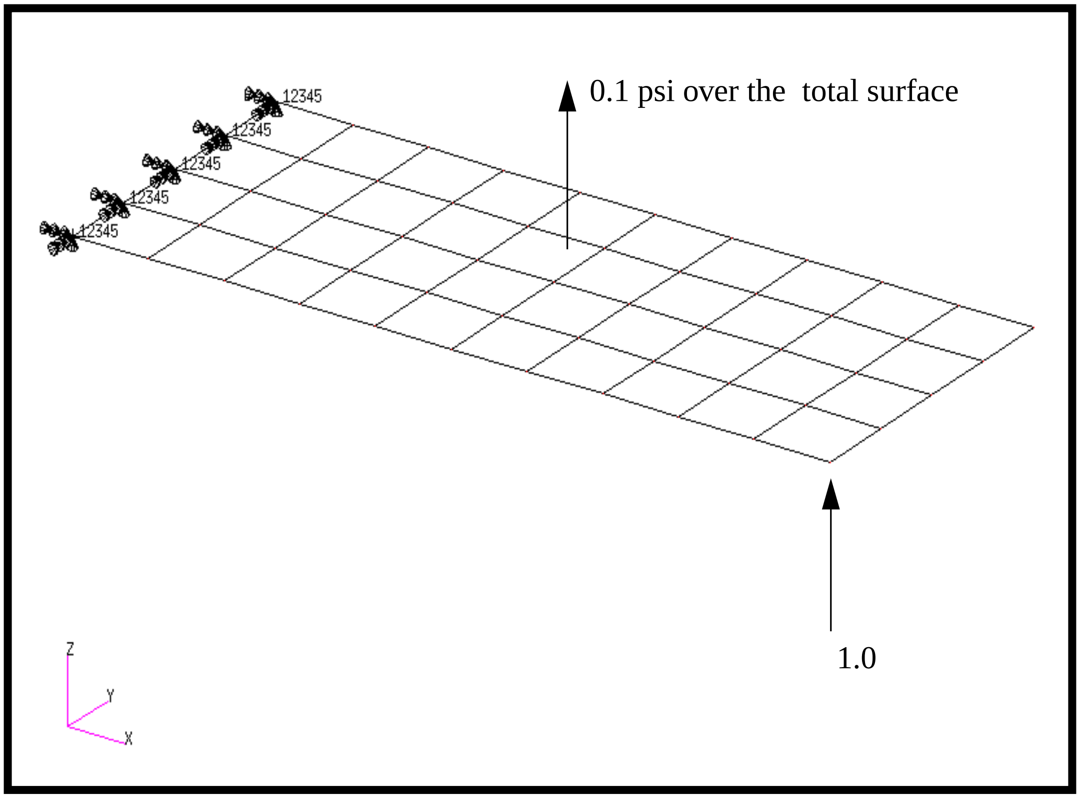
- Define a frequency-varying excitation.
- Produce a MSC.Nastran input file from a dynamic math model created in Workshop 1.
- Submit the file for analysis in MSC.Nastran.
- Compute nodal displacements for desired frequency domain.

Model Description:

Using the modal method, determine the frequency response of the flat rectangular plate, created in Workshop 1, excited by a 0.1 psi pressure load over the total surface of the plate and a 1.0 lb. force at a corner of the tip lagging 45^{o}. Use a modal damping of $\xi = 0.03$. Use a frequency step of 20 hz between a range of 20 and 1000 hz; in addition, specify five evenly spaced excitation frequencies between the half power points of each resonant frequency between the range of 20-1000 hz.

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

Figure 6.1-Loads and Boundary Conditions



Suggested Exercise Steps:

- Reference a previously created dynamic math model, **plate.bdf**, by using the INCLUDE statement.
- Specify modal damping as a tabular function of natural frequency (TABDMP1).
- Define the frequency-varying pressure loading (PLOAD2, LSEQ and RLOAD2).
- Define the frequency-varying tip load (DAREA and RLOAD2).
- Define a set of frequencies to be used in the solution (FREQ1, FREQ4).
- Prepare the model for a modal frequency response analysis (SOL 111).
- Define the dynamic load phase lead modal frequency response (DPHASE).
- Request response in terms of nodal displacement at Grids 11, 33, and 55.
- Generate an input file and submit it to the MSC.Nastran solver for direct transient analysis.
- Review the results, specifically the nodal displacements and phase angles.

```
ID SEMINAR,PROB6
```

```
CEND
```

```
BEGIN BULK
```

1	2	3	4	5	6	7	8	9	10

1	2	3	4	5	6	7	8	9	10

ENDDATA

Exercise Procedure:

1. Users who are not utilizing MSC.Patran for generating an input file should go to Step 10, otherwise, proceed to step 2.

2. Create a new database called **prob6.db**.

File/New

New Database Name **prob6**

OK

In the *New Model Preferences* form, set the following:

Tolerance ◆ **Default**

Analysis Code: **MSC/NASTRAN**

Analysis Type: **Structural**

OK

3. Create the model by importing an existing MSC.Nastran input file, **(plate.bdf)**.

◆ **Analysis**

Action: **Read Input file**

Object: **Model Data**

Method **Translate**

Select Input File...

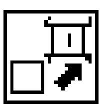
Select File **plate.bdf**

OK

Apply

OK

4. Activate the entity labels by selecting the Show Labels icon on the toolbar.



Show Labels

5. Create a frequency dependent load case for the frequency response.

◆ **Load Cases**

Action:	**Create**
Load Case Name:	**frequency_Response**
Load Case Type:	**Time Dependent**
Assign/Prioritize Loads/BCs *(Highlight the following:)*	**Displ_spc1.1**

OK

Apply

6. Create the frequency dependent field for the frequency response.

◆ **Fields**

Action:	**Create**
Object:	**Non Spatial**
Method	**Tabular Input**
Field Name	**frequency_dependent_load**
Active Independent Variables	■ **Frequency (f)**

[Options...]

Maximum Number of f	**2**

OK

Input Data...

Enter the ***Time/Frequency Scalar Table Data*** form with the data below.

	Freq (f)	*Value*
1	**10.**	**1.0**
2	**1000.**	**1.0**

OK

Apply

7. Create the frequency dependent unit force.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Pressure**
Type:	**Element Uniform**
New Set Name	**pressure**
Target Element Type:	**2D**

Input Data...

Top Surf Pressure	**-0.1**
** Time/Freq. Dependence* (Select from the **Time Dependent Fields** box.)	**f:frequency_dependent_load**

OK

Select Application Region...

■ **FEM**

Select 2D Elements or Edge:	**Elem 1:40**

Add

OK

Apply

8. Create forces.

◆ **Load/BCs**

Action:	**Create**
Object:	**Force**
Type:	**Nodal**
New Set Name	**force**

Input Data...

Force <F1 F2 F3>	**<0, 0 , 1>**
** Time/Freq. Dependence:* (Select from the **Time Dependent Fields** box.)	**f:frequency_dependent_load**

OK

Select Application Region...

■ **FEM**

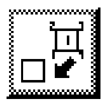
Select Nodes **Node 11**

Add

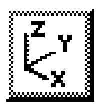
OK

Apply

To better visualize the model, hide the labels using the following toolbar icon:



Hide Labels



Iso 3 View

Action: **Plot Markers**

Under ***Assigned Load/BC Sets***, highlight:

Displ_spc1.1

Press_pressure

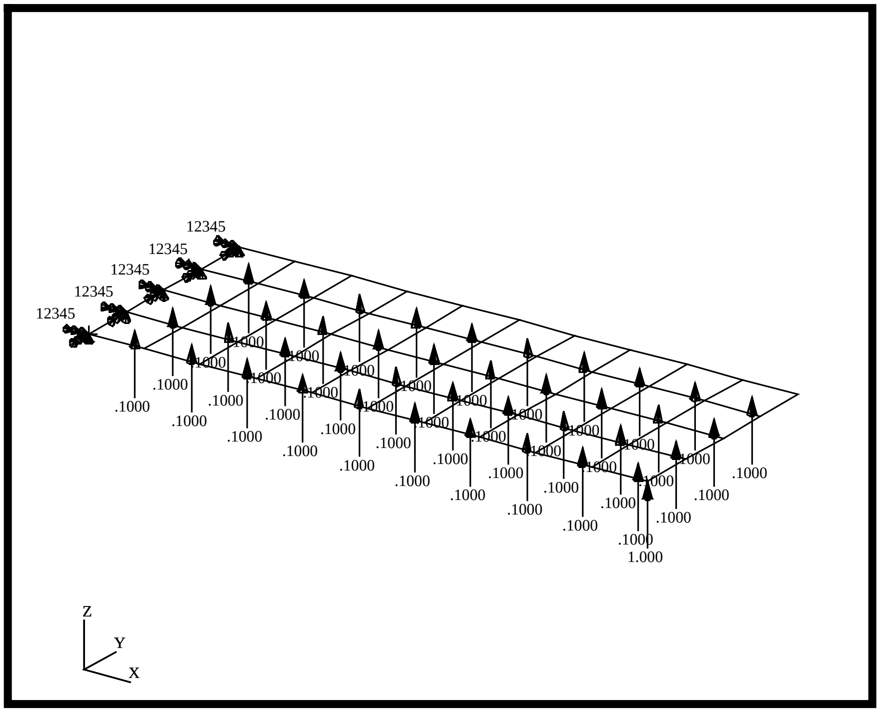
Force_force

Under ***Select Groups***, highlight:

default_group

The model should be similar to Figure 6.2.

Figure 6.2



In order to make the next step easier, turn the entity labels back on.

Show Labels

9. Now you are ready to generate an input file for analysis

   Click on the **Analysis** radio button on the Top Menu Bar and complete the entries as shown here.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method	**Analysis Deck**
Job Name	**prob6**
Translation Parameters...	
Data Output:	**XDB and Print**
OK	
Solution Type...	

Solution Type: ◆ **FREQUENCY RESPONSE**

Formulation: **Modal**

Solution Parameters...

Mass Calculation: **Coupled**

Wt.-Mass Conversion = **0.00259**

Eigenvalue Extraction...

Frequency Range of Interest:

Lower = **10.**

Upper = **2000.**

OK

OK

OK

Subcase Create...

Available Subcases **frequency_dependent**

Subcase Parameters...

DEFINE FREQUENCIES...

Add Row

	Incr. Type	Start Freq.	End Freq.	No. Incr.	Cluster/Spread
1	Linear	20	1000	49	Not Used
2	Linear	20	1000	5	Not Used

OK

Modal Damping **Crit. Damp. (CRIT)**

DEFINE MODAL DAMPING

Add Row

	Frequency	Value
1	0	.03
2	10	.03

OK

OK

Output Requests...

Under ***Output Requests***, highlight:

SPCFORCES(SORT1,Real)=All FEM

Delete

OK

Apply

Cancel

Subcase Select...

Subcases Selected: *(Click to de-select.)* — **Default**

Subcases for Solution Sequence: 111 *(Click to select.)* — **frequency_dependent**

OK

Apply

NOTE: Not so Fast!! Before submitting the job, we still need to define the phase lead.

An input file called **prob6.bdf** will be generated. This process of translating your model into an input file is called the Forward Translation. The Forward Translation is complete when the Heartbeat turns green.

10. However, since the phase lead term in the equation of the dynamic loading function (DPHASE) is currently not supported by PATRAN, you will need to manually edit the file, **prob6.bdf**, to insert the appropriate phase for the point load.

Search for:

```
RLOAD1    8       9                               1
```

Insert the identification number of the DPHASE entry in the 5th field. The revised RLOAD1 card should look as follows;

```
RLOAD1   8       9               92      1
```

Also, insert the necessary DPHASE card;

```
DPHASE   92      11      3       -45.
```

(NOTE: The placement of the numbers must fit the within the alloted 8 character “cell” widths)

MSC.Patran users should now proceed to Step 12.

Generating an input file for MSC.Nastran Users:

MSC.Nastran users can generate an input file using the data from pages 6-3 (general model description). The result should be similar to the output below.

11. MSC.Nastran input file: **prob6.dat**.

```
ID SEMINAR, PROB6
SOL 111
TIME 30
CEND
TITLE = FREQUENCY RESPONSE WITH PRESSURE AND POINT LOADS
SUBTITLE = USING THE MODAL METHOD WITH LANCZOS
ECHO = UNSORTED
SEALL = ALL
SPC = 1
SET 111 = 11, 33, 55
DISPLACEMENT(PHASE, PLOT) = 111
METHOD = 100
FREQUENCY = 100
SDAMPING = 100
SUBCASE 1
DLOAD = 100
LOADSET = 100
$
OUTPUT (XYPLOT)
$
XTGRID= YES
YTGRID= YES
XBGRID= YES
YBGRID= YES
YTLOG= YES
YBLOG= NO
XTITLE= FREQUENCY (HZ)
YTTITLE= DISPLACEMENT RESPONSE AT LOADED CORNER, MAGNITUDE
YBTITLE= DISPLACEMENT RESPONSE AT LOADED CORNER, PHASE
XYPLOT DISP RESPONSE / 11 (T3RM, T3IP)
YTTITLE= DISPLACEMENT RESPONSE AT TIP CENTER, MAGNITUDE
YBTITLE= DISPLACEMENT RESPONSE AT TIP CENTER, PHASE
XYPLOT DISP RESPONSE / 33 (T3RM, T3IP)
YTTITLE= DISPLACEMENT RESPONSE AT OPPOSITE CORNER, MAGNITUDE
YBTITLE= DISPLACEMENT RESPONSE AT OPPOSITE CORNER, PHASE
```

```
XYPLOT DISP RESPONSE / 55 (T3RM, T3IP)
$
BEGIN BULK
$
$ PARAMETERS FOR POST-PROCESSING
PARAM,COUPMASS,1
PARAM,WTMASS,0.00259
$
$ PLATE MODEL DESCRIBED IN NORMAL MODES EXAMPLE
$
INCLUDE 'plate.bdf'
$
$ EIGENVALUE EXTRACTION PARAMETERS
$
EIGRL, 100, 10., 2000.
$
$ SPECIFY MODAL DAMPING
$
TABDMP1, 100, CRIT,
+, 0., .03, 10., .03, ENDT
$
$ APPLY UNIT PRESSURE LOAD TO PLATE
$
LSEQ, 100, 300, 400
$
PLOAD2, 400, 1., 1, THRU, 40
$
$ APPLY PRESSURE LOAD
$
RLOAD2, 400, 300, , ,310
$
TABLED1, 310,
, 10., 1., 1000., 1., ENDT
$
$ POINT LOAD
$
$ IF 'DAREA' CARDS ARE REFERENCED, THEN
$ 'DPHASE' AND 'DELAY' CAN BE USED
$
RLOAD2, 500, 600, , 320, 310
$
DPHASE, 320, 11, 3, -45.
$
```

```
$
DAREA, 600, 11, 3, 1.0
$
$ COMBINE LOADS
$
DLOAD, 100, 1., .1, 400, 1.0, 500
$
$ SPECIFY FREQUENCY STEPS
$
FREQ1, 100, 20., 20., 49
FREQ4, 100, 20., 1000., .03, 5
$
ENDDATA
```

Submitting the input file for analysis:

12. Submit the input file to MSC.Nastran for analysis.

12a. To submit the MSC.Patran **.bdf** file, find an available UNIX shell window. At the command prompt enter **nastran prob6.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

12b. To submit the MSC.Nastran **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran prob6 scr=yes**. Monitor the run using the UNIX **ps** command.

13. When the run is completed, use **plotps** utility to create a postscript file, **prob6.ps**, from the binary plot file **prob6.plt**. The displacement response plots for Grids 11, 33 and 55 are shown in figures 6.2 to 6.4.

14. When the run is completed, edit the **prob6.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing WARNING messages indicate modeling errors.

For MSC.Nastran users only. MSC.Patran users should skip to step 16.

15. While still editing **prob6.f06**, search for the word:

X Y - O U T P U T   S U M M A R Y (spaces are necessary).

Displacement at Grid 11

Frequency (X)		Displacement (Y)
140	=	__________
440	=	__________

Displacement at Grid 33

Frequency (X)		Displacement (Y)
140	=	__________
660	=	__________

Displacement at Grid 55

Frequency (X)		Displacement (Y)
140	=	__________
1000	=	__________

Comparison of Results

16. Compare the results obtained in the .f06 file with the following results:

```
                                        R E A L   E I G E N V A L U E S
                          X Y - O U T P U T   S U M M A R Y  ( R E S P O N S E )
SUBCASE  CURVE FRAME                   XMIN-FRAME/  XMAX-FRAME/  YMIN-FRAME/    X FOR      YMAX-FRAME/    X FOR
  ID      TYPE  NO. CURVE ID.             ALL DATA     ALL DATA     ALL DATA     YMIN          ALL DATA     YMAX
   1  DISP      1     11(  5,--)  2.000000E+01 1.000000E+03 3.481836E-04 4.400000E+02 1.699121E-01 1.336996E+0
                                  2.000000E+01 1.000000E+03 3.481836E-04 4.400000E+02 1.699121E-01 1.336996E+0
   1  DISP      1     11(--, 11)  2.000000E+01 1.000000E+03 1.390213E+02 1.000000E+03 3.258276E+02 2.000000E+0
                                  2.000000E+01 1.000000E+03 1.390213E+02 1.000000E+03 3.258276E+02 2.000000E+0
   1  DISP      2     33(  5,--)  2.000000E+01 1.000000E+03 2.271459E-04 6.600000E+02 1.700317E-01 1.336996E+0
                                  2.000000E+01 1.000000E+03 2.271459E-04 6.600000E+02 1.700317E-01 1.336996E+0
   1  DISP      2     33(--, 11)  2.000000E+01 1.000000E+03 1.385571E+02 1.000000E+03 3.263339E+02 2.000000E+0
                                  2.000000E+01 1.000000E+03 1.385571E+02 1.000000E+03 3.263339E+02 2.000000E+0
   1  DISP      3     55(  5,--)  2.000000E+01 1.000000E+03 1.278678E-04 1.000000E+03 1.696787E-01 1.336996E+0
                                  2.000000E+01 1.000000E+03 1.278678E-04 1.000000E+03 1.696787E-01 1.336996E+0
   1  DISP      3     55(--, 11)  2.000000E+01 1.000000E+03 1.687413E+01 7.001384E+02 3.573561E+02 7.104853E+0
                                  2.000000E+01 1.000000E+03 1.687413E+01 7.001384E+02 3.573561E+02 7.104853E+0
```

17. MSC.Nastran Users have finished this exercise. MSC.Patran Users should proceed to the next step.

18. Proceed with the Reverse Translation process, that is attaching the **prob6.xdb** results file into MSC.Patran. To do this, return to the Analysis form and proceed as follows.

◆ **Analysis**

Action:	**Attach XDB**
Object:	**Result Entities**
Method	**Local**
Select Results File...	
Select File	**prob6.xdb**
OK	
Apply	

19. Plot the results in XY Plots.

The first plot is the Displacement versus Frequency plot at Node 11.

◆ **Results**

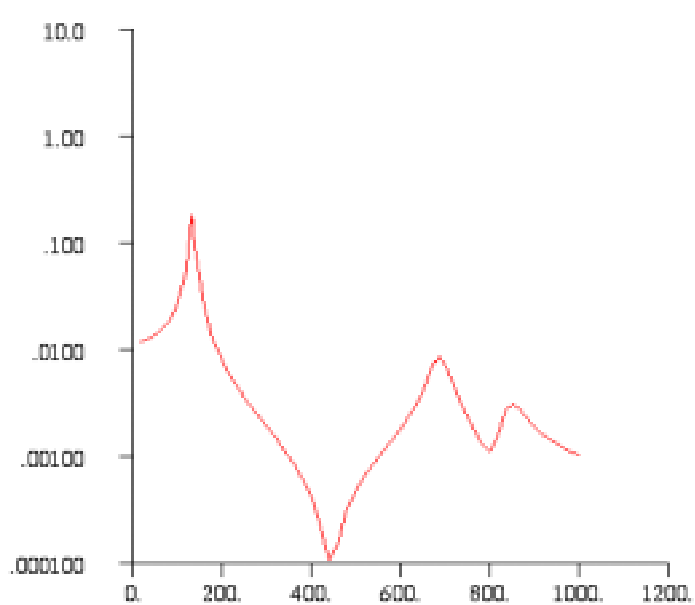
Action:	**Create**
Object:	**Graph**
Select Results Cases	**Frequency_dependent_0 of 54 subcases**
Filter Method	**All**
Filter	
Apply	
Close	
Select y Result:	**Displacement, Translational**
Quanity:	**Z Component**
x:	**Global Variable**
Variable:	**Frequency**

Plot Options

Complex No. as:	**Magnitude**
Display Attributes	
y axis scale	**Log**
Target Entities	
Select Nodes:	**Node 11**

Apply

Figure 6.3-Displacement Response at Loaded Corner

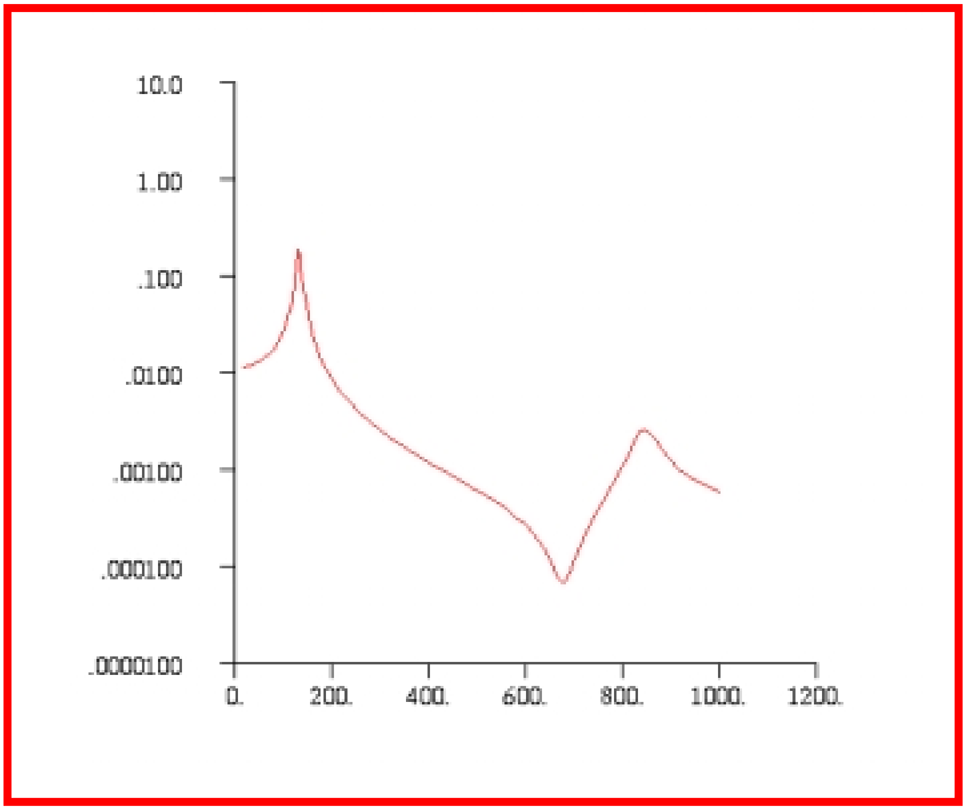


The second plot is the Displacement versus Frequency at Node 33..

Select Nodes:	**Node 33**

Apply

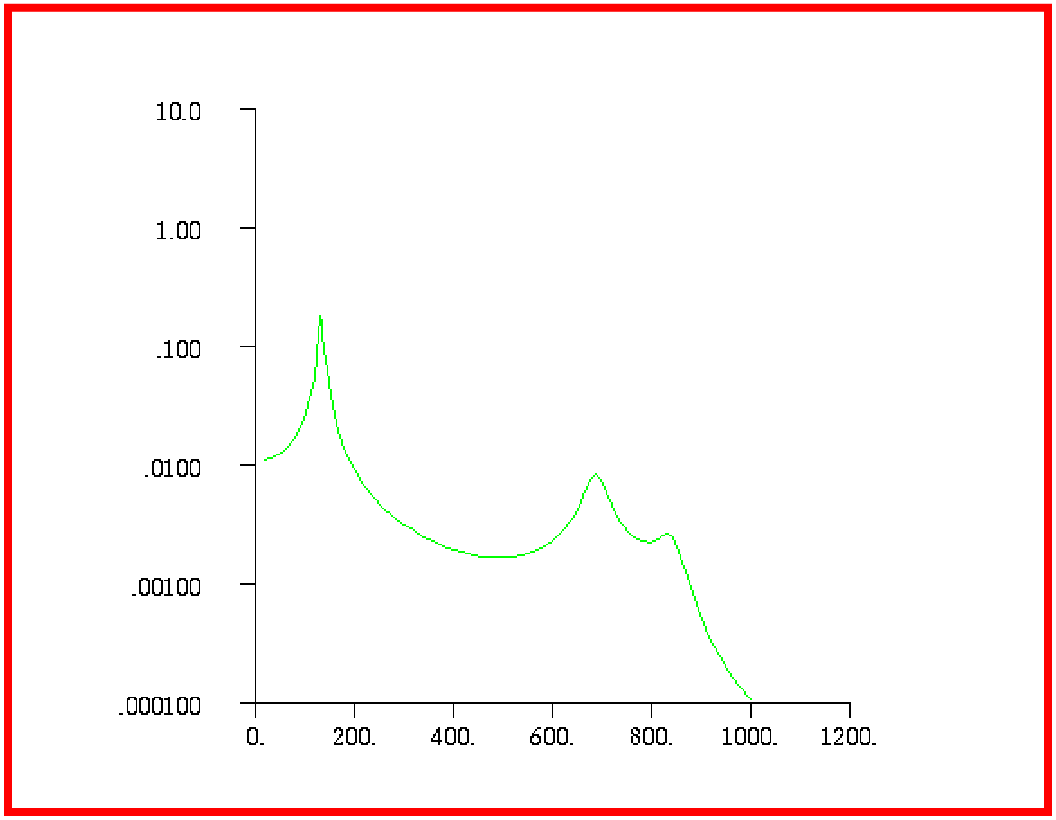
Figure 6.4-Displacement Response at Tip Center



The third plot is the Displacement versus Frequency at Node 55.

Select Nodes:	**Node 55**
Apply	

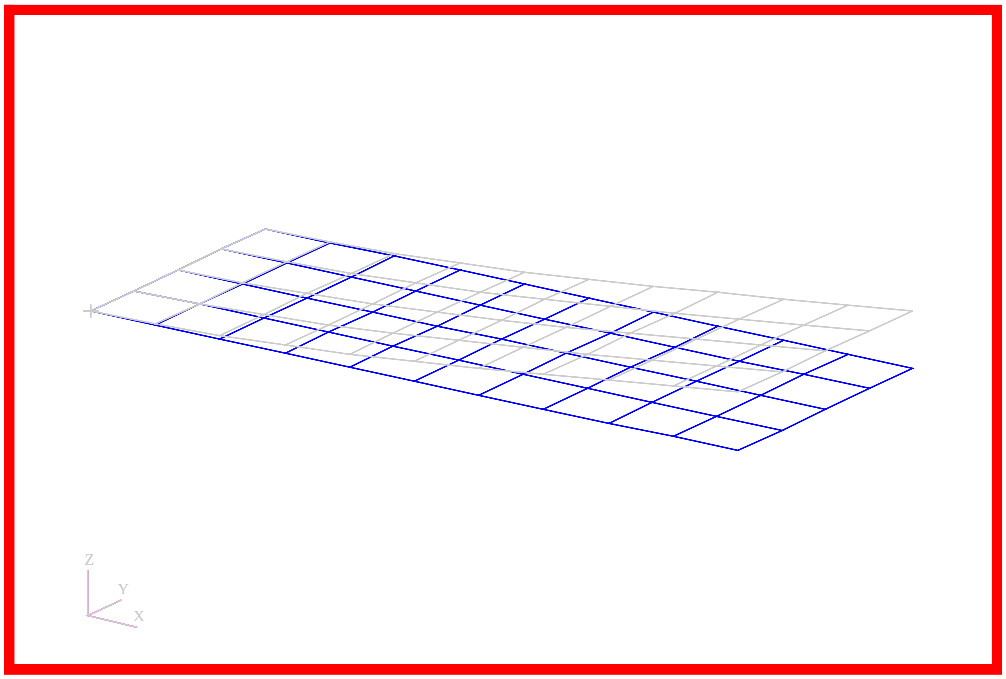
Figure 6.5-Displacement Response at Opposite Corner



Quit MSC.Patran when you have completed this exercise.

WORKSHOP 7

Direct Transient Response with Base Excitation



Objectives

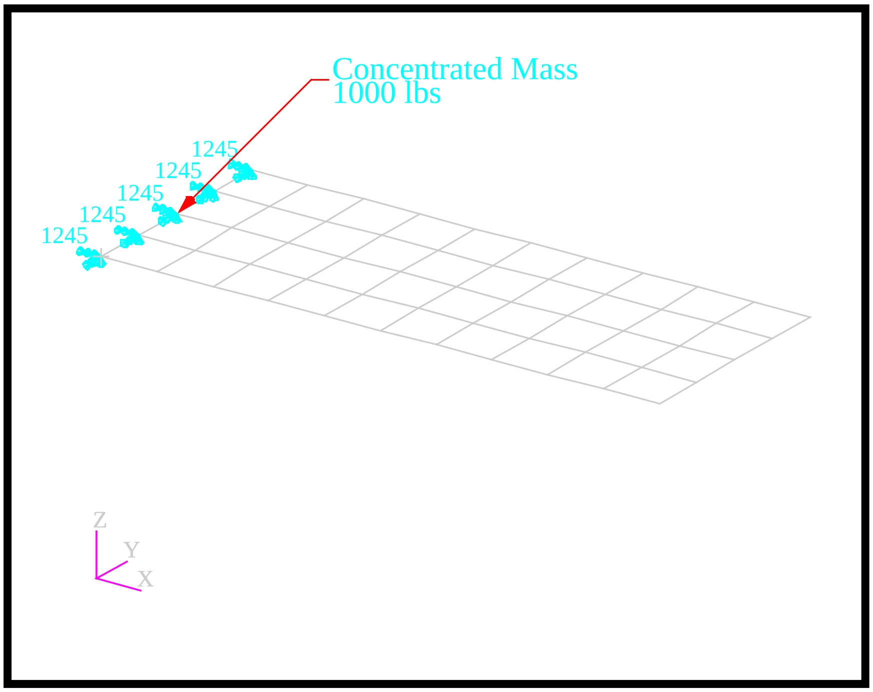
- Define a time-varying unit acceleration.
- Use the large mass method.
- Produce a MSC.Nastran input file from a dynamic math model, created in Workshop 1.
- Submit the file for analysis in MSC.Nastran.
- Compute nodal displacements for desired time domain.

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



Suggested Exercise Steps

- Reference previously created dynamic math model, **plate.bdf**, by using the INCLUDE statement.
- Modify base constraints and release displacements in the Z-direction.
- Define the time-varying unit acceleration (TLOAD2 and DAREA).
- Create the large mass at the base (CMASS2 and RBE2).
- Specify the structural damping and convert this damping to equivalent viscous damping.
  - PARAM, G, 0.06
  - PARAM, W3, 1571
- Specify integration time steps (TSTEP).
- Prepare the model for a direct transient analysis (SOL 109).
- Request response in terms of nodal displacement, velocity and acceleration at Grids 11, 33 and 55.
- Generate an input file and submit it to the MSC.Nastran solver for normal modes analysis.
- Review the results, specifically the nodal displacements, velocities, and acceleration.

```
ID SEMINAR,PROB7
```

```
CEND
```

```
BEGIN BULK
```

1	2	3	4	5	6	7	8	9	10

1	2	3	4	5	6	7	8	9	10

ENDDATA

Exercise Procedure:

1. Users who are not utilizing MSC.Patran for generating an input file should go to Step 11, otherwise, proceed to step 2.

2. Create a new database and named **prob7.db**

   File/New Database

   New Database Name **prob7**

   OK

   In the ***New Model Preference*** form set the following:

   Tolerance ◆ **Default**

   Analysis code: **MSC/NASTRAN**

   OK

3. Create the model by importing an existing MSC.Nastran input file, **(plate.bdf)**.

   ◆ **Analysis**

   Action: **Read Input File**

   Object: **Model Data**

   Method: **Translate**

   Select Input File

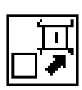
   Select Input File **plate.bdf**

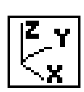
   OK

   Apply

   OK

4. Activate the entity labels by selecting the Show Labels icon on the tool-bar.

 **Show Labels**

 **Iso 3 View**

5. Create a time dependent load case for the transient response..

◆ **Load Cases**

Action:	**Create**
Load Case Name:	**transient_response**
Load Case Type:	**Time Dependent**
Assign/Prioritize Loads/BCs	
Select Individual Load/BCs *(Select from menu.)*	**Displ_spc1.1**
OK	
Apply	

6. Place the large mass at the base. To do this, it will be necessary to create a point element at Node 23.

◆ **Finite Elements**

Action:	**Create**
Object:	**Element**
Method:	**Edit**
Shape:	**Point**
Node 1 =	**Node 23**
Apply	

7. Next, define the scalar mass.

◆ **Properties**

Action:	**Create**
Dimension:	**0 D**
Type:	**Mass**
Property Set Name:	**scalar_mass**
Option(s)	**Grounded**

Input Properties...

Mass **1000**

DOF at Node 1 **UZ**

*(Select **String (UZ)** from the button on the right.)*

OK

Select Members: **Elm 41**

*(Change the select menu from **point** to **point element**, then select the point **element** which was previously created.)*

△ **Point element**

Add

Apply

8. Create the RBE2, which will connect the mass to the remaining base points.

◆ **Finite Elements**

Action: **Create**

Object: **MPC**

Type: **RBE2**

Define Terms ...

*(Turn off **Auto Execute**.)* ❒ **Auto Execute**

◆ **Create Independent**

Node List: **Node 23**

Apply

◆ **Create Dependent**

Node List: **Node 1, 12, 34, 45**

DOFs | **UZ**

Apply

Cancel

Apply

9. Create the time-dependent field for the unit acceleration.

◆ **Fields**

Action:	**Create**
Object:	**Non Spatial**
Method:	**Tabular Input**
Field Name:	**time_dependent_acceleration**

■ **Time (t)**

Options ...

Maximum Number of t:	**21**

OK

Input Data ...

Map Function to Table...

PCL Expression f'(t):	**sind(360.*250.*'t)**
Start time:	**0.0**
End time:	**0.004**
Number of Points:	**20**

Apply

Cancel

Go back to the ***Time/Frequency Scalar Table Data*** window. Add the following to row 21:

	Time(t)	Value
21	**0.04**	**0.0**

OK

Apply

9a. Now define the unit acceleration.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Force**
New Set Name:	**unit_acceleration**

Input Data...

Spatial Dependence:	**<0, 0, 2.588>**
Time/Freq. Dependence:	**f:time_dependent_acceleration**

OK

Select Application Region ...

◆ **FEM**

Node 23

Add

OK

Apply

9b. Finally, modify the existing constrants by releasing DOF3.

◆ **Load/BCs**

Action:	**Modify**
Object:	**Displacement**
Type:	**Nodal**
Select Set to Modify	**spc1.1**

Modify Data...

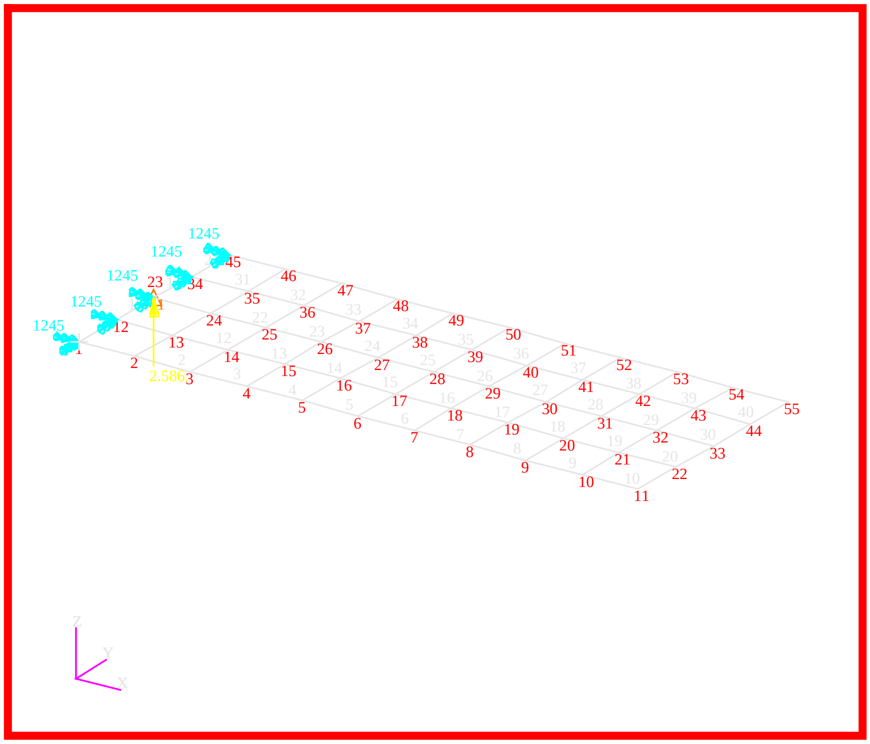
Translations < T1 T2 T3 >	**< 0, 0, >**
Rotations < R1 R2 R3 >	**< 0, 0, >**

OK

Apply

9c. The result should look like Figure 7.2

Figure 7.2



10. Generate the input file.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Analysis Deck**
Job Name:	**prob7**
Translation Parameters...	
Data Output:	**XDB and Print**
OK	
Solution Type...	
Solution Type:	**◆ TRANSIENT RESPONSE**

Formulation	**Direct**
Solution Parameters ...	
Mass Calculation	**Coupled**
Wt.-Mass Conversion	**.00259**
Struct. Dampening Coeff. =	**.06**
W3, Damping Factor =	**1571**
OK	
OK	
Subcase Create ...	
Available Subcases	**transient_response**
Subcase Parameters...	
Time Recovery Points	**DEFINE TIME STEPS ...**
Number of Time Steps =	**200**
Delta - T *(Hit Return to Input Data.)*	**.0002**
OK	
Output Request	
under ***Select Result Type***, highlight:	**Velocities** **Accelerations**

under ***Output Requests***, highlight:

SPCFORCES(SORT1,Real)=ALL FEM

Delete

OK

Apply

Cancel

Subcase Select ...

Subcases Selected: *(Click to deselect.)*	**Default**
Subcases for Solution Sequence: 109 *(Click to select.)*	**transient_response**

OK

Apply

An MSC.Nastran input file called **prob7.bdf** will be generated. This process of translating your model into an input file is called the Forward Translation. The Forward Translation is complete when the Heartbeat turns green. MSC.Patran Users should proceed to step 12.

Generating an input file for MSC.Nastran Users:

MSC.Nastran users can generate an input file using the data previously stated. The result should be similar to the output below.

11. MSC.Nastran Input File: **prob7.dat**

```
ID SEMINAR, PROB7
SOL 109
TIME 30
CEND
TITLE = TRANSIENT RESPONSE WITH BASE EXCITATION
SUBTITLE = USING DIRECT TRANSIENT METHOD, NO REDUCTION
ECHO = UNSORTED
SPC = 200
SET 111 = 23, 33
DISPLACEMENT (SORT2) = 111
VELOCITY (SORT2) = 111
ACCELERATION (SORT2) = 111
SUBCASE 1
DLOAD = 500
TSTEP = 100
$
OUTPUT (XYPLOT)
XGRID=YES
YGRID=YES
XTITLE= TIME (SEC)
YTITLE= BASE ACCELERATION
XYPLOT ACCELERATION RESPONSE / 23 (T3)
YTITLE= BASE DISPLACEMENT
XYPLOT DISP RESPONSE / 23 (T3)
YTITLE= TIP CENTER DISPLACEMENT RESPONSE
XYPLOT DISP RESPONSE / 33 (T3)
$
BEGIN BULK
$
$ PLATE MODEL DESCRIBED IN NORMAL MODES EXAMPLE
$
INCLUDE 'plate.bdf'
PARAM, COUPMASS, 1
PARAM, WTMASS, 0.00259
$
$ SPECIFY STRUCTURAL DAMPING
$
PARAM, G, 0.06
```

```
PARAM, W3, 1571.
$
$ APPLY EDGE CONSTRAINTS
$
SPC1, 200, 12456, 1, 12, 23, 34, 45
$
$ PLACE BIG FOUNDATION MASS (BFM) AT BASE
$
CMASS2, 100, 1000., 23, 3
$
$ RBE MASS TO REMAINING BASE POINTS
$
RBE2, 101, 23, 3, 1, 12, 34, 45
$
$ APPLY LOADING TO FOUNDATION MASS
$
TLOAD2, 500, 600, , 0, 0.0, 0.004, 250., -90.
$
DAREA, 600, 23, 3, 2.588
$
$ SPECIFY INTEGRATION TIME STEPS
$
TSTEP, 100, 200, 2.0E-4, 1
$
ENDDATA
```

Submitting the input file for analysis:

12. Submit the input file to MSC.Nastran for analysis.

    12a. To submit the MSC.Patran **.bdf** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran prob7.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

    12b. To submit the MSC.Nastran **.dat** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran prob7 scr=yes**. Monitor the run using the UNIX **ps** command.

13. When the run is completed, use **plotps** utility to create a postscript file, **prob7.ps**, from the binary plot file **prob7.plt**. The displacement, velocity, and acceleration response plots for Grids 11, 33 and 55 are shown in figures 7.3 to 7.8.

14. When the run is completed, edit the **prob7.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing WARNING messages indicate modeling errors.

15. While still editing **prob7.f06**, search for the word:

    D I S P L (spaces are necessary)

    Displacement at Grid 23

    Time T3

    .0 = __________

    .02 = __________

    .04 = __________

    Displacement at Grid 33

    Time T3

    .0 = __________

    .02 = __________

    .04 = __________

    V E L O C (spaces are necessary)

    Velocity at Grid 23

    Time T3

    .0 = __________

    .02 = __________

    .04 = __________

    Velocity at Grid 33

    Time T3

    .0 = __________

    .02 = __________

    .04 = __________

A C C E L (spaces are necessary)

Acceleration at Grid 23

Time T3

.0 = __________

.02 = __________

.04 = __________

Acceleration at Grid 33

Time T3

.0 = __________

.02 = __________

.04 = __________

Comparison of Results

16. Compare the results obtained in the **.f06** file with the following results:

POINT-ID = 23

D I S P L A C E M E N T   V E C T O R

TIME	TYPE	T1	T2	T3	R1	R2
.0	G	.0	.0	.0	.0	.0
1.999998E-02	G	.0	.0	2.511800E-06	.0	.0
3.999996E-02	G	.0	.0	2.523053E-06	.0	.0

POINT-ID = 33

D I S P L A C E M E N T   V E C T O R

TIME	TYPE	T1	T2	T3	R1	R2
.0	G	.0	.0	.0	.0	.0
1.999998E-02	G	.0	.0	4.559710E-06	7.870200E-20	-5.699044E-07
3.999996E-02	G	.0	.0	7.363514E-07	-8.475326E-19	4.972196E-07

POINT-ID = 23

V E L O C I T Y   V E C T O R

TIME	TYPE	T1	T2	T3	R1	R2
.0	G	.0	.0	1.029922E-05	.0	.0
1.999998E-02	G	.0	.0	4.035838E-07	.0	.0
3.999996E-02	G	.0	.0	4.712748E-07	.0	.0

POINT-ID = 33

V E L O C I T Y   V E C T O R

TIME	TYPE	T1	T2	T3	R1	R2
.0	G	.0	.0	-4.379307E-07	1.492096E-20	-1.481305E-07
1.999998E-02	G	.0	.0	1.220225E-03	-2.594927E-15	-3.394969E-04
3.999996E-02	G	.0	.0	6.578242E-04	3.797481E-15	-1.829342E-04

POINT-ID = 23

A C C E L E R A T I O N   V E C T O R

TIME	TYPE	T1	T2	T3	R1	R2
.0	G	.0	.0	1.029922E-01	.0	.0
1.999998E-02	G	.0	.0	1.615566E-04	.0	.0
3.999996E-02	G	.0	.0	-1.353748E-04	.0	.0

POINT-ID = 33

A C C E L E R A T I O N   V E C T O R

TIME	TYPE	T1	T2	T3	R1	R2
.0	G	.0	.0	-4.379307E-03	1.492096E-16	-1.481305E-03
1.999998E-02	G	.0	.0	-1.464230E+00	-1.464654E-11	4.074106E-01
3.999996E-02	G	.0	.0	1.231636E+00	1.381082E-11	-3.427888E-01

17. MSC.Nastran Users have finished this exercise. MSC.Patran Users should proceed to the next step.

18. Proceed with the Reverse Translation process, that is attaching the **prob7.xdb** results file into MSC.Patran. To do this, return to the Analysis form and proceed as follows:

◆ **Analysis**

Action:	**Attach XDB**
Object:	**Result Entities**
Method:	**Local**
Select Results File...	
Select Results File	**prob7.xdb**
OK	
Apply	

When the translation is complete bring up the form.

◆ **Results**

Action:	**Create**
Object:	**Graph**
Method:	**y vs x**
Select Result Case(s)	**Transient_response, 0 of 201 subcases**
Filter Method	**All**
Filter	
Apply	
Close	
Select y Result:	**Displacements, Translational**
Quanity:	**Z Component**
x:	**Global Variable**
Variable:	**Time**

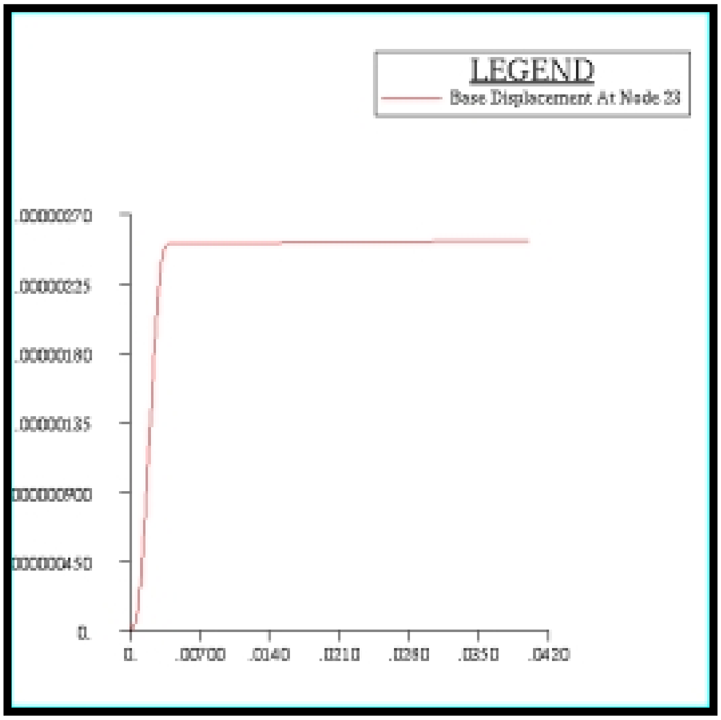
Target Entities

Select Nodes: **Node 23**

Apply

The output should look similar to Figure 7.3

Figure 7.3-Base Displacement at Node 23



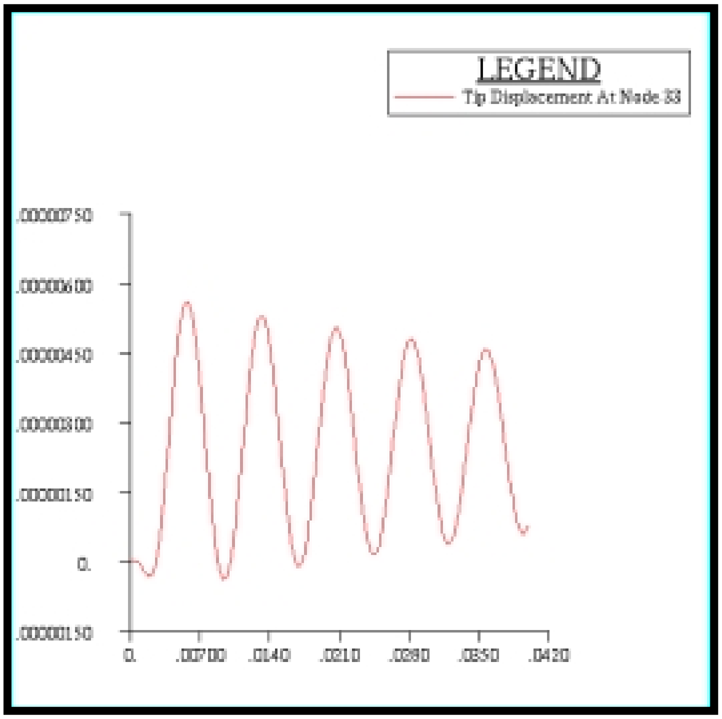
19. Repeat the procedure to find the nodal displacement for Node 33.

Select Nodes: **Node 33**

Apply

The output should look similar to Figure 7.4

Figure 7.4-Tip Displacement at Node 33

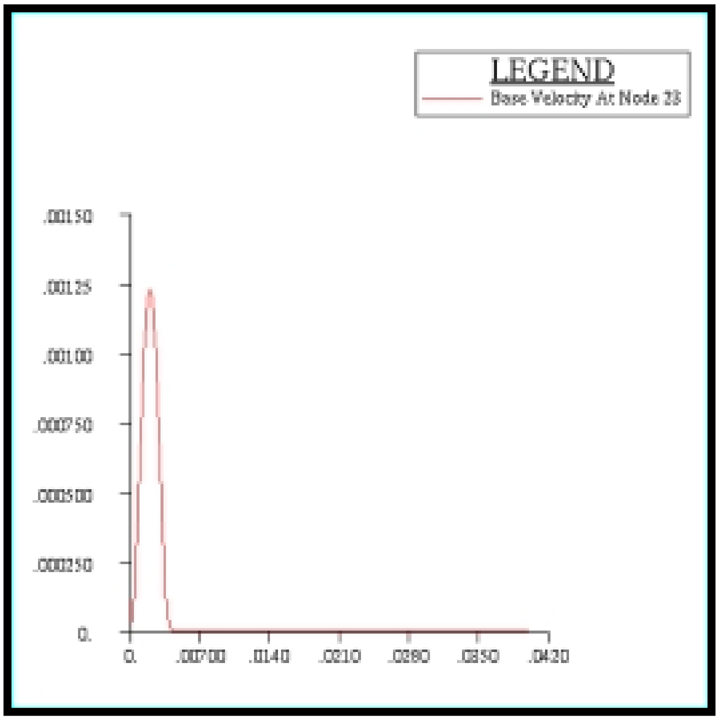


20. Repeat the procedure to find the nodal velocity for Node 23.

Select Results	
Select y Result:	**Velocities, Translational**
Quanity:	**Z Component**
x:	**Global Variable**
Variable:	**Time**
Target Entities	
Select Nodes:	**Node 23**
Apply	

The output should look similar to Figure 7.5

Figure 7.5-Base Velocity at Node 23



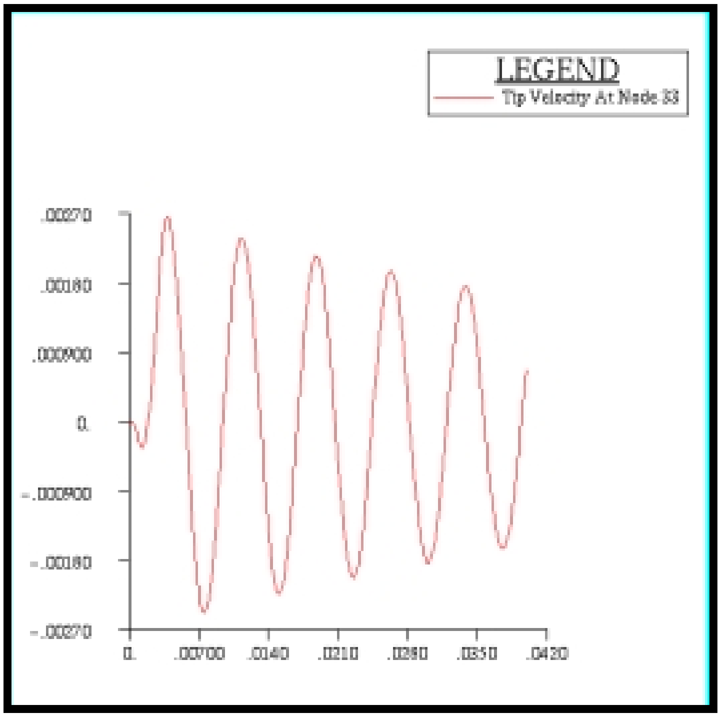
21. Repeat the procedure to find the velocity at Node 33.

Select Nodes: **Node 33**

Apply

The output should look similar to Figure 7.6

Figure 7.6-Tip Velocity at Node 33



22. Repeat the procedure to find the nodal acceleration for Node 23.

Select Results

Select y Result:	**Accelerations, Translational**
Quanity:	**Z Component**
x:	**Global Variable**
Variable:	**Time**

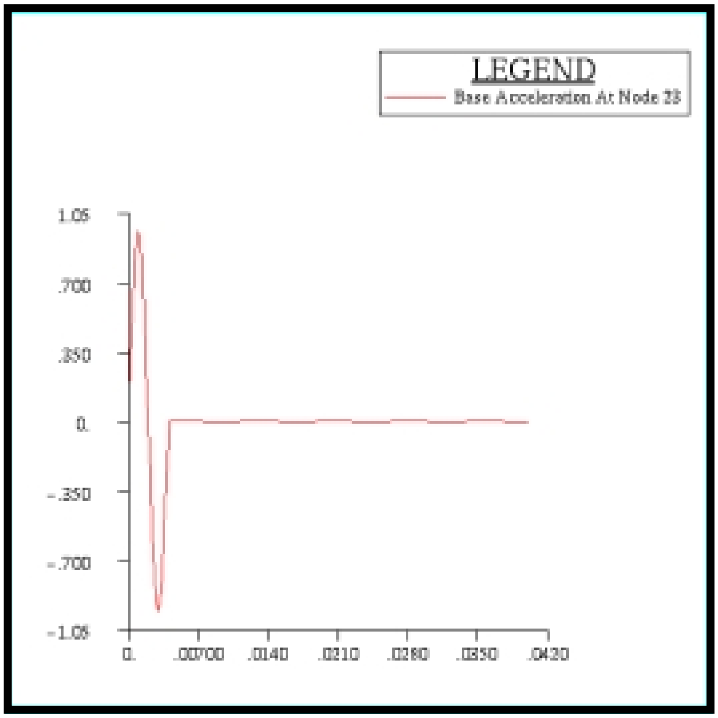
Target Entities

Select Nodes:	**Node 23**

Apply

The output should be similar to Figure 7.7

Figure 7.7-Base Acceleration at Node 23

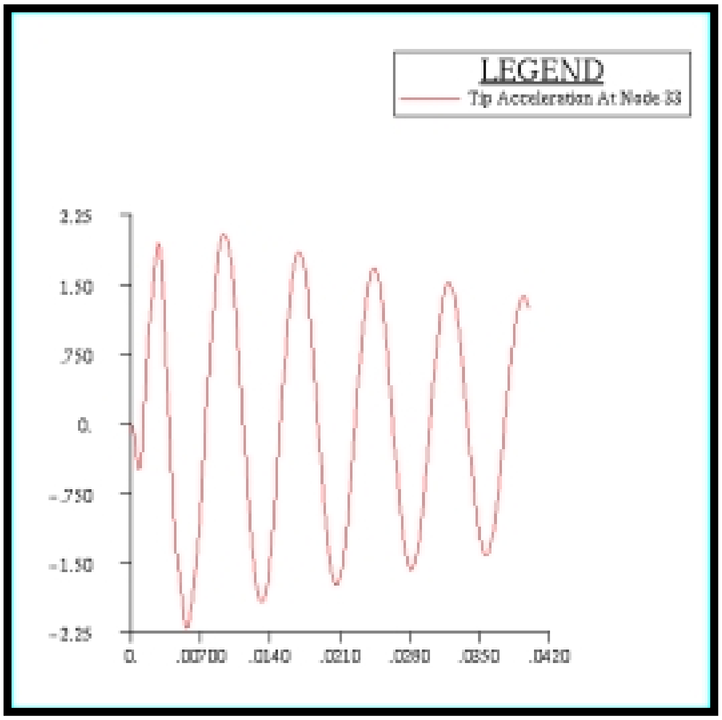


23. Repeat the procedure to find the acceleration at Node 33.

Select Nodes:	**Node 33**
Apply	

The output should look similar to Figure 7.8

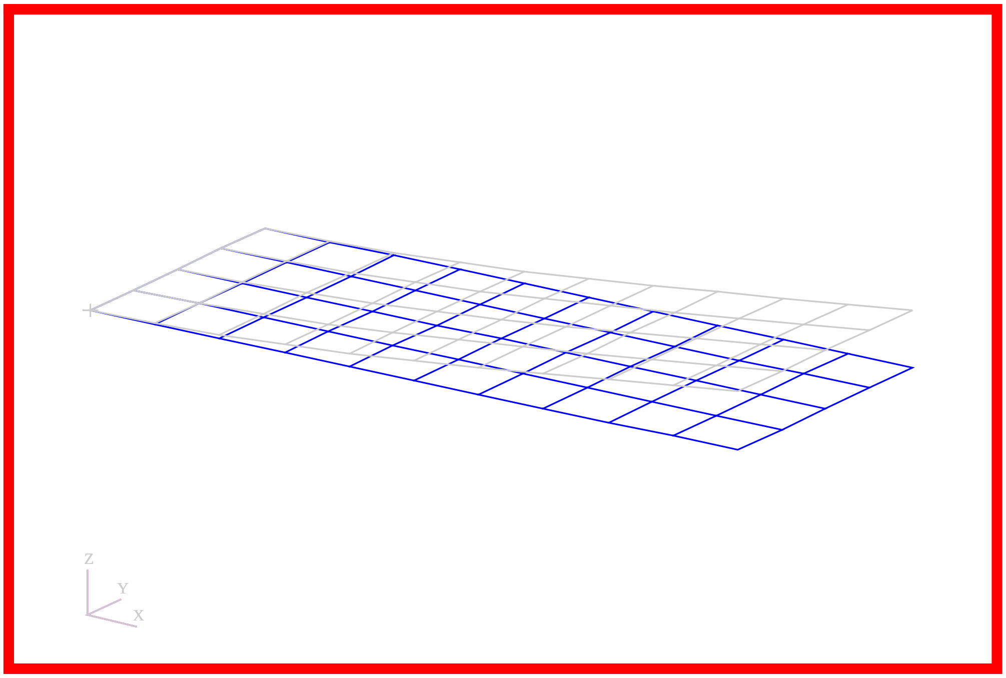
Figure 7.8-Tip Acceleration at Node 33



Quit MSC.Patran when you are finished with this exercise.

WORKSHOP 8

Enforced Motion with Direct Frequency Response



Objectives

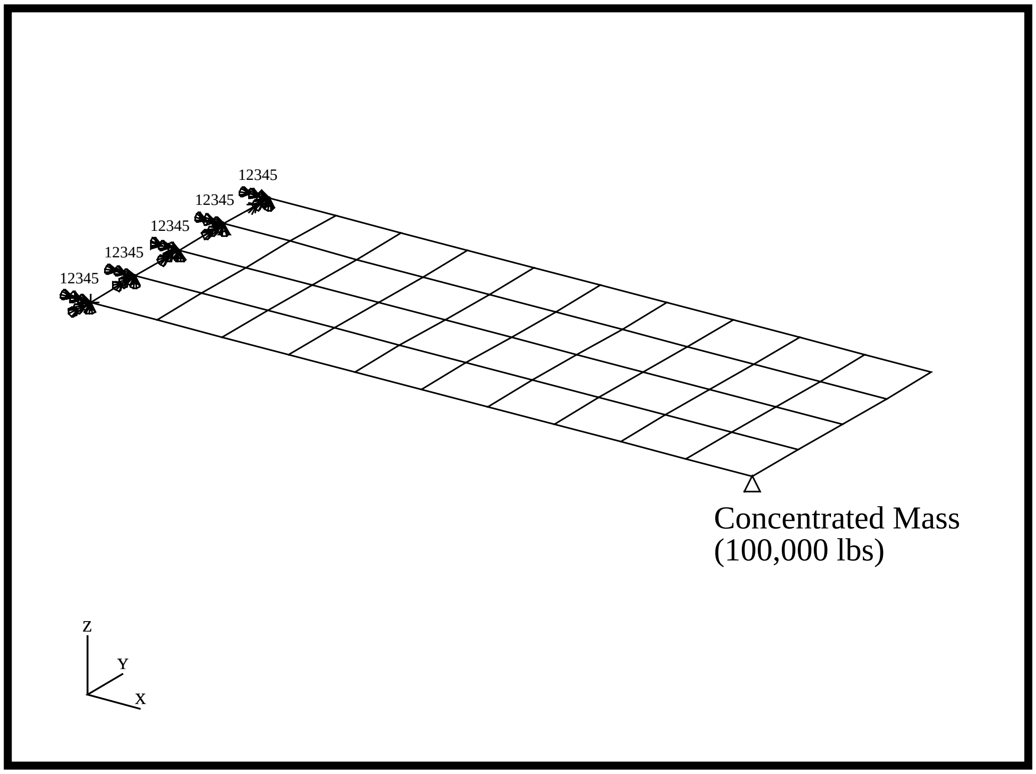
- Define frequency-varying tip displacement.
- Use the large mass method.
- Produce a MSC.Nastran input file from a dynamic math model created in Workshop 1.
- Submit the file for analysis in MSC.Nastran.
- Compute nodal displacements for desired time domain.

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



Suggested Exercise Steps

- Reference previously created dynamic math model, **plate.bdf**, by using the INCLUDE statement.
- Create the large mass at a corner of the tip (CMASS2).
- Define the frequency-varying tip displacement (RLOAD2, TABLED4, DAREA).
- Define a set of frequencies to be used in the solution (FREQ1).
- Prepare the model for a direct frequency analysis (SOL 108).
- Sprecify the structural damping.
  - PARAM, G, 0.06
- Request response in terms of nodal displacement and grid points 11, 33, and 55.
- Generate an input file, and submit it to the MSC.Nastran solver for direct transient analysis.
- Review the results, specifically the grid displacements.

```
ID SEMINAR,PROB8
```

```
CEND
```

```
BEGIN BULK
```

1	2	3	4	5	6	7	8	9	10

1	2	3	4	5	6	7	8	9	10

ENDDATA

Exercise Procedure:

1. Users who are not utilizing MSC.Patran for generating an input file should go to Step 9, otherwise, proceed to step 2.

2. Create a new database and named **prob8.db**.

File/New Database

New Database Name	**prob8**
OK	

In the *New Model Preference* form set the following:

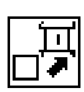
Tolerance	◆ **Default**
Analysis code:	**MSC/NASTRAN**
OK	

3. Create the model by importing an existing MSC.Nastran input file, **(plate.bdf)**.

◆ **Analysis**

Action:	**Read Input File**
Object:	**Model Data**
Method:	**Translate**
Select Input File	
Select Input File	**plate.bdf**
OK	
Apply	
OK	

4. Activate the entity labels by selecting the Show Labels icon on the tool-bar.

 **Show Labels**

5. Create the frequency dependent load case.

◆ **Load Cases**

Action:	**Create**
Load Case Name:	**frequency_response**
Load Case Type:	**Time Dependent**
Assign/Prioritize Loads/BCs	
Select Individual Load/BCs *(Select from menu.)*	**Displ_spc1.1**
OK	
Apply	

6. Place a large mass at a corner of the tip (**Node 11**). However, a point element must be created first.

◆ **Finite Element**

Action:	**Create**
Object:	**Element**
Method:	**Edit**
Shape:	**Point**
Node 1 =	**Node 11**
Apply	

7. Then define the scalar mass.

◆ **Properties**

Action:	**Create**
Dimension:	**0D**
Type:	**Mass**
Property Set Name:	**scalar_mass**
Option(s):	**Grounded**
Input Properties ...	

Mass:	**1.0E+5**
Dof at Node 1 (Value Type)	**UZ**

OK

Select Members: *(Click on point element in select menu. Then select Elm 41.)*	**Elm 41**

△ **Point element**

Add

Apply

8. Start the analysis.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Analysis Deck**
Jobname:	**prob8**

Translation Parameters...

Data Output:	**XDB and Print**

OK

Solution Type...

Solution Type:	◆ **FREQUENCY RESPONSE**
Formulation:	**Direct**

Solution Parameters ...

Mass Calculation	**Coupled**
Wt.-Mass Conversion	**.00259**
Structure Damping Coeff:	**0.06**

OK

OK

Direct Text Input ...

◆ **Bulk Data Section**

Clear

Bulk Data Section:
(Each line in the box is a separate line to input!)

RLOAD2,500,600, , ,310
TABLED4,310,0.,1.,0.,10000.,
+,0.,0.,-39.4784,ENDT
DAREA,600,11,3,25.8799

◆ **Case Control Section**

Clear

Case Control Section: **DLOAD=500**

OK

Subcase Create...

Available Subcases **frequency_response**

Subcase Parameters...

DEFINE FREQUENCIES...

Starting Frequency = **20**
(Hit Return to Input Data.)

Ending Frequency = **1000**
(Hit Return to Input Data.)

# of Freq. Increments = **49**
(Hit Return to Input Data.)

OK

Output Requests...

under ***Output Request*** highlight:

SPCFORCES(SORT1,Real)=All FEM

Delete

OK

Apply

Cancel

Subcase Select ...

Subcases Selected: click on **Default**

Subcases for Solution Sequence: 108 click on **frequency_response**

OK

Apply

An MSC.Nastran input file called **prob8.bdf** will be generated. This process of translating your model into an input file is called the Forward Translation. The Forward Translation is complete when the Heartbeat turns green. MSC.Patran Users should proceed to step 10.

Generating an input file for MSC.Nastran Users:

MSC.Nastran users can generate an input file using the data previously stated. The result should be similar to the output below.

9. MSC.Nastran input file: **prob8.dat**

```
ID SEMINAR, PROB8
SOL 108
TIME 30
CEND
TITLE= FREQUENCY RESPONSE DUE TO .1 DISPLACEMENT AT TIP
SUBTITLE= DIRECT METHOD
ECHO= UNSORTED
SPC= 1
SET 111= 11, 33, 55
DISPLACEMENT(PHASE, SORT2)= 111
SDISP(PHASE, SORT2)= ALL
set 222 = 11
OLOAD= 222
SUBCASE 1
DLOAD= 500
FREQUENCY= 100
$
OUTPUT (XYPLOT)
$
XTGRID= YES
YTGRID= YES
XBGRID= YES
YBGRID= YES
YTLOG= YES
YBLOG= NO
XTITLE= FREQUENCY (HZ)
YTTITLE= DISPLACEMENT RESPONSE AT LOADED CORNER, MAGNITUDE
YBTITLE= DISPLACEMENT RESPONSE AT LOADED CORNER, PHASE
XYPLOT DISP RESPONSE / 11 (T3RM, T3IP)
YTTITLE= DISPLACEMENT RESPONSE AT TIP CENTER, MAGNITUDE
YBTITLE= DISPLACEMENT RESPONSE AT TIP CENTER, PHASE
XYPLOT DISP RESPONSE / 33 (T3RM, T3IP)
YTTITLE= DISPLACEMENT RESPONSE AT OPPOSITE CORNER, MAGNITUDE
YBTITLE= DISPLACEMENT RESPONSE AT OPPOSITE CORNER, PHASE
XYPLOT DISP RESPONSE / 55 (T3RM, T3IP)
$
BEGIN BULK
$
```

```
$ PLATE MODEL DESCRIBED IN NORMAL MODES EXAMPLE
$
INCLUDE 'plate.bdf'
PARAM, COUPMASS, 1
PARAM, WTMASS, 0.00259
$
$ SPECIFY STRUCTURAL DAMPING
$
PARAM, G, 0.06
$
$ APPLY UNIT DISPLACEMENT AT TIP POINT
$
CMASS2, 5000, 1.0E+5, 11, 3
$
RLOAD2, 500, 600, , ,310
$
TABLED4, 310, 0., 1., 0., 10000.,
 ,0., 0., -39.4784, ENDT
$
DAREA, 600, 11, 3, 25.8799
$
$ SPECIFY FREQUENCY STEPS
$
FREQ1, 100, 20., 20., 49
$
ENDDATA
```

Submitting the input file for analysis:

10. Submit the input file to MSC.Nastran for analysis.

    10a. To submit the MSC.Patran **.bdf** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran prob8.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

    10b. To submit the MSC.Nastran **.dat** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran prob8 scr=yes**. Monitor the run using the UNIX **ps** command.

11. When the run is completed, use **plotps** utility to create a postscript file, **prob8.ps**, from the binary plot file **prob8.plt**. The displacement response plots for Grids 11, 33 and 55 are shown in figures 8.2 to 8.7**.**

12. Edit the **prob8.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing WARNING messages indicate modeling errors.

13. While still editing **prob8.f06**, search for the word:

X Y - O U T P U T   S U M M A R Y (spaces are necessary).

Displacement at Grid 11

Frequency (X)		Displacement (Y)
140	=	___________
380	=	___________

Displacement at Grid 33

Frequency (X)		Displacement (Y)
140	=	___________
600	=	___________

Displacement at Grid 55

Frequency (X)		Displacement (Y)
140	=	___________
1000	=	___________

Comparison of Results

14. Compare the results obtained in the **.f06** file with the following results:

```
                        X Y - O U T P U T   S U M M A R Y  ( R E S P O N S E )
SUBCASE CURVE FRAME               XMIN-FRAME/   XMAX-FRAME/   YMIN-FRAME/     X FOR       YMAX-FRAME/     X FOR
  ID    TYPE   NO.   CURVE ID.     ALL DATA      ALL DATA      ALL DATA       YMIN         ALL DATA        YMAX
     1 DISP     1   11(  5,--)  2.000000E+01  1.000000E+03  9.992202E-02  3.600000E+02  9.992512E-02   2.000000E+0
                                2.000000E+01  1.000000E+03  9.992202E-02  3.600000E+02  9.992512E-02   2.000000E+01
     1 DISP     1   11(--, 11)  2.000000E+01  1.000000E+03  7.680080E-07  1.000000E+03  3.828149E-04   3.800000E+02
                                2.000000E+01  1.000000E+03  7.680080E-07  1.000000E+03  3.828149E-04   3.800000E+02
     1 DISP     2   33(  5,--)  2.000000E+01  1.000000E+03  2.312926E-03  6.000000E+02  8.446401E-01   3.800000E+02
                                2.000000E+01  1.000000E+03  2.312926E-03  6.000000E+02  8.446401E-01   3.800000E+02
     1 DISP     2   33(--, 11)  2.000000E+01  1.000000E+03  3.348117E-01  9.799999E+02  3.599947E+02   2.000000E+01
                                2.000000E+01  1.000000E+03  3.348117E-01  9.799999E+02  3.599947E+02   2.000000E+01
     1 DISP     3   55(  5,--)  2.000000E+01  1.000000E+03  2.434351E-02  1.000000E+03  1.624350E+00   3.800000E+02
                                2.000000E+01  1.000000E+03  2.434351E-02  1.000000E+03  1.624350E+00   3.800000E+02
     1 DISP     3   55(--, 11)  2.000000E+01  1.000000E+03  3.690138E+00  1.000000E+03  3.599892E+02   2.000000E+01
                                2.000000E+01  1.000000E+03  3.690138E+00  1.000000E+03  3.599892E+02   2.000000E+01
```

15. **MSC.Nastran Users have finished this exercise. MSC.Patran Users should proceed to the next step.**

16. Proceed with the Reverse Translation process, that is attaching the **prob8.xdb** results file into MSC.Patran. To do this, return to the Analysis form and proceed as follows:

◆ **Analysis**

Action:	**Attach XDB**
Object:	**Result Entities**
Method:	**Local**
Select Results File...	
Select Results File	**prob8.xdb**
OK	
Apply	

17. Plot the results in XY plots.

The first plot is to make the Displacement versus Frequency plot at Node 11.

◆ **Results**

Action:	**Create**
Object:	**Graph**
Method	**y vs x**
Select Results Cases	**Frequency_response, 0 of 50 subcases**
Filter Method	**All**
Filter	
Apply	
Close	
Select y Result:	**Displacement, Translational**
Quanity:	**Z Component**
x:	**Global Variable**
Variable:	**Frequency**

Plot Options

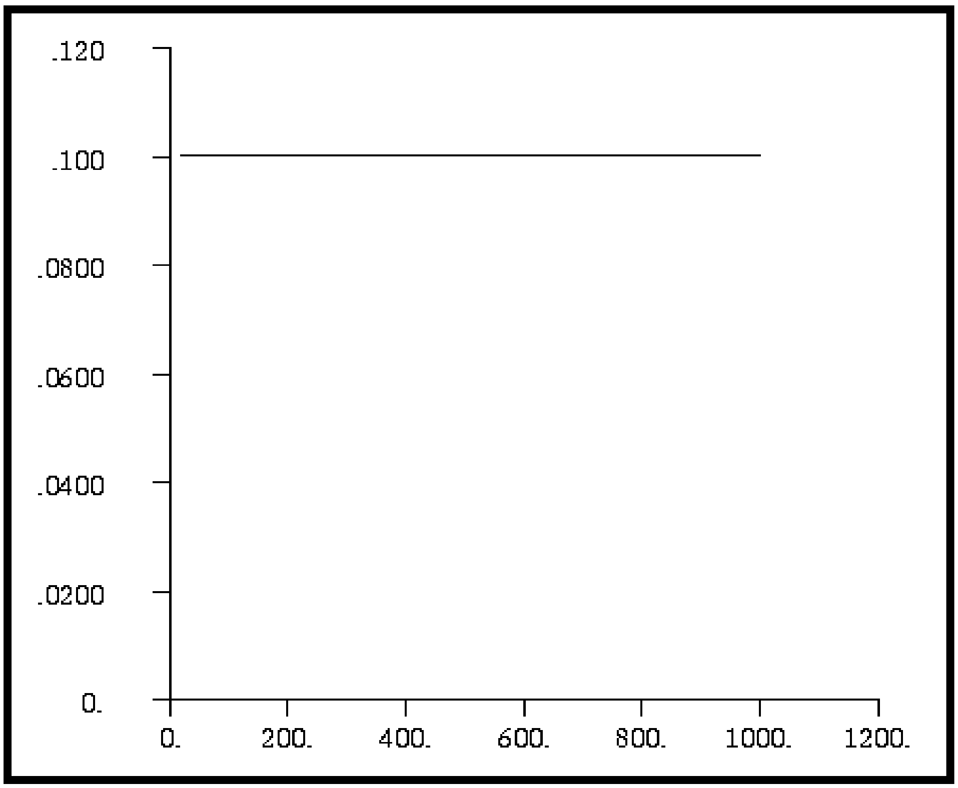
Complex No. as: **Magnitude**

Target Entities

Select Nodes: **Node 11**

Apply

Figure 8.2-Displacement Response at Node 11.



The next step is to make the plot of Phase versus Frequency. .

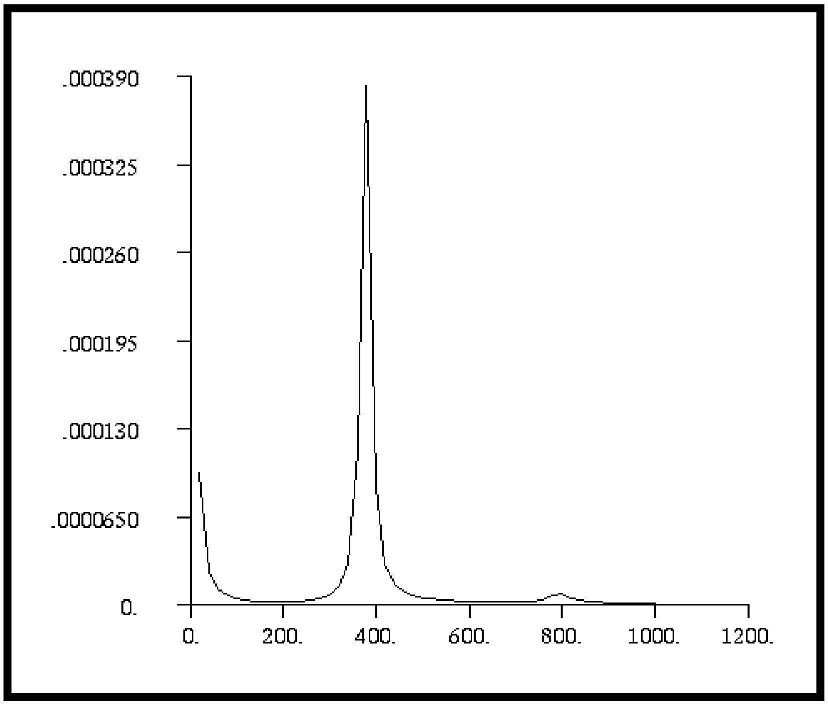
Plot Options

Complex No as: **Phase**

Apply

Note the scale on the y axis is in degrees.

Figure 8.3-Phase Angle at Node 11



Repeat the above steps of plotting the XY plots of Grids 11 for Node 33 and 55.

Plot Options

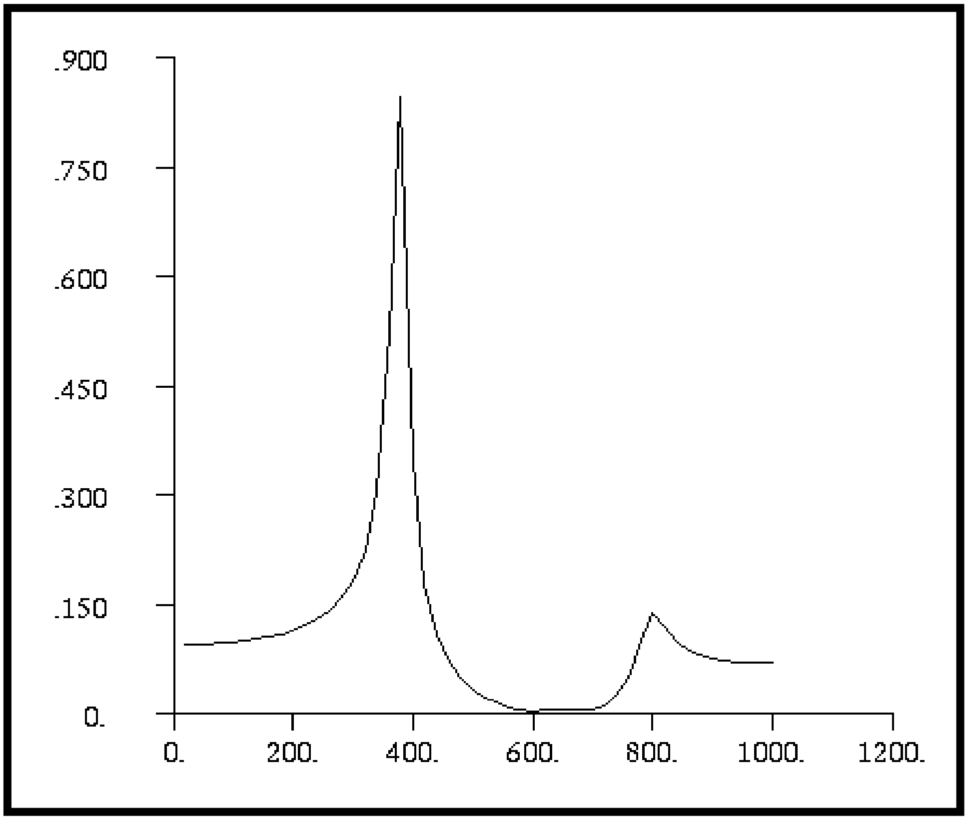
Complex No as:	**Magnitude**

Target Entities

Select Nodes:	**Node 33**

Apply

Figure 8.4-Displacement Response at Node 33

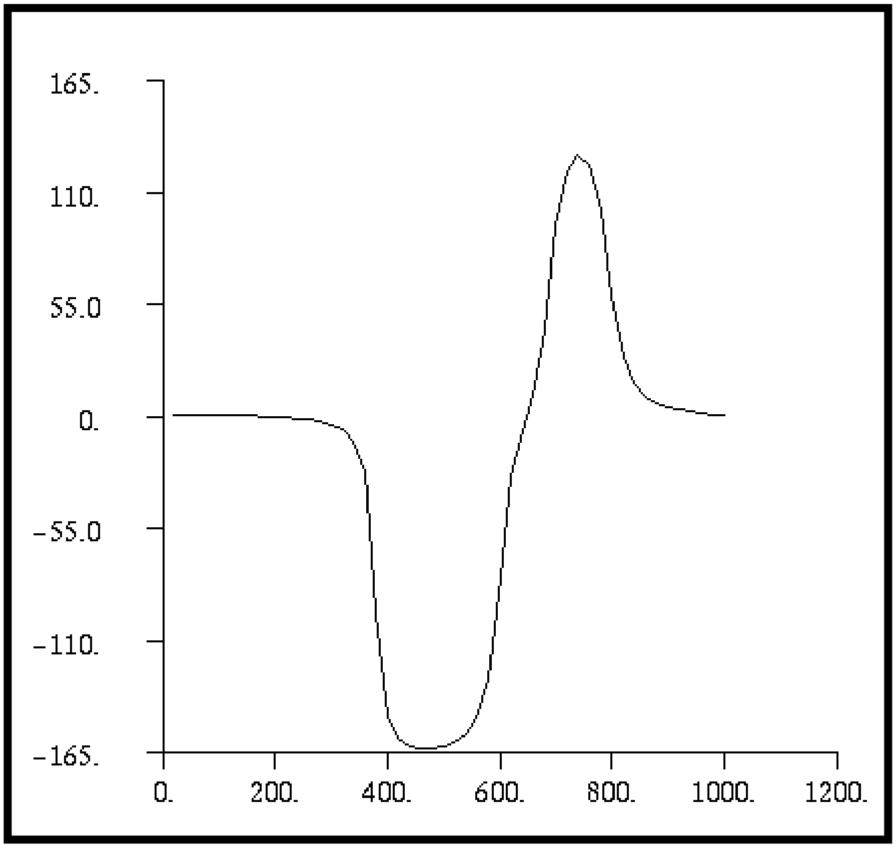


Plot Options

Complex No as: **Phase**

Apply

Figure 8.5-Phase Angle at Node 33



Plot Options

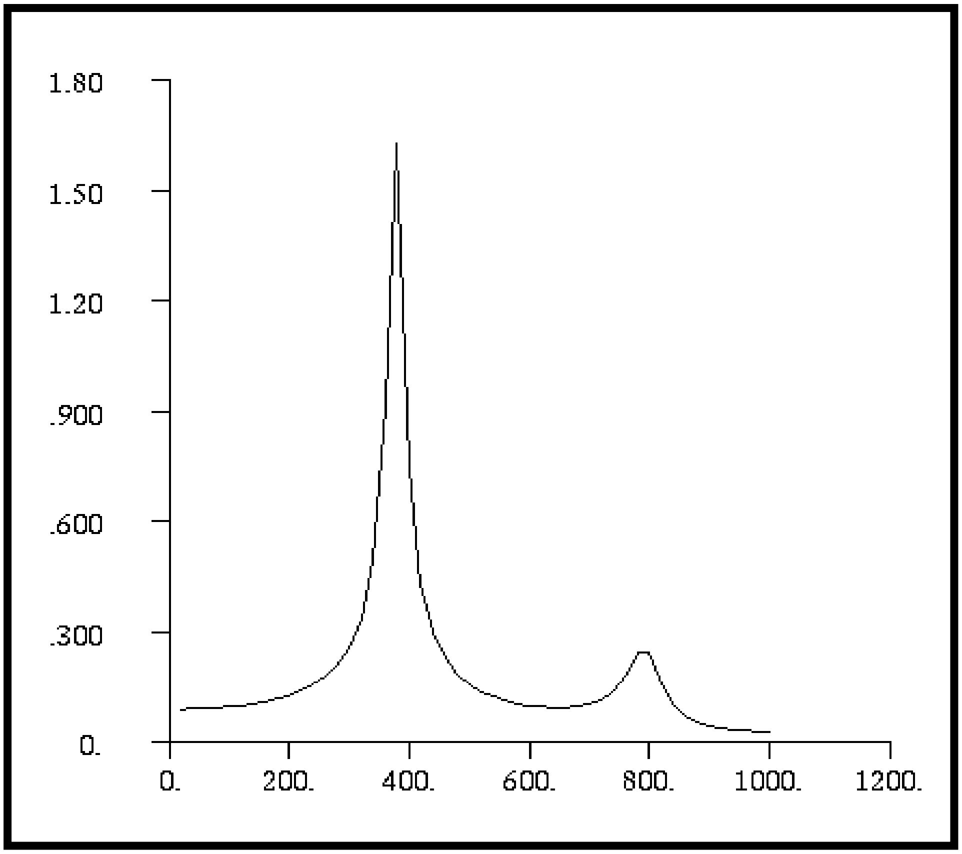
Complex No as: **Magnitude**

Target Entities

Select Nodes: **Node 55**

Apply

Figure 8.6-Displacement Response at Node 55

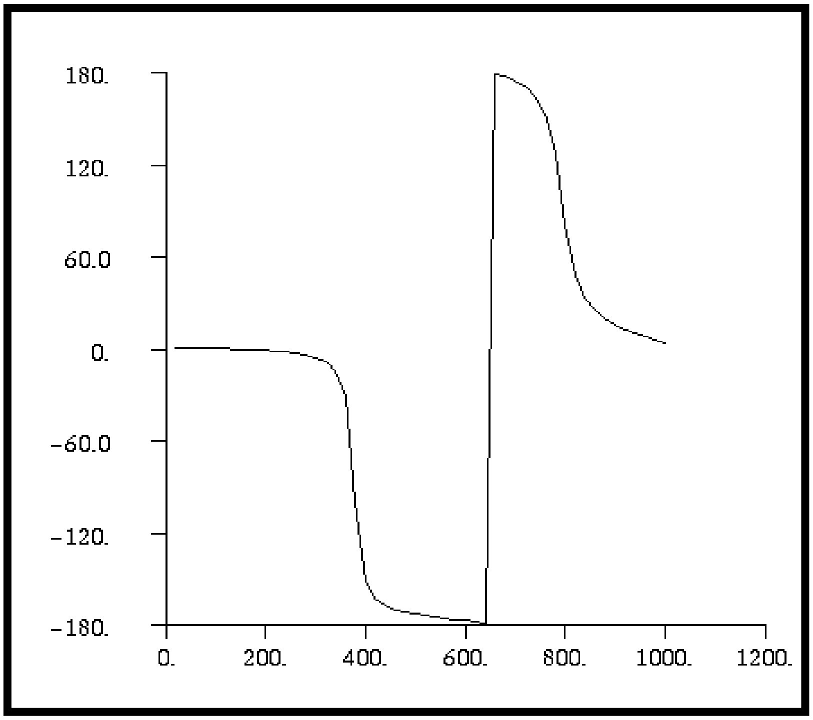


Plot Options

Complex No as: **Phase**

Apply

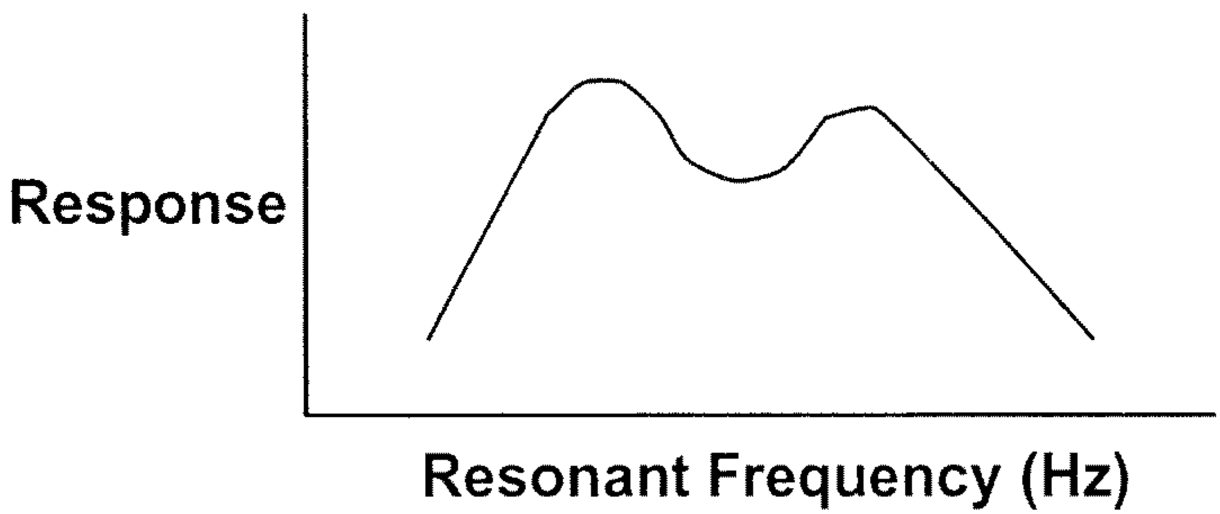
Fig 8.7-Phase Angle at Node 55



Quit MSC.Patran when you have completed this exercise.

WORKSHOP 9a

Response Spectra



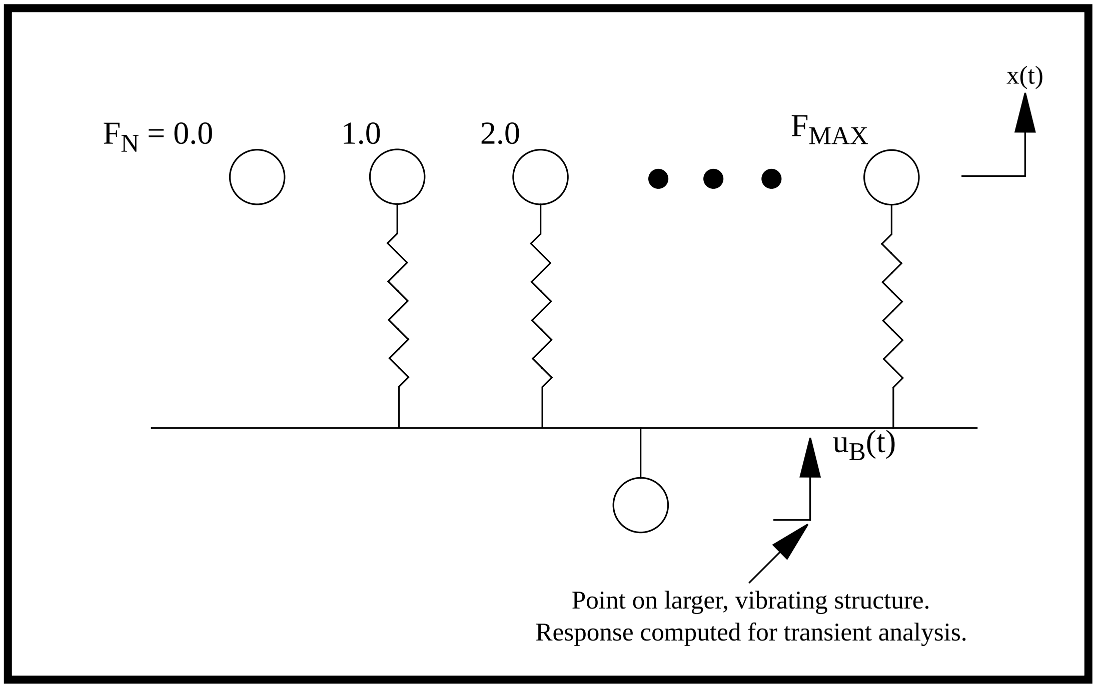
Objectives:

- Generate the shock spectrum.
- Submit the file for analysis in MSC.Nastran.
- Generate the shock spectrum using the direct transient method.

Model Description:

Define the shock response of the plate due to a 2.0 in/sec^2 sine pulse applied at the clamped edge. Use modes to a frequency of 1000 Hz with 3% critical damping. Use the SRSS option for model response summation.

Figure 9a.1-Model Description and Loading Diagram.



Suggested Exercise Steps:

- Generate the finite element representation of the model using (GRID) and (CMASS2) elements.
- Apply loading to mass, (TLOAD2) and (DAREA).
- Specify integration time steps (TSTEP).
- Define frequency and damping values for the SDOF oscillators (DTI).
- Specify damping information (FREQ) and natural frequency (FREQ1).
- Define the parameter to calculate shock spectrum.
  - PARAM, RESPECTRA, 0
- Generate an input file and submit it to the MSC.Nastran solver for direct transient analysis.
- Review the results.

```
ID SEMINAR,PROB9A
```

```
CEND
```

BEGIN BULK

1	2	3	4	5	6	7	8	9	10

1	2	3	4	5	6	7	8	9	10

ENDDATA

Generating an input file for MSC.Nastran Users:

MSC.Nastran users can generate an input file using the data from page 9-3 (Model Description). The result should be similar to the output below.

1. MSC.Nastran input file: **prob9a.dat**.

```
ID SEMINAR, PROB9a
SOL 109
TIME 30
CEND
TITLE= TRANSIENT RESPONSE
SUBTITLE= USING DIRECT TRANSIENT METHOD
LABEL= SHOCK SPECTRUM CALCULATION
ECHO= UNSORTED
SPC= 100
SET 111= 3000
DISPLACEMENT (SORT2)= 111 $ AT LEAST DISP AND VEL MUST APPEAR
VELOCITY (SORT2)= 111
ACCELERATION ()= 111
DLOAD= 500
TSTEP= 100
$
OUTPUT (XYPLOT)
$
$ SHOCK RESPONSE IS ONLY AVAILABLE IN PLOT OR PUNCH OUTPUT. THEREFORE,
$ THE ‘OUTPUT(XYPLOT)’ SECTION OF THE CASE CONTROL MUST BE USED.
$
XGRID=YES
YGRID=YES
XYPLOT ACCE / 3000(T1)
XLOG= YES
YLOG= YES
$
$ RELATIVE SHOCK RESPONSES ARE CONTAINED IN THE IMAGINARY/PHASE
$ COMPONENTS OF THE OUTPUT
$ ABSOLUTE SHOCK RESPONSES ARE CONTAINED IN THE REAL/MAGNITUDE
$ COMPONENTS OF THE OUTPUT
$
XTITLE= FREQUENCY (CYCLES/SEC)
YTITLE= RELATIVE DISPLACEMENT
XYPLOT DISP SPECTRAL 1 / 3000 (T1IP)
YTITLE= RELATIVE VELOCITY
XYPLOT VELOCITY SPECTRAL 1 / 3000 (T1IP)
YTITLE= ABSOLUTE ACCELERATION
XYPLOT ACCELERATION SPECTRAL 1 / 3000 (T1RM)
$
```

```
$ PUNCH SHOCK SPECTRUM FOR LATER USE
$
XYPUNCH ACCELERATION SPECTRAL 1 / 3000(T1RM)
$
BEGIN BULK
$
$ DEFINE GRID POINT
$
GRID, 3000, ,0.,0.,0., ,23456
$
$ DEFINE MASS
$
CMASS2, 100, 1.0, 3000, 1
$
$ APPLY LOADING TO MASS
$
TLOAD2, 500, 600, , 0, 0., 0.004, 250., -90.
$
DAREA, 600, 3000, 1, 1.
$
$ SPECIFY INTEGRATION TIME STEPS
$
TSTEP, 100, 100, 4.0E-4, 1
$
$ PARAMETER TO CALCULATE SHOCK SPECTRUM
$
PARAM, RSPECTRA, 0
$
$ SPECIFY FREQUENCY AND DAMPING VALUES FOR
$ THE SDOF OSCILLATORS AT GRID 3000
$
DTI, SPSEL, 0
DTI, SPSEL, 1, 111, 222, 3000
$ 1= SUBCASE... 111= DAMPING... 222= FREQUENCIES... 3000= GRID NUMBER
$
$ DAMPING INFORMATION FOR OSCILLATORS
$
FREQ, 111, 0., 0.02, 0.04
$
$ NATURAL FREQUENCIES OF OSCILLATORS
$
FREQ1, 222, 20., 20., 49
$
ENDDATA
```

Submitting the input file:

2. Submit the input file to MSC.Nastran for analysis.

   To submit the MSC.Nastran **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran prob9a scr=yes**. Monitor the run using the UNIX **ps** command.

3. When the run is completed, use **plotps** utility to create a postscript file, **prob9a.ps**, from the binary plot file **prob9a.plt**. The nonlinear force and displacement plots are shown on the following pages.

4. When the run is completed, edit the **prob9a.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing WARNING messages indicate modeling errors.

Comparison of Results

5. Compare the plot made from the exercise with the plots on the following pages.

Figure 9a.2

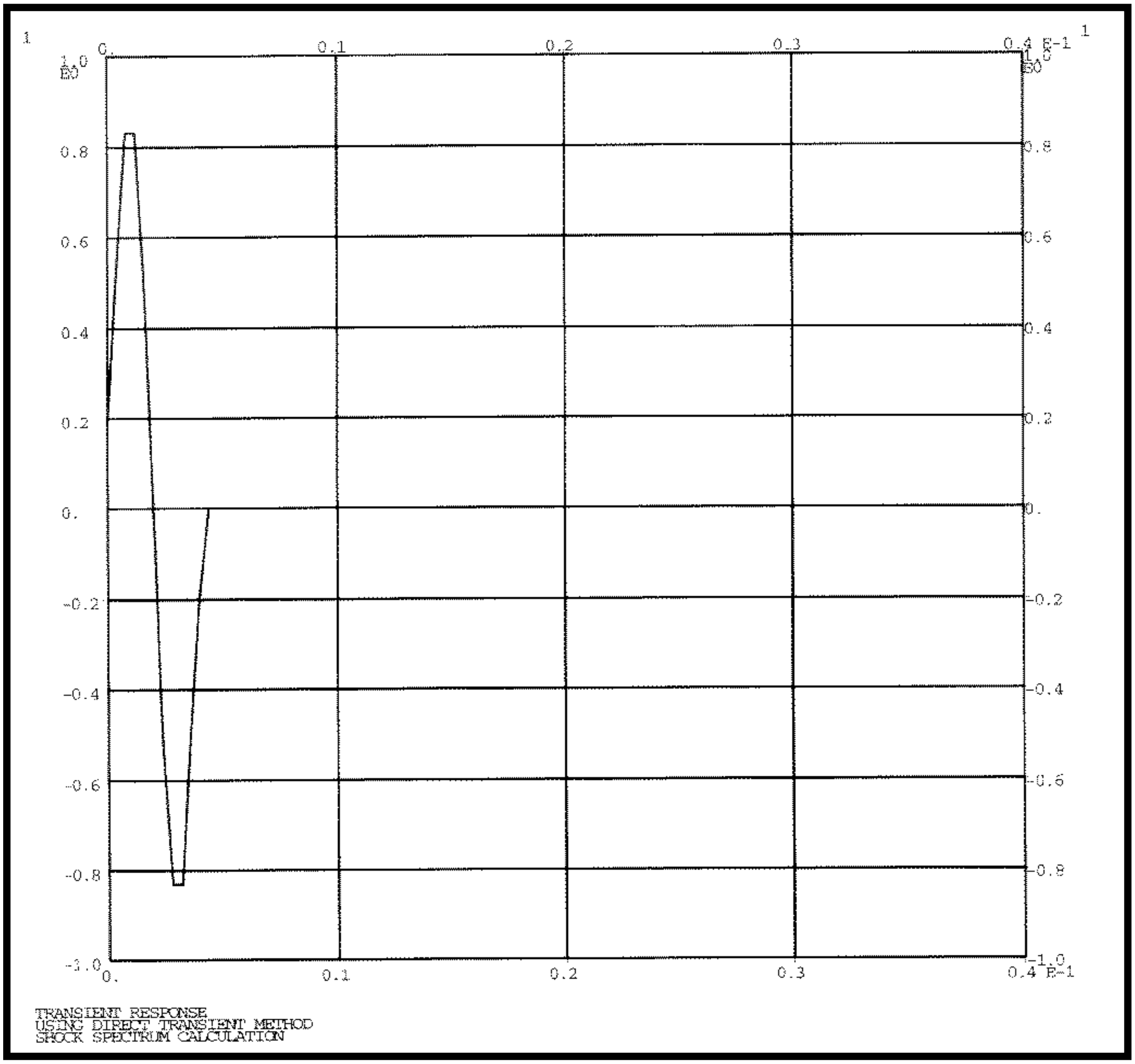


Figure 9a.3

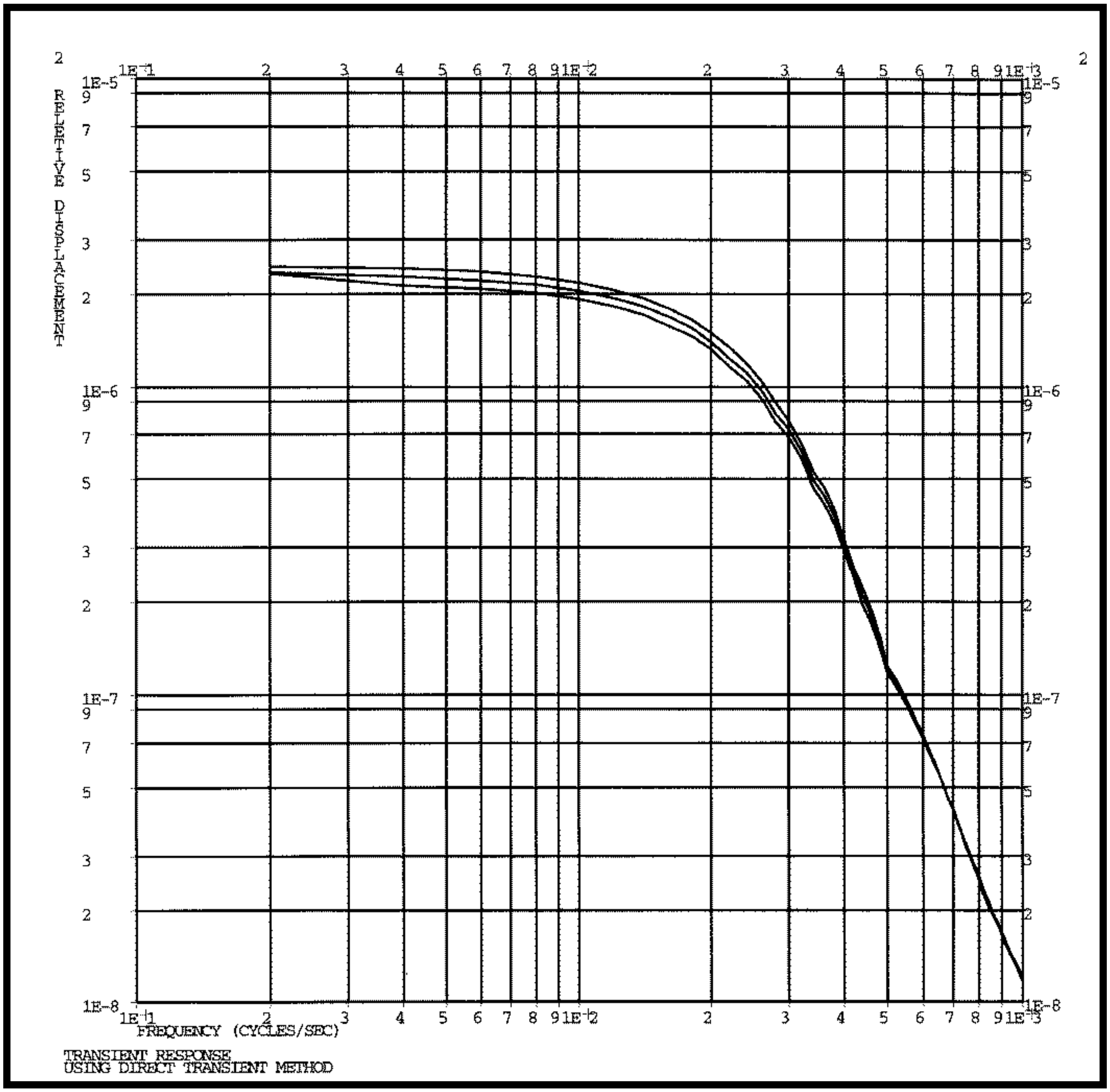

RELATIVE DISPLACEMENT
FREQUENCY (CYCLES/SEC)
TRANSIENT RESPONSE
USING DIRECT TRANSIENT METHOD

Figure 9a.4

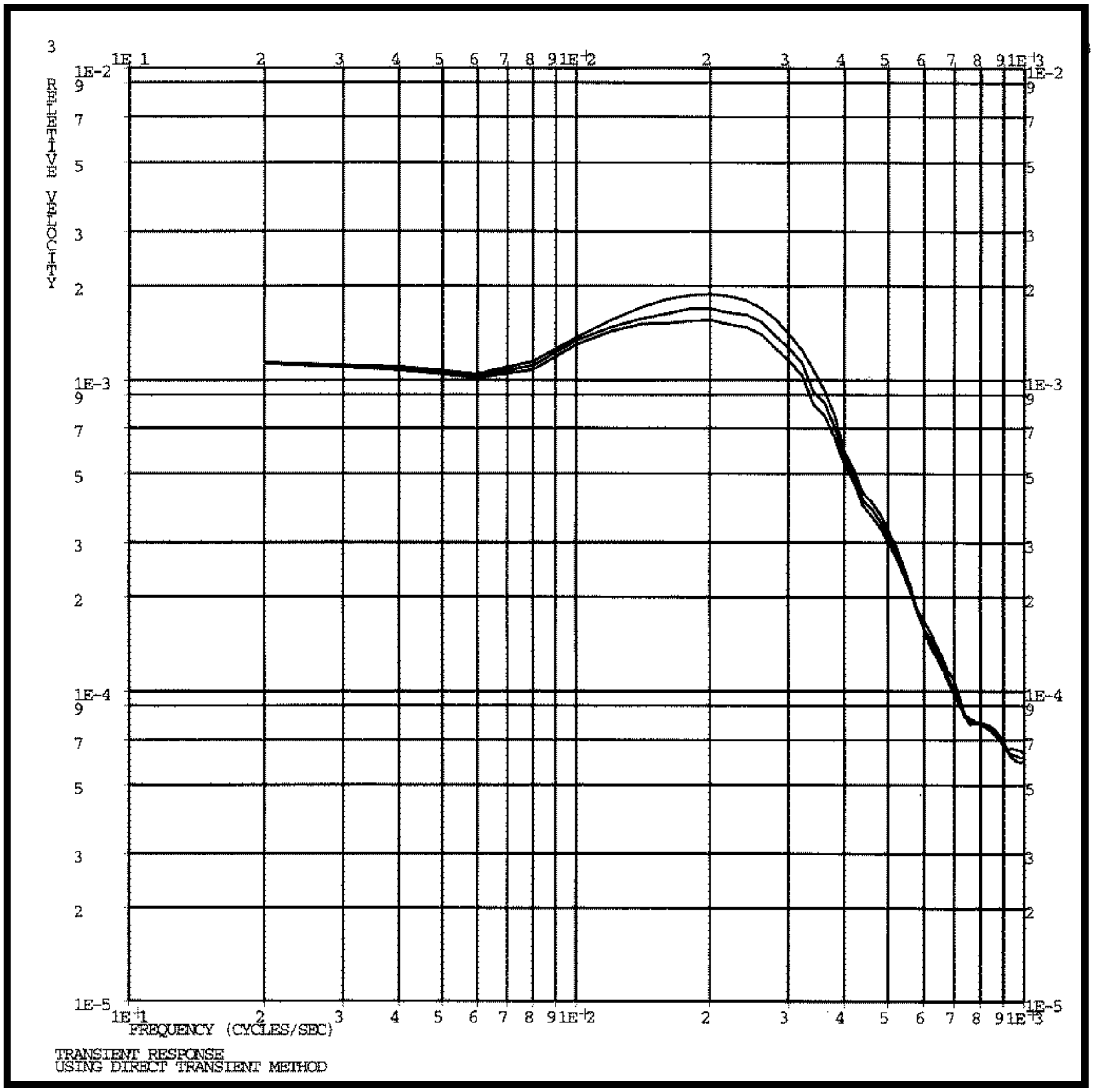
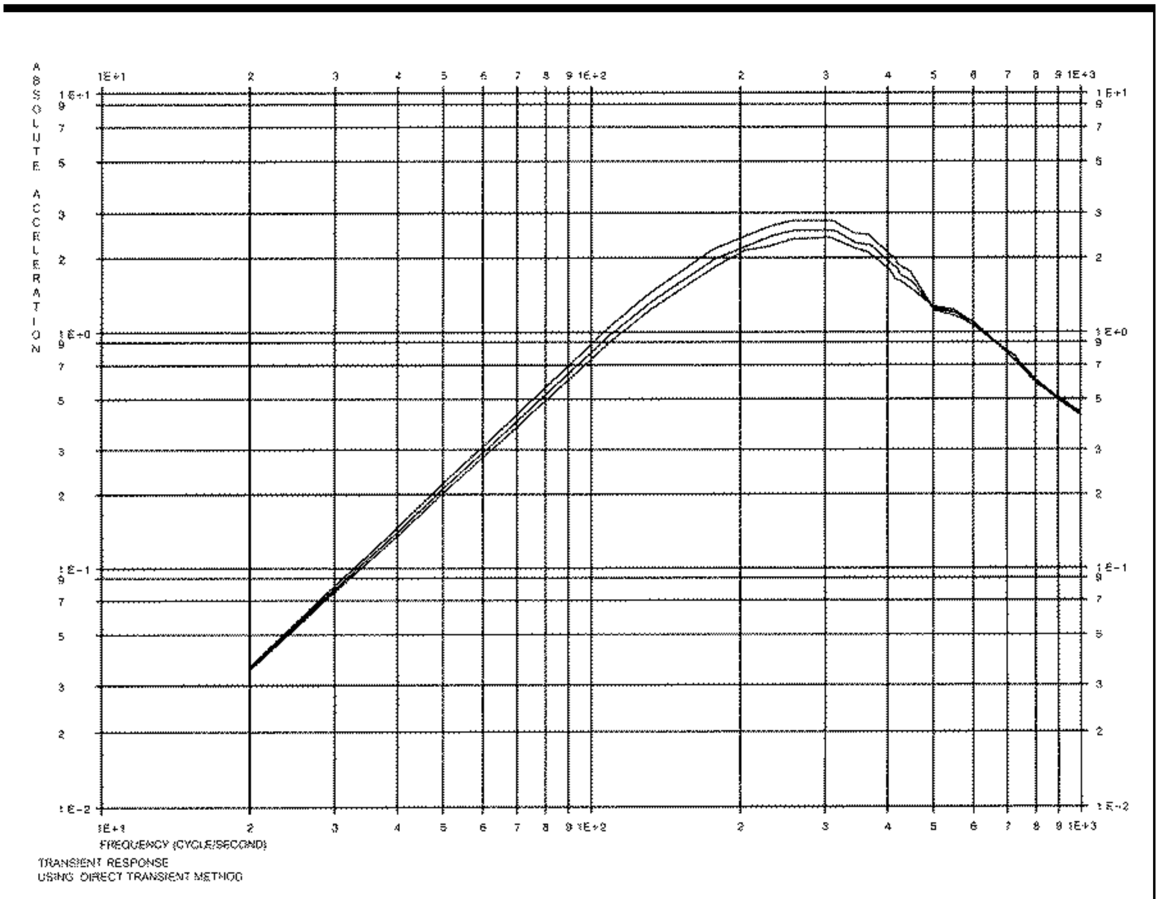

RELATIVE VELOCITY
FREQUENCY (CYCLES/SEC)
TRANSIENT RESPONSE
USING DIRECT TRANSIENT METHOD

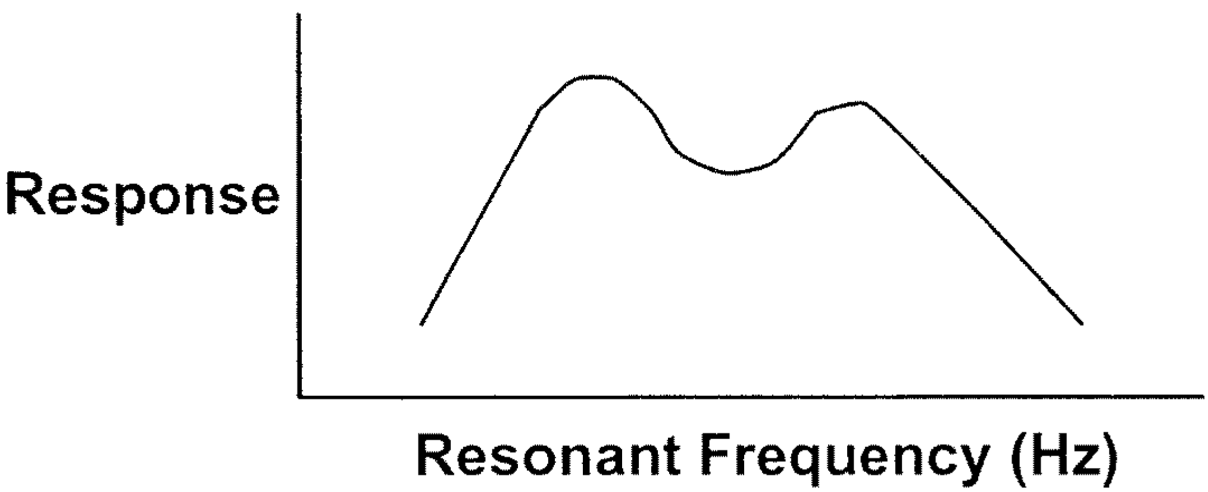
Figure 9a.5



ABSOLUTE ACCELERATION
FREQUENCY (CYCLE/SECOND)
TRANSIENT RESPONSE
USING DIRECT TRANSIENT METHOD

WORKSHOP 9b

Response Spectra (cont.)



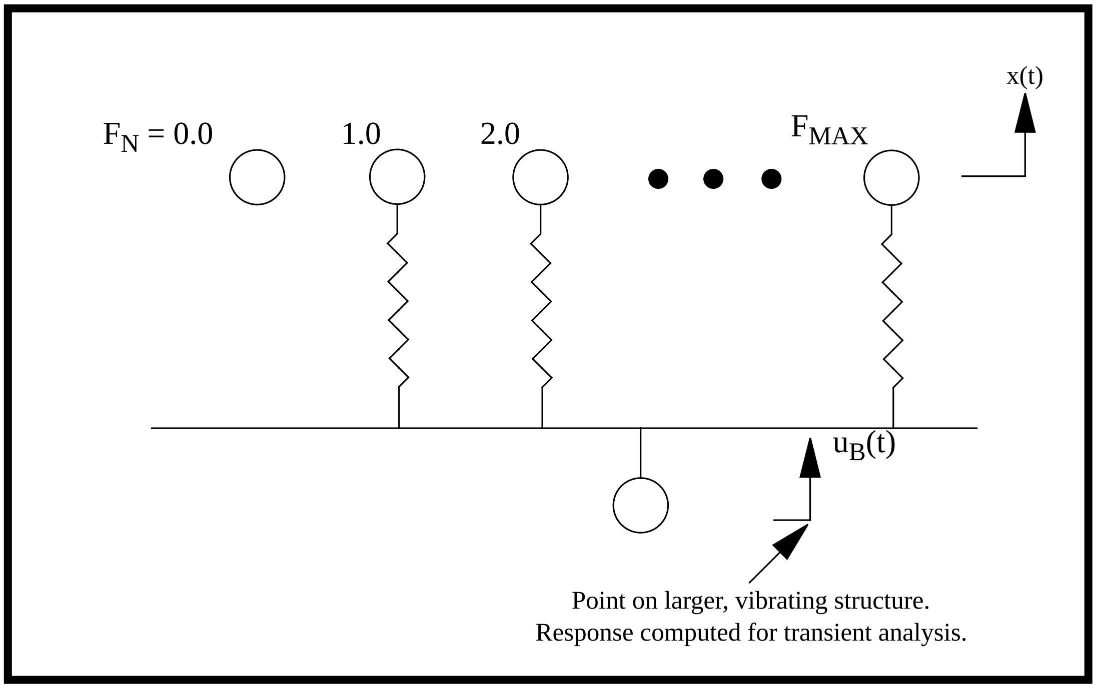
Objectives:

- Apply the shock spectrum.
- Submit the file for analysis in MSC.Nastran.
- Calculate the shock response using SOL 103.

Model Description:

Define the shock response of the plate due to a 2.0 in/sec^2 sine pulse applied at the clamped edge. Use modes to a frequency of 1000 Hz with 3% critical damping. Use the SRSS option for model response summation.

Figure 9b.1-Model Description and Loading Diagram



Suggested Exercise Steps:

- Reference a previously created dynamic math model, **plate.bdf**, by using the INCLUDE statement.
- Modify boundary conditions for clamped modes.
- Place big foundation mass (BFM) at base to simulate ‘clamped’ modes (CMASS2).
- RBE mass to remaining base point (RBE2).
- Identify excitation DOFs (SUPORT).
- Specify damping table (TABDMP1).
- Specify shock spectrum to be used (DLOAD).
- Specify shock tables (DTI).
- Insert punch output for shock spectrum calculation.
- Specify the appropriate parameters.
  - PARAM, SCRSPEC, 0
  - PARAM, OPTION, SRSS
  - PARAM, LFREQ, 0.1
  - PARAM, HFREQ, 1000
- Generate an input file and submit it to the MSC.Nastran solver (SOL103).
- Review the results.

```
ID SEMINAR,PROB9B
CEND
```

BEGIN BULK

1	2	3	4	5	6	7	8	9	10

1	2	3	4	5	6	7	8	9	10

ENDDATA

Generating an input file for MSC.Nastran Users:

MSC.Nastran users can generate an input file using the data from page 9-3 (Model Description). The result should be similar to the output below.

1. MSC.Nastran input file: **prob9b.dat**.

```
ID SEMINAR, PROB9B
SOL 103
TIME 30
CEND
TITLE= RESPONSE SPECTRUM ANALYSIS
SUBTITLE= USING CALCULATED SHOCK RESPONSE
LABEL= SHOCK WILL BE INPUT IN Z DIRECTION
ECHO= UNSORTED
SET 111= ALL
DISPLACEMENT= 111
SPC= 200
SUBCASE 1
METHOD= 100
SDAMP= 200
DLOAD= 500
$
BEGIN BULK
$
$ PLATE MODEL DESCRIBED IN NORMAL MODES EXAMPLE
$
INCLUDE 'plate.bdf'
PARAM,COUPMASS,1
PARAM,WTMASS,0.00259
$
$ BOUNDARY CONDITIONS FOR `CLAMPED' MODES
$
SPC1, 200, 1245, 1, 12, 23, 34, 45
$
$ PLACE BIG FOUNDATION MASS (BFM) AT BASE
$ TO STIMULATE `CLAMPED' MODES
$
CMASS2, 110, 1000., 23, 3
$
$ RBE MASS TO REMAINING BASE POINTS
$
RBE2, 101, 23, 3, 1, 12, 34, 45
```

```
$
$ SUPPORT CARD TO IDENTIFY EXCITATION DOFS
$
SUPORT, 23, 3
$
$ EIGENVALUE EXTRACTION
$ MUST BE MASS NORMALIZED (DEFAULT)
$
EIGR, 100, MGIV, 0., 1000.
$
$ TABLE TO SPECIFY DAMPING FOR USE IN THE ANALYSIS
$
TABDMP1, 200, CRIT,
, 0., 0.03, 1000., 0.03, ENDT
$
$ SPECIFICATION OF SHOCK SPECTRUM TO BE USED
$
DLOAD, 500, 1.0, 2.0, 1
$
$ DLOAD, ID, OVERALL SCALE, SCALE FOR R-SET DOF# 1, SHOCK TABLE FOR DOF# 1,
$ SCALE FOR R-SET DOF# 2, SHOCK TABLE FOR DOF# 2, ETC.
$
$ SELECT SHOCK RESPONSE CALCULATION
$
PARAM, SCRSPEC, 0
$
$ SELECT SUMMATION OPTION
$
PARAM, OPTION, SRSS
$
$ MODAL FREQUENCY RANGE CAN BE SELECTED USING
PARAM, LFREQ, 0.1
PARAM, HFREQ, 1000.
$
$ SPECIFICATION FOR SHOCK TABLES
$
DTI, SPECSEL, 0
DTI, SPECSEL, 1, , A, 2, 0., 3, 0.02,
, 4, 0.04, ENDREC
$
$ DTI, SPECSEL, SHOCK TABLE NUMBER, , [(A)CCELERATION, (V)ELOCITY, OR (D)ISP],
$ TABLED1 POINTER, DAMPING FOR TABLE, ETC.
$
```

```
$ PUNCH OUTPUT FOR SHOCK SPECTRUM CALCULATION
$
$ ACCE    4       3000            3       1
$       0.000000E+00
$
TABLED1 2
        20.     .038683 40.     .152539 60.     .33511  80.     .576059
        100.    .862049 120.    1.17619 140.    1.50169 160.    1.82018
        180.    2.11404 200.    2.36801 220.    2.56617 240.    2.70027
        260.    2.76275 280.    2.75073 300.    2.74632 320.    2.61887
        340.    2.4218  360.    2.39068 380.    2.24931 400.    2.02296
        420.    1.78538 440.    1.70355 460.    1.57056 480.    1.40493
        500.    1.22608 520.    1.20483 540.    1.17631 560.    1.14097
        580.    1.10048 600.    1.05582 620.    1.00818 640.    .958761
        660.    .908725 680.    .859158 700.    .827667 720.    .782127
        740.    .728996 760.    .694088 780.    .668602 800.    .635044
        820.    .598496 840.    .571831 860.    .563072 880.    .550499
        900.    .528854 920.    .509281 940.    .500534 960.    .498016
        980.    .488793 1000.   .468321 ENDT
$ACCE    4       3000            3       52
$   2.0000000E-02
TABLED1 3
        20.     .037708 40.     .143365 60.     .314936 80.     .541342
        100.    .80976  120.    1.10506 140.    1.40671 160.    1.69567
        180.    1.98167 200.    2.22217 220.    2.35249 240.    2.53055
        260.    2.56231 280.    2.55577 300.    2.58668 320.    2.45921
        340.    2.29411 360.    2.25956 380.    2.12901 400.    1.92605
        420.    1.68656 440.    1.61355 460.    1.4968  480.    1.35263
        500.    1.19796 520.    1.17707 540.    1.14947 560.    1.11613
        580.    1.07807 600.    1.03637 620.    .992124 640.    .946383
        660.    .900171 680.    .854434 700.    .810016 720.    .767647
        740.    .727923 760.    .691288 780.    .658039 800.    .628311
        820.    .602091 840.    .579207 860.    .559362 880.    .542128
        900.    .526973 920.    .51329  940.    .500403 960.    .487602
        980.    .474171 1000.   .459408 ENDT
$ACCE    4       3000            3       103
$   4.0000000E-02
TABLED1 4
        20.     .039336 40.     .137673 60.     .297382 80.     .511244
        100.    .764891 120.    1.04406 140.    1.31588 160.    1.58461
        180.    1.85678 200.    2.10175 220.    2.19165 240.    2.3921
        260.    2.39929 280.    2.42782 300.    2.44263 320.    2.317
        340.    2.17923 360.    2.14283 380.    2.0227  400.    1.8407
        420.    1.62279 440.    1.53417 460.    1.43168 480.    1.30597
        500.    1.17212 520.    1.15165 540.    1.12513 560.    1.09349
```

```
        580.    1.05768 600.    1.01868 620.    .977462 640.    .934986
        660.    .892143 680.    .849752 700.    .808538 720.    .769114
        740.    .731968 760.    .69746  780.    .665814 800.    .637115
        820.    .611319 840.    .588261 860.    .567655 880.    .549125
        900.    .532205 920.    .516369 940.    .501047 960.    .485644
        980.    .469568 1000.   .452243 ENDT
$
ENDDATA
```

Submitting the input file for analysis:

2. Submit the input file to MSC.Nastran for analysis.

   To submit the MSC.Nastran **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran prob9b scr=yes**. Monitor the run using the UNIX **ps** command.

3. When the run is completed, edit the **prob9b.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing WARNING messages indicate modeling errors.

Comparison of Results

4. Compare the results obtained in the **.f06** file with the results on the following page:

```
*** USER INFORMATION MESSAGE 3035 FOR DATA BLOCK   KLR

   SUPPORT PT.NO.        EPSILON        STRAIN   ENERGY     EPSILONS LARGER THAN 0.001 ARE FLAGGED WITH ASTI
               1     9.5369362E-15       2.2118911E-09
    RESPONSE SPECTRUM ANALYSIS                                JULY   5, 1997  MSC.Nastran   1/23/97   PAGE
     USING CALCULATED SHOCK RESPONSE
  SHOCK WILL BE INPUT IN Z DIRECTION                         SUBCASE 1

*** USER    INFORMATION MESSAGE 4415,
           THE FOLLOWING A-SET DEGREES OF FREEDOM HAVE EITHER NULL MASSES OR NULL MASSES AND STIFFNESS.
    RESPONSE SPECTRUM ANALYSIS                                JULY   5, 1997  MSC.Nastran   1/23/97   PAGE
     USING CALCULATED SHOCK RESPONSE
     SHOCK WILL BE INPUT IN Z DIRECTION                                                                SUB

                                          R E A L   E I G E N V A L U E S
  MODE    EXTRACTION     EIGENVALUE           RADIANS              CYCLES           GENERALIZED        GENERALI
   NO.       ORDER                                                                      MASS             STIFFNE
      1      101        0.0                 0.0                 0.0                1.000000E+00       0.0
      2      102        7.058213E+05        8.401317E+02        1.337111E+02       1.000000E+00       7.058213E
      3      103        1.878432E+07        4.334088E+03        6.897916E+02       1.000000E+00       1.878432E
      4      104        2.811620E+07        5.302471E+03        8.439145E+02       1.000000E+00       2.811620E
      5      105        1.931709E+08        1.389859E+04        2.212030E+03       0.0                0.0
      6        1        2.234434E+08        1.494802E+04        2.379052E+03       0.0                0.0
      7      106        2.328846E+08        1.526056E+04        2.428793E+03       0.0                0.0
      8      107        6.845925E+08        2.616472E+04        4.164244E+03       0.0                0.0
      9      108        9.602310E+08        3.098759E+04        4.931829E+03       0.0                0.0
     10      109        1.365612E+09        3.695418E+04        5.881441E+03       0.0                0.0
^^^ USER INFORMATION MESSAGE 9047 (POSTREIG) -  SCALED RESPONSE SPECTRA FOR RESIDUAL STRUCTURE ONLY
   RESPONSE SPECTRUM ANALYSIS                                  JULY   5, 1997  MSC.Nastran   1/23/97   PAGI
     USING CALCULATED SHOCK RESPONSE
     SHOCK WILL BE INPUT IN Z DIRECTION                                                                SUB
      MATRIX FN      (GINO NAME 101 ) IS A DB  PREC          1 COLUMN X          3 ROW RECTANG  MATRIX.
COLUMN      1      ROWS      1 THRU      3     --------------------------------------------------
ROW
     1)    1.337111500777D+02  6.897918198043D+02  8.439147827213D+02
THE NUMBER OF NON-ZERO TERMS IN THE DENSEST COLUMN =       3
THE DENSITY OF THIS MATRIX IS 100.00 PERCENT.
```

```
RESPONSE SPECTRUM ANALYSIS                                   JULY   5, 1997  MSC.Nastran   1/23/97   PAGE
USING CALCULATED SHOCK RESPONSE
SHOCK WILL BE INPUT IN Z DIRECTION                                                                    SUBCAS
           PSIT
POINT           VALUE     POINT      VALUE     POINT       VALUE     POINT       VALUE     POINT           VALU

COLUMN        1
    23 T3 -2.11560E-02
COLUMN        2
    23 T3 -4.40398E-16
COLUMN        3
    23 T3  1.18599E-02
RESPONSE SPECTRUM ANALYSIS                                   JULY   5, 1997  MSC.Nastran   1/23/97   PAGE
USING CALCULATED SHOCK RESPONSE
SHOCK WILL BE INPUT IN Z DIRECTION                                                                    SUBCAS
            U S E T   D E F I N I T I O N   T A B L E   ( I N T E R N A L   S E Q U E N C E ,   R O W   S O R
                                               R    DISPLACEMENT SET
          -1-        -2-        -3-        -4-        -5-        -6-        -7-        -8-        -9-        -

     1=       23-3
  SCALED SPECTRAL RESPONSE, SRSS OPTION, DLOAD =        500  CLOSE =       1.00

RESPONSE SPECTRUM ANALYSIS                                   JULY   5, 1997  MSC.Nastran   1/23/97   PAGE
USING CALCULATED SHOCK RESPONSE
SHOCK WILL BE INPUT IN Z DIRECTION                                                                    SUBCAS
      MATRIX UHVR     (GINO NAME 101 ) IS A REAL              3 COLUMN X          3 ROW SQUARE   MATRIX.
COLUMN      1      ROWS      1 THRU      3     --------------------------------------------------
ROW
     1)    7.6201E-08  0.0000E+00  4.8912E-10
COLUMN      2      ROWS      1 THRU      3     --------------------------------------------------
ROW
     1)    6.4019E-05  0.0000E+00  2.5935E-06
COLUMN      3      ROWS      1 THRU      3     --------------------------------------------------
ROW
     1)    5.3784E-02  0.0000E+00  1.3752E-02
THE NUMBER OF NON-ZERO TERMS IN THE DENSEST COLUMN =       2
THE DENSITY OF THIS MATRIX IS  66.67 PERCENT.
RESPONSE SPECTRUM ANALYSIS                                   JULY   5, 1997  MSC.Nastran   1/23/97   PAGE
```

USING CALCULATED SHOCK RESPONSE
SHOCK WILL BE INPUT IN Z DIRECTION SUBCASE 1
TIME = 0.000000E+00

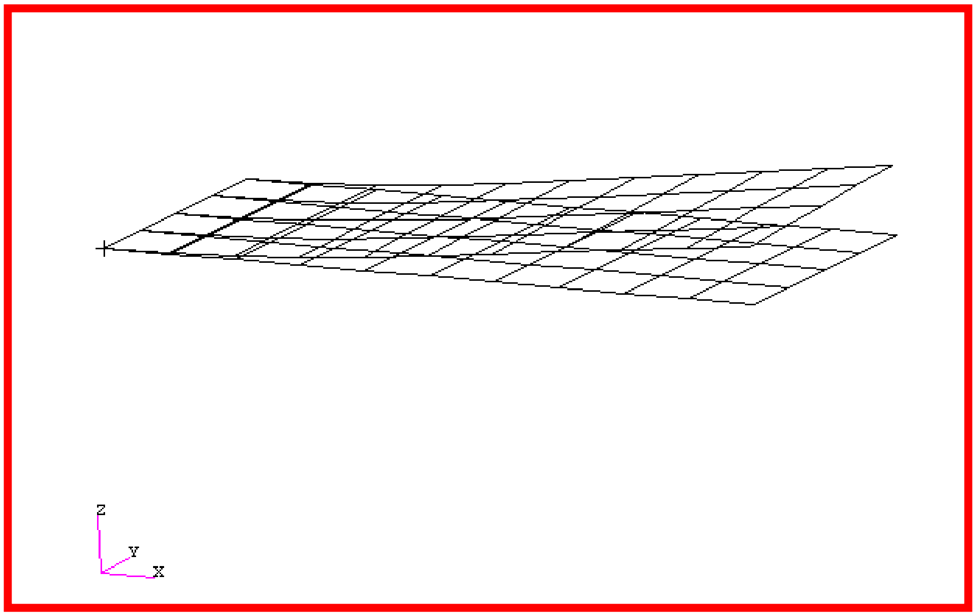
D I S P L A C E M E N T V E C T O R

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1	G	0.0	0.0	6.222643E-10	0.0	0.0	0.0
2	G	0.0	0.0	7.374101E-08	8.308994E-08	3.059527E-07	0.0
3	G	0.0	0.0	3.173752E-07	1.214202E-07	6.616859E-07	0.0
4	G	0.0	0.0	7.194814E-07	1.167081E-07	9.376687E-07	0.0
5	G	0.0	0.0	1.246589E-06	1.039592E-07	1.160147E-06	0.0
6	G	0.0	0.0	1.870825E-06	8.258849E-08	1.326412E-06	0.0
7	G	0.0	0.0	2.565942E-06	6.149954E-08	1.444516E-06	0.0
8	G	0.0	0.0	3.308983E-06	4.203442E-08	1.519611E-06	0.0
9	G	0.0	0.0	4.080410E-06	2.671514E-08	1.559985E-06	0.0
10	G	0.0	0.0	4.865109E-06	1.691765E-08	1.575028E-06	0.0
11	G	0.0	0.0	5.653596E-06	1.301237E-08	1.577945E-06	0.0
12	G	0.0	0.0	6.222643E-10	0.0	0.0	0.0
13	G	0.0	0.0	9.569825E-08	1.430303E-08	3.703413E-07	0.0
14	G	0.0	0.0	3.577567E-07	4.357941E-08	6.684602E-07	0.0
15	G	0.0	0.0	7.614155E-07	5.321706E-08	9.362424E-07	0.0
16	G	0.0	0.0	1.284521E-06	4.871796E-08	1.147647E-06	0.0
17	G	0.0	0.0	1.901587E-06	4.101944E-08	1.312728E-06	0.0
18	G	0.0	0.0	2.589113E-06	3.141624E-08	1.430590E-06	0.0
19	G	0.0	0.0	3.325147E-06	2.263403E-08	1.507938E-06	0.0
20	G	0.0	0.0	4.091029E-06	1.555984E-08	1.551409E-06	0.0
21	G	0.0	0.0	4.872190E-06	1.101371E-08	1.570698E-06	0.0
22	G	0.0	0.0	5.659176E-06	8.953050E-09	1.576591E-06	0.0
23	G	0.0	0.0	6.222643E-10	0.0	0.0	0.0
24	G	0.0	0.0	9.883527E-08	0.0	3.798185E-07	0.0
25	G	0.0	0.0	3.680177E-07	0.0	6.836093E-07	0.0
26	G	0.0	0.0	7.745986E-07	0.0	9.334775E-07	0.0
27	G	0.0	0.0	1.296574E-06	0.0	1.145700E-06	0.0
28	G	0.0	0.0	1.911807E-06	0.0	1.307858E-06	0.0
29	G	0.0	0.0	2.596946E-06	0.0	1.426281E-06	0.0
30	G	0.0	0.0	3.330809E-06	0.0	1.503975E-06	0.0

.
.

WORKSHOP 10

Random Analysis

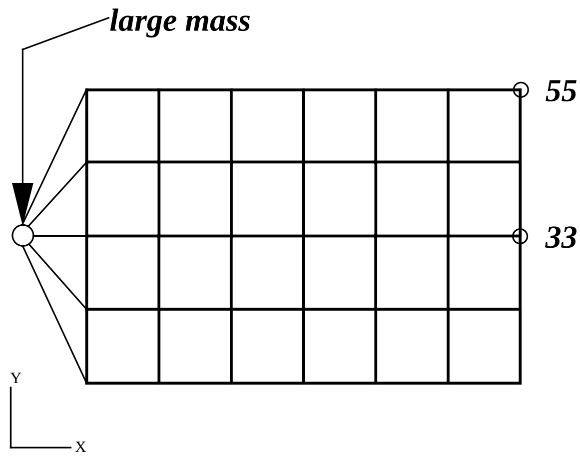


Objectives:

- Define a frequency-varying excitation.
- Define load set power spectral density functions.
- Produce a MSC.Nastran input file from a dynamic math model created in Workshop 1.
- Submit the file for random analysis in MSC.Nastran.
- Compute nodal displacements for desired frequency domain.

Model Description:

For the plate model, enforce a base motion in the z-direction described by the following power spectral density, (PSD).



Autospectra of the Base Excitation	
Frequency (Hz)	**G^2/Hz**
20	**0.1**
30	**1**
100	**1**
500	**0.1**
1000	**0.1**

Use the modal method with a large mass attached to the edge with an RBE2 entry.

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

Determine:

- The response displacement and acceleration PSD at the drive location, (the large mass).
- The displacement PSD at the corner and center of the free edge, (Grids 33 and 55).
- Use modal solution.
- Assume a constant critical damping ratio of 3% across the whole frequency range.

Suggested Exercise Steps:

- Reference a previously created dynamic math model, **plate.bdf**, by using the INCLUDE statement.
- Attach the large mass to the edge of the plate (CONM2 and RBE2).
- Specify modal damping as a tabular function of natural frequency (TABDMP1).
- Define the frequency-varying tip load (DAREA and RLOAD2).
- Define a set of frequencies to be used in the solution (FREQ, FREQ1, and FREQ 4).
- Specify Spectral Density (RANDPS and TABRND1).
- Prepare the model for a direct transient analysis (SOL 111).
- Request acceleration responses at base, tip center, and opposite corner.
- Generate an input file and submit it to the MSC.Nastran solver for direct transient analysis.
- Review the results.

```
ID SEMINAR,PROB10
```

```
CEND
```

BEGIN BULK

1	2	3	4	5	6	7	8	9	10

1	2	3	4	5	6	7	8	9	10

ENDDATA

Generating an input file for MSC.Nastran Users:

MSC.Nastran users can generate an input file using the data from pages 10-3 (general model description). The result should be similar to the output below.

1. MSC.Nastran input file: **prob10.dat**.

```
ID SEMINAR, PROB10
SOL 111
TIME 30
CEND
TITLE= RANDOM ANALYSIS - BASE EXCITATION
SUBTITLE= USING THE MODAL METHOD WITH LANCZOS
ECHO= UNSORTED
SPC= 101
SET 111= 33, 55, 9999
ACCELERATION(SORT2, PHASE)= 111
METHOD= 100
FREQUENCY= 100
SDAMPING= 100
RANDOM= 100
DLOAD= 100
$
OUTPUT(XYPLOT)
XTGRID= YES
YTGRID= YES
XBGRID= YES
YBGRID= YES
YTLOG= YES
XTITLE= FREQUENCY
YTTITLE= ACCEL RESPONSE BASE, MAGNITUDE
YBTITLE= ACCEL RESPONSE AT BASE, PHASE
XYPLOT ACCEL RESPONSE / 9999 (T3RM, T3IP)
YTTITLE= ACCEL RESPONSE AT TIP CENTER, MAGNITUDE
YBTITLE= ACCEL RESPONSE AT TIP CENTER, PHASE
XYPLOT ACCEL RESPONSE / 33 (T3RM, T3IP)
YTTITLE= ACCEL RESPONSE AT OPPOSITE CORNER, MAGNITUDE
YBTITLE= ACCEL RESPONSE AT OPPOSITE CORNER, PHASE
XYPLOT ACCEL RESPONSE / 55 (T3RM, T3IP)
$
$ PLOT OUTPUT IS ONLY MEANS OF VIEWING PSD DATA
$
XGRID= YES
YGRID= YES
```

```
XLOG= YES
YLOG= YES
YTITLE= ACCEL P S D AT LOADED CORNER
XYPLOT ACCEL PSDF / 9999(T3)
YTITLE= ACCEL P S D AT TIP CENTER
XYPLOT ACCEL PSDF / 33(T3)
YTITLE= ACCEL P S D AT OPPOSITE CORNER
XYPLOT ACCEL PSDF / 55(T3)
$
BEGIN BULK
PARAM,COUPMASS,1
PARAM,WTMASS,0.00259
$
INCLUDE 'plate.bdf'
$
GRID, 9999, , 0., 0., 0.
$
RBE2, 101, 9999, 12345, 1, 12, 23, 34, 45
$
SPC1, 101, 12456, 9999
$
CONM2, 6000, 9999, , 1.0E8
$
$ EIGENVALUE EXTRACTION PARAMETERS
$
EIGRL, 100 , , 2000.
$
$ SPECIFY MODAL DAMPING
$
TABDMP1, 100, CRIT,
+, 0., .03, 10., .03, ENDT
$
$ POINT LOADING AT TIP CENTER
$
RLOAD2, 100, 600, , , 310
$
TABLED1, 310,
+, 10., 1., 1000., 1., ENDT
$
DAREA, 600, 9999, 3, 1.E8
$
$ SPECIFY FREQUENCY STEPS
$
```

```
FREQ,100,30.
FREQ1,100,20.,20.,50
FREQ4,100,20.,1000.,.03,5
$
$ SPECIFY SPECTRAL DENSITY
$
RANDPS, 100, 1, 1, 1., 0., 111
$
TABRND1, 111,LOG,LOG
+, 20., 0.1, 30., 1., 100., 1., 500., .1,
+, 1000., .1, ENDT
$
ENDDATA
```

Submitting the input file:

2. Submit the input file to MSC.Nastran for analysis.

   To submit the MSC.Nastran **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran prob10 scr=yes**. Monitor the run using the UNIX **ps** command.

3. When the run is completed, use **plotps** utility to create a postscript file, **prob10.ps**, from the binary plot file **prob10.plt**. The nonlinear force and displacement plots are shown on the following pages.

4. When the run is completed, edit the **prob10.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing WARNING messages indicate modeling errors.

Comparison of Results

5. Compare the plot made from the exercise with the plots on the following pages.

Figure 10.1

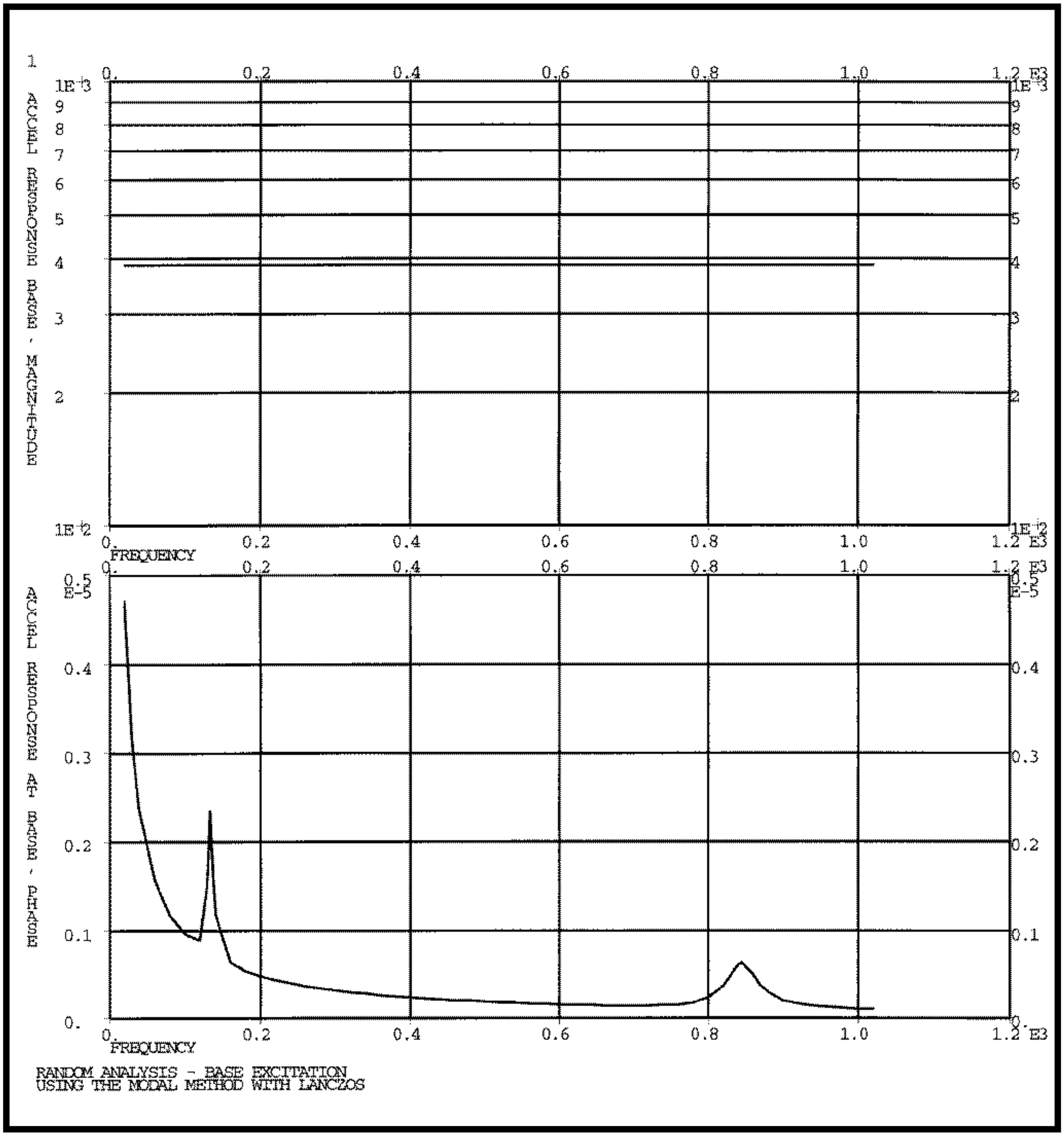

1
ACCEL RESPONSE BASE - MAGNITUDE
1E-3
9
8
7
6
5
4
3
2
1E-2
0.
0.2
0.4
0.6
0.8
1.0
1.2 E3
FREQUENCY
ACCEL RESPONSE AT BASE - PHASE
0.5 E-5
0.4
0.3
0.2
0.1
0.
RANDOM ANALYSIS - BASE EXCITATION
USING THE MODAL METHOD WITH LANCZOS

Figure 10.2

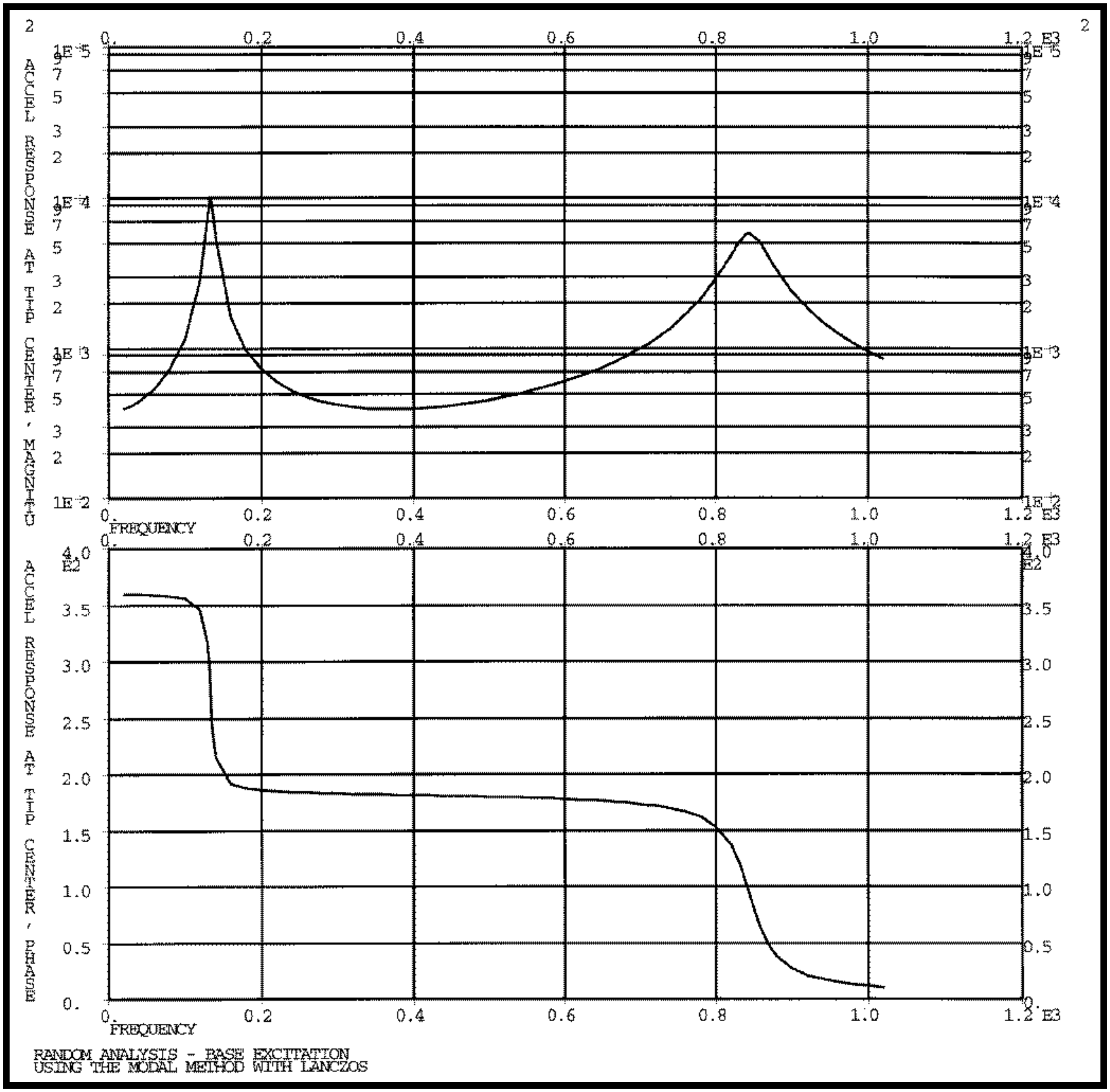

ACCEL RESPONSE AT TIP CENTER, MAGNITU
ACCEL RESPONSE AT TIP CENTER, PHASE
FREQUENCY
RANDOM ANALYSIS - BASE EXCITATION
USING THE MODAL METHOD WITH LANCZOS

Figure 10.3

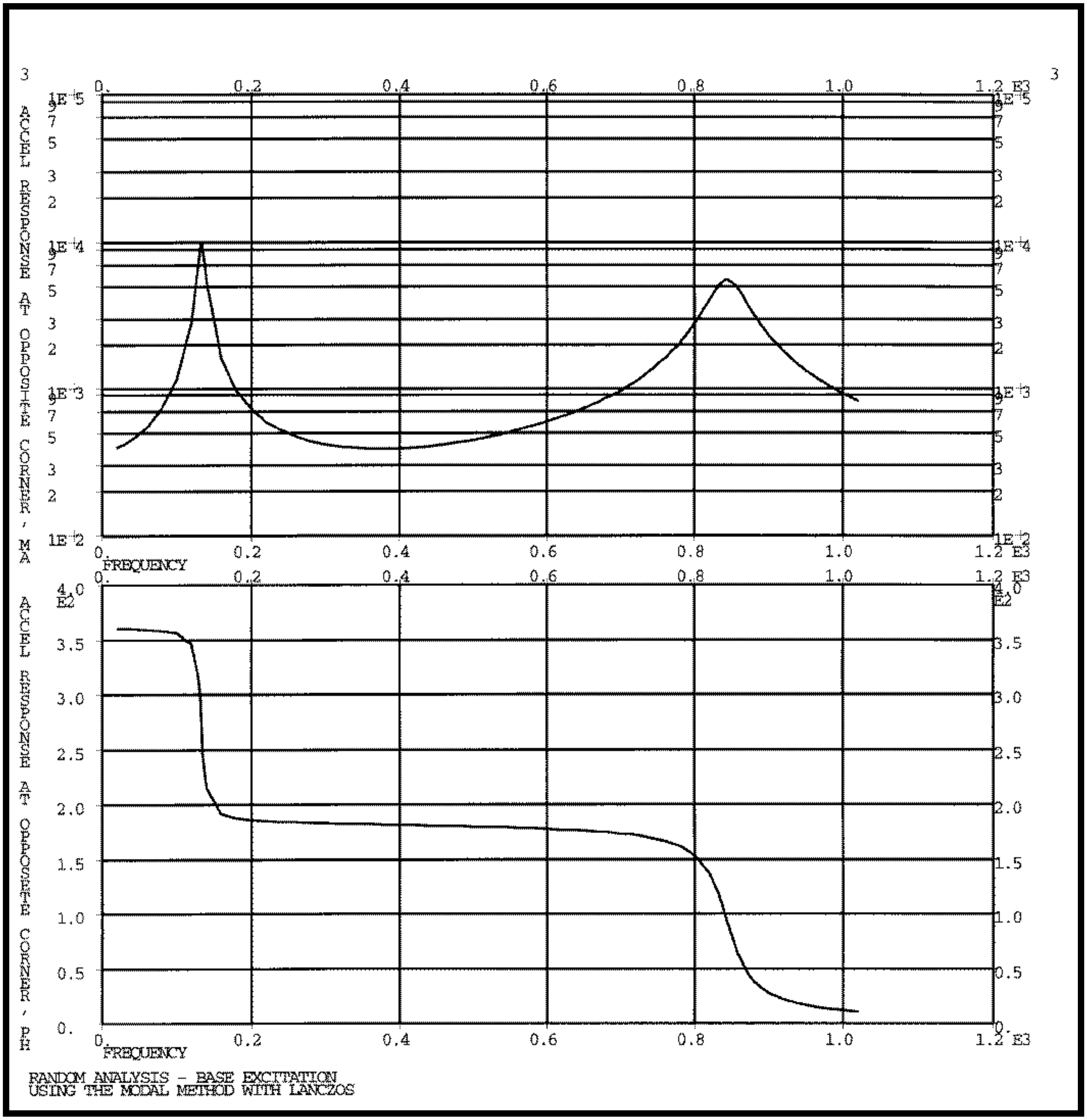
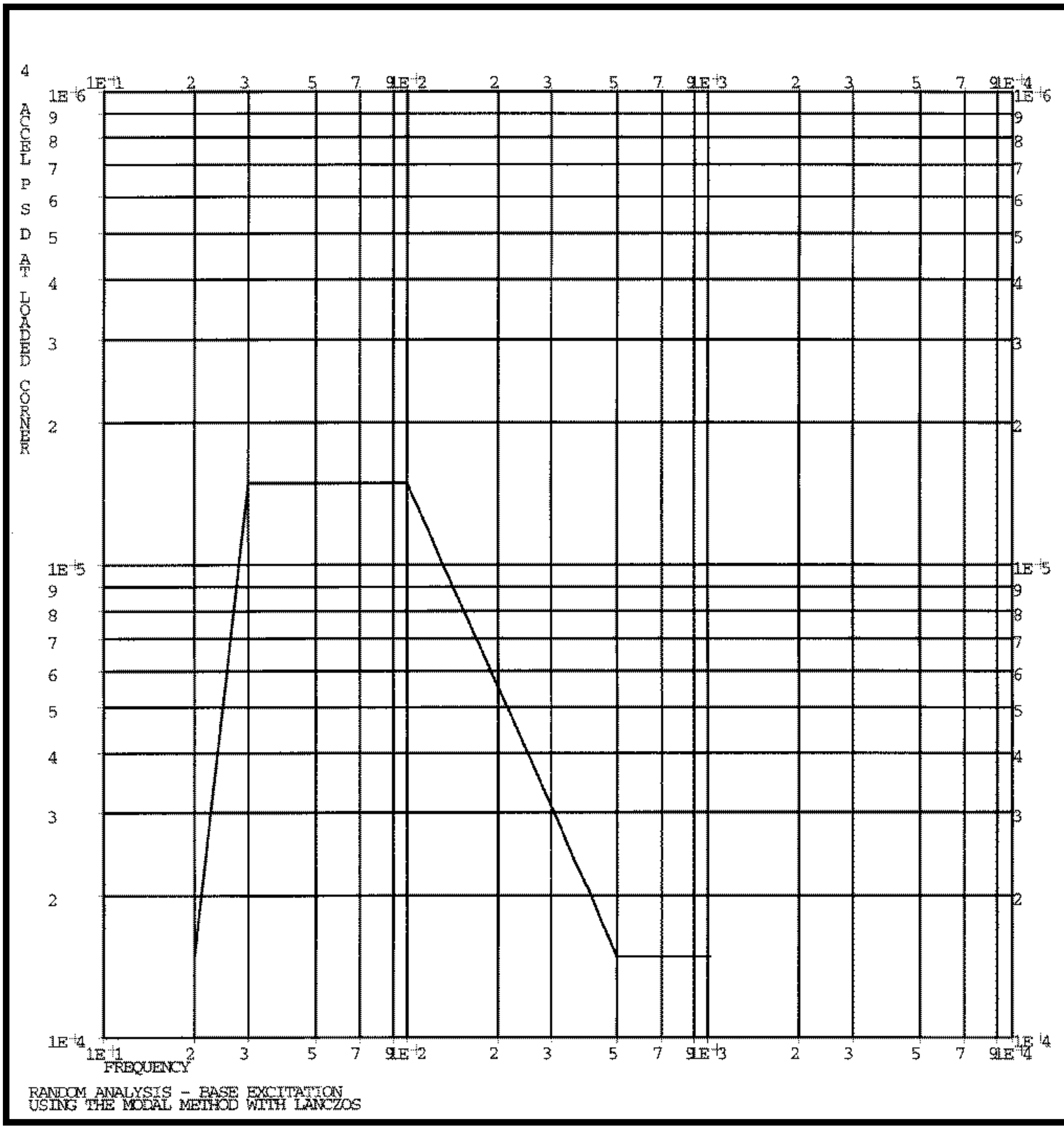
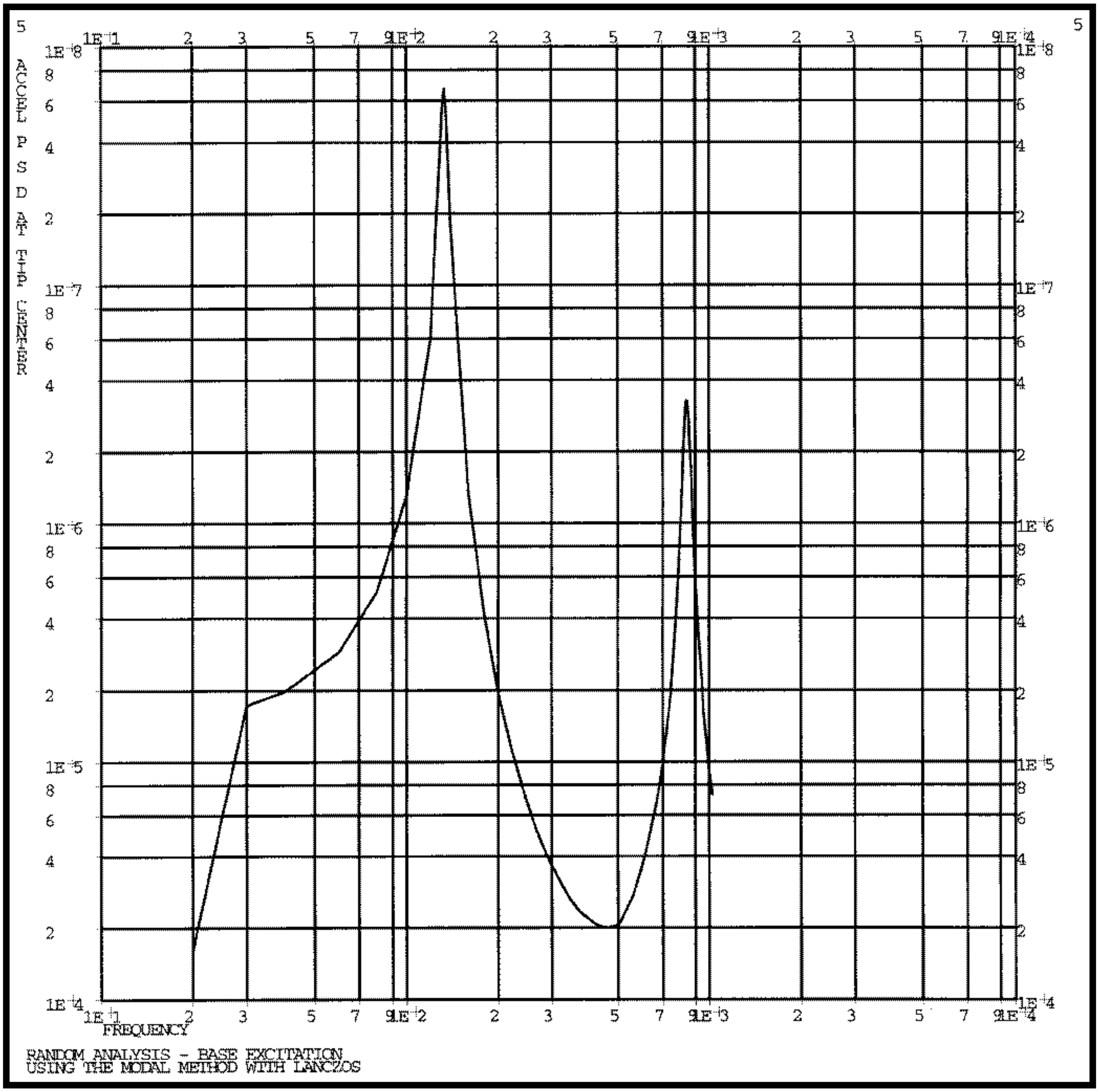

ACCEL RESPONSE AT OPPOSITE CORNER, MA
ACCEL RESPONSE AT OPPOSITE CORNER, PH
FREQUENCY
FREQUENCY
RANDOM ANALYSIS - BASE EXCITATION
USING THE MODAL METHOD WITH LANCZOS

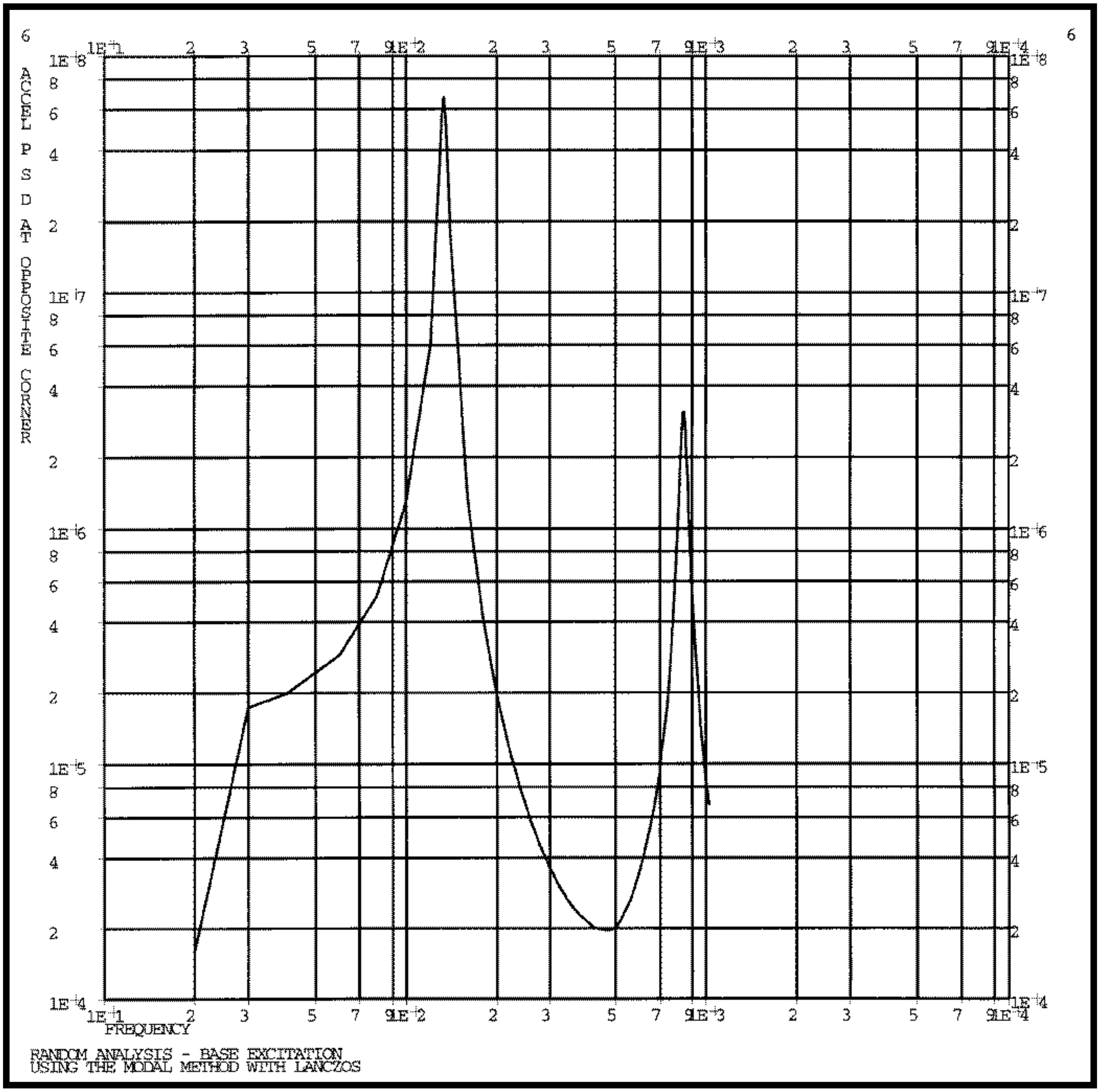
Figure 10.4



ACCEL PSD AT LOADED CORNER
FREQUENCY
RANDOM ANALYSIS - BASE EXCITATION
USING THE MODAL METHOD WITH LANCZOS

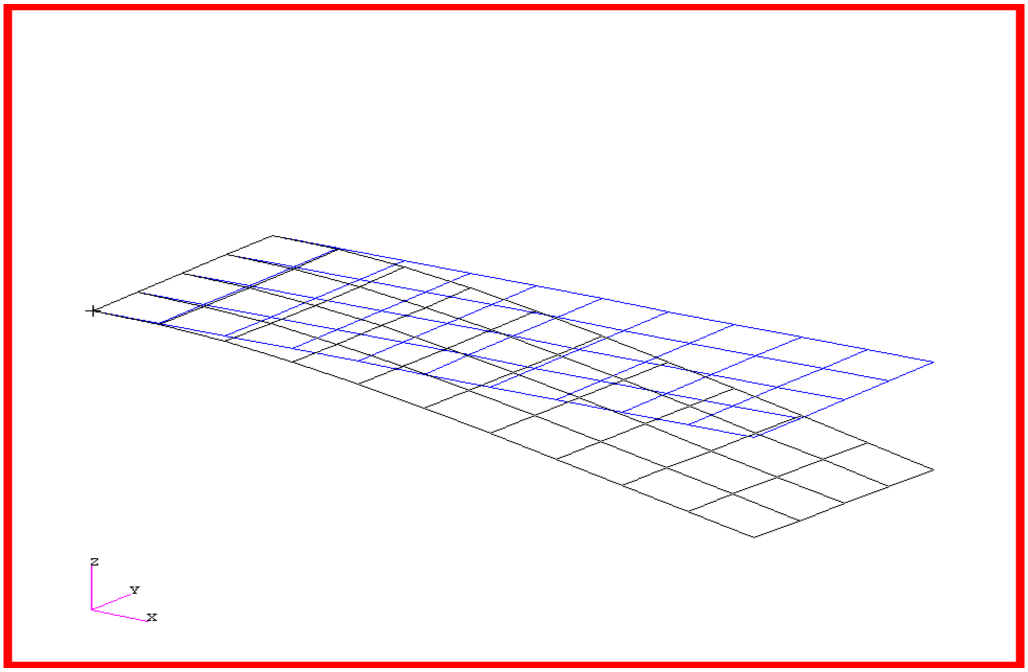
Figure 10.5


ACCEL PSD AT TIP CENTER
FREQUENCY
RANDOM ANALYSIS - BASE EXCITATION
USING THE MODAL METHOD WITH LANCZOS


ACCEL PSD AT OPPOSITE CORNER
FREQUENCY
RANDOM ANALYSIS - BASE EXCITATION
USING THE MODAL METHOD WITH LANCZOS

WORKSHOP 11

Random Analysis



Objectives:

- Define a frequency-varying excitation.
- Produce a MSC.Nastran input file from a dynamic math model created in Workshop 1.
- Submit the file for analysis in MSC.Nastran.
- Compute nodal displacements for desired frequency domain.

Model Description:

Using the modal method, determine the displacement response spectrum of the tip center point due to the input spectrum of the pressure and point loads listed below. Solve using the complex matrix representation [Sab] for the cross spectrum.

Table 11.1

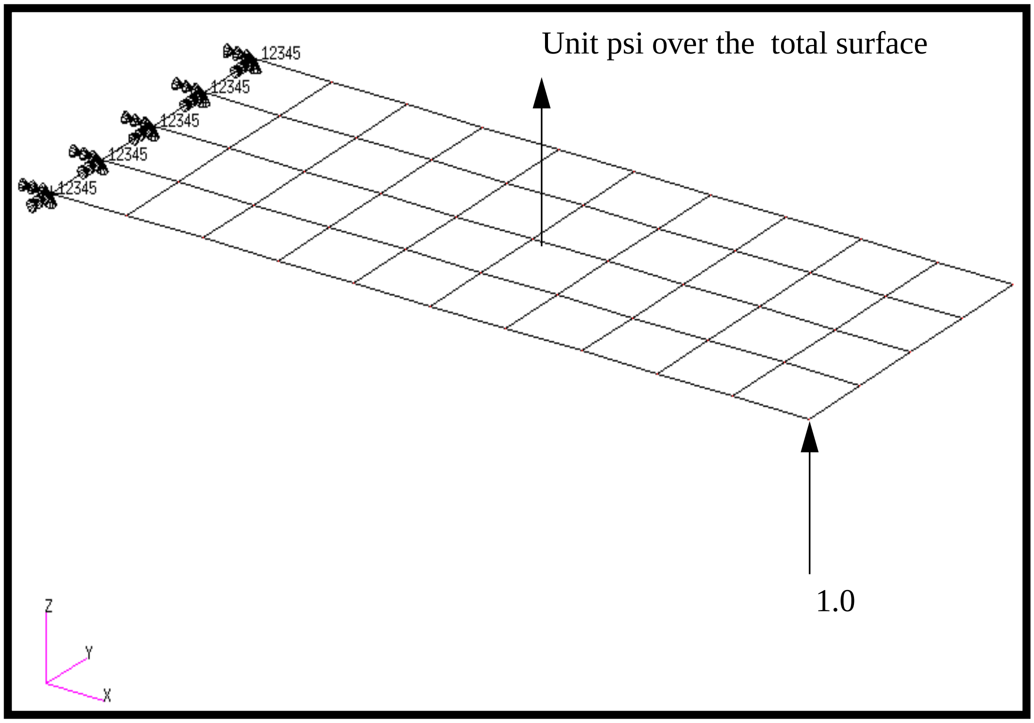
Autospectra of Pressure Load		**Autospectra of Corner Load**	
Frequency (Hz)	**psi^2/Hz**	**Frequency (Hz)**	**lb^2/Hz**
20	0.1	20	0.5
30	1	30	2.5
100	1	500	2.5
500	0.1	1000	0
1000	0.1		

Table 11.2

Cross-Spectrum of Pressure and Corner Loads Real/ Imaginary		
Frequency (Hz)	**Real Part**	**Imaginary Part**
20	-0.099619	0.007816
100	-0.498097	0.043579
500	0.070711	-0.070711
1000	0	0

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

Figure 11.1-Loads and Boundary Conditions



Suggested Exercise Steps:

- Reference a previously created dynamic math model, **plate.bdf**, by using the INCLUDE statement.
- Specify modal damping as a tabular function of natural frequency (TABDMP1).
- Define the frequency-varying pressure loading (PLOAD2, LSEQ and RLOAD2).
- Define the frequency-varying tip load (DAREA and RLOAD2).
- Define a set of frequencies to be used in the solution (FREQ1).
- Specify spectral density (RANDPS and TABRND1).
- Prepare the model for a random analysis (SOL 111).
- Request displacement response at loaded corner, tip center, and opposite corner.
- Generate an input file and submit it to the MSC.Nastran solver for random analysis.
- Review the results, specifically the nodal displacements.

```
ID SEMINAR,PROB11
```

```
CEND
```

BEGIN BULK

1	2	3	4	5	6	7	8	9	10

1	2	3	4	5	6	7	8	9	10

ENDDATA

Generating an input file for MSC.Nastran Users:

MSC.Nastran users can generate an input file using the data from pages 11-3 (general model description). The result should be similar to the output below.

1. MSC.Nastran input file: **prob11.dat**.

```
ID SEMINAR, PROB11
SOL 111
TIME 30
CEND
TITLE= FREQUENCY RESPONSE WITH PRESSURE AND POINT LOADS
SUBTITLE= USING THE MODAL METHOD WITH LANCZOS
ECHO= UNSORTED
SPC= 1
SET 111= 11, 33, 55
DISPLACEMENT(SORT2, PHASE)= 111
METHOD= 100
FREQUENCY= 100
SDAMPING= 100
RANDOM= 100
SUBCASE 1
LABEL= PRESSURE LOAD
DLOAD= 100
LOADSET= 100
SUBCASE 2
LABEL CORNER LOAD
DLOAD= 200
LOADSET= 100
$
OUTPUT (XYPLOT)
$
XTGRID= YES
YTGRID= YES
XBGRID= YES
YBGRID= YES
YTLOG= YES
YBLOG= NO
XTITLE= FREQUENCY (HZ)
YTTITLE= DISPLACEMENT RESPONSE AT LOADED CORNER, MAGNITUDE
YBTITLE= DISPLACEMENT RESPONSE AT LOADED CORNER, PHASE
XYPLOT DISP RESPONSE / 11 (T3RM, T3IP)
YTTITLE= DISPLACEMENT RESPONSE AT TIP CENTER, MAGNITUDE
```

```
YBTITLE= DISPLACEMENT RESPONSE AT TIP CENTER, PHASE
XYPLOT DISP RESPONSE / 33 (T3RM, T3IP)
YTTITLE= DISPLACEMENT RESPONSE AT OPPOSITE CORNER, MAGNITUDE
YBTITLE= DISPLACEMENT RESPONSE AT OPPOSITE CORNER, PHASE
XYPLOT DISP RESPONSE / 55 (T3RM, T3IP)
$
$ PLOT OUTPUT IS ONLY MEANS OF VIEWING PSD DATA
$
XGRID= YES
YGRID= YES
XLOG= YES
YLOG= YES
YTITLE= DISP P S D AT LOADED CORNER
XYPLOT DISP PSDF / 11(T3)
YTITLE= DISP P S D AT TIP CENTER
XYPLOT DISP PSDF / 33(T3)
YTITLE= DISP P S D AT OPPOSITE CORNER
XYPLOT DISP PSDF / 55(T3)
$
BEGIN BULK
$
PARAM,COUPMASS,1
$
PARAM,WTMASS,0.00259
$
$ MODEL DESCRIBED IN NORMAL MODES EXAMPLE
$
INCLUDE 'plate.bdf'
$
$ EIGENVALUE EXTRACTION PARAMETERS
$
EIGRL, 100, 10., 2000.
$
$ SPECIFY MODAL DAMPING
$
TABDMP1, 100, CRIT,
+, 0., .03, 10., .03, ENDT
$
$ FIRST LOADING
$
RLOAD2, 100, 300, , , 310
$
TABLED1, 310,
+, 10., 1., 1000., 1., ENDT
$
```

```
$ UNIT PRESSURE LOAD TO PLATE
$
LSEQ, 100, 300, 400
$
PLOAD2, 400, 1., 1, THRU, 40
$
$ SECOND LOADING
$
RLOAD2, 200, 600, , , 310
$
$ POINT LOAD AT TIP CENTER
$
DAREA, 600, 11, 3, 1.
$
$ SPECIFY FREQUENCY STEPS
$
FREQ1, 100, 20., 20., 49
$
$ SPECIFY SPECTRAL DENSITY
$
RANDPS, 100, 1, 1, 1., 0., 100
RANDPS, 100, 2, 2, 1., 0., 200
RANDPS, 100, 1, 2, 1., 0., 300
RANDPS, 100, 1, 2, 0., 1.0, 400
$
TABRND1, 100,
+, 20., 0.1, 30., 1., 100., 1., 500., .1,
+, 1000., .1, ENDT
$
TABRND1, 200,
+, 20., 0.5, 30., 2.5, 500., 2.5, 1000., 0.,
+, ENDT
$
TABRND1, 300,
+, 20., -.099619, 100., -.498097, 500., .070711, 1000., 0.,
+, ENDT
$
TABRND1, 400,
+, 20., .0078158, 100., .0435791, 500., -.70711, 1000., 0.,
+, ENDT
$
ENDDATA
```

Submitting the input file for analysis:

2. Submit the input file to MSC.Nastran for analysis.

   2a. To submit the MSC.Nastran **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran prob11 scr=yes**. Monitor the run using the UNIX **ps** command.

3. When the run is completed, use **plotps** utility to create a postscript file, **prob11.ps**, from the binary plot file **prob11.plt**. Compare the results with the plots below.

Figure 11.2

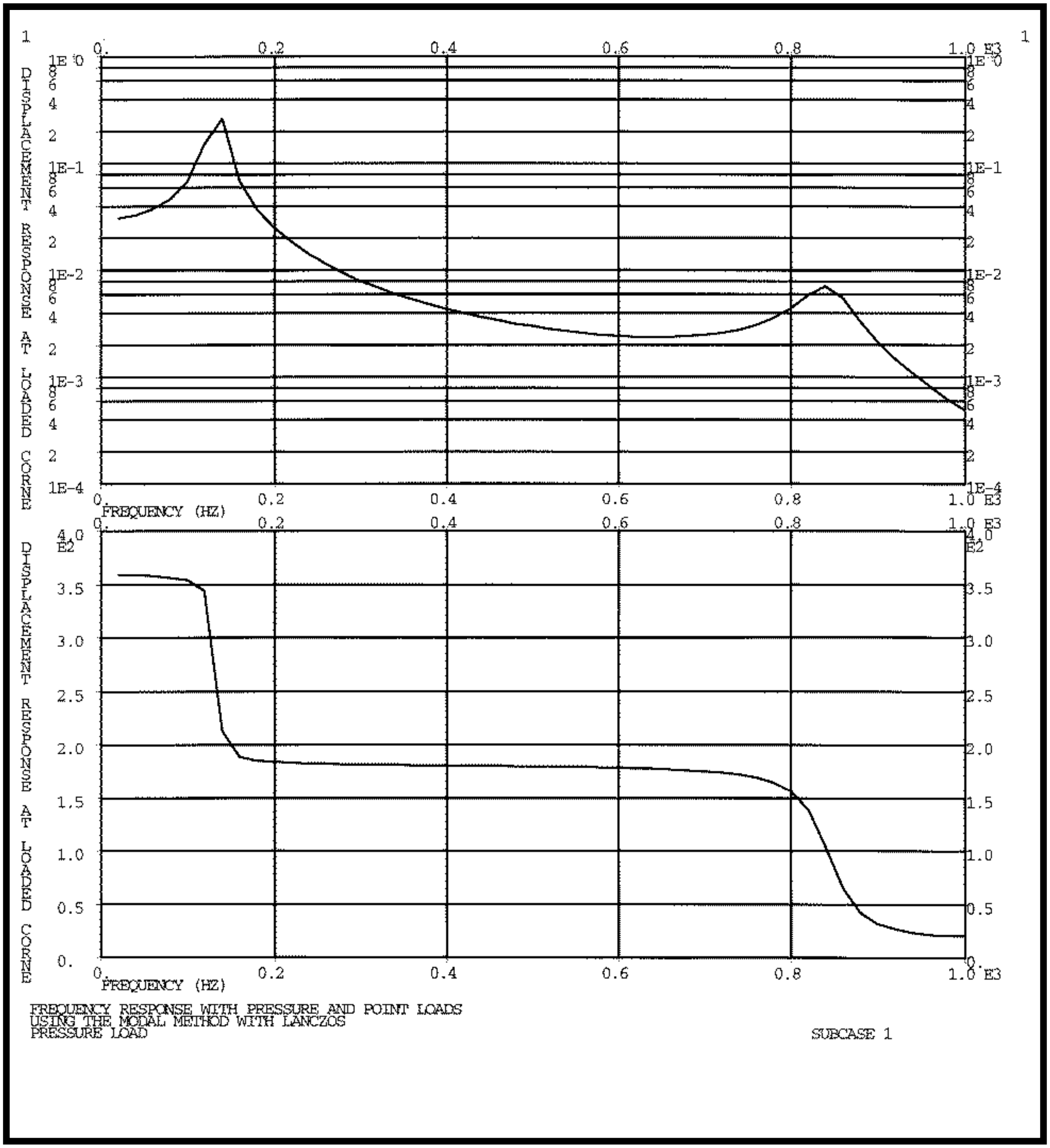

1
1
DISPLACEMENT RESPONSE AT LOADED CORNE
FREQUENCY (HZ)
0. 0.2 0.4 0.6 0.8 1.0 E3
1E 0
1E-1
1E-2
1E-3
1E-4
4.0 E2
3.5
3.0
2.5
2.0
1.5
1.0
0.5
0.
FREQUENCY RESPONSE WITH PRESSURE AND POINT LOADS
USING THE MODAL METHOD WITH LANCZOS
PRESSURE LOAD
SUBCASE 1

Figure 11.3

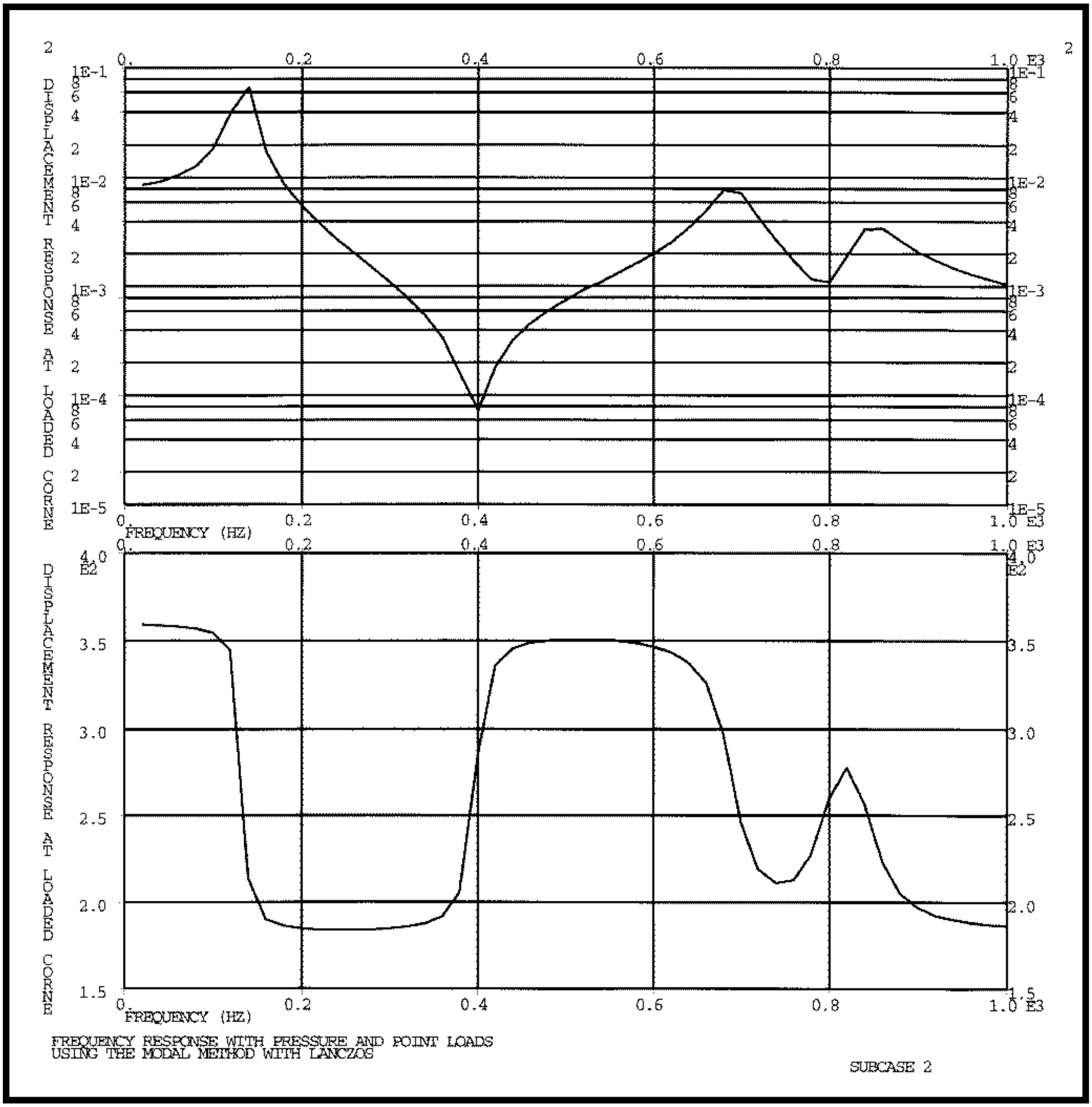

DISPLACEMENT RESPONSE AT LOADED CORNE
FREQUENCY (HZ)
FREQUENCY RESPONSE WITH PRESSURE AND POINT LOADS
USING THE MODAL METHOD WITH LANCZOS
SUBCASE 2

Figure 11.4

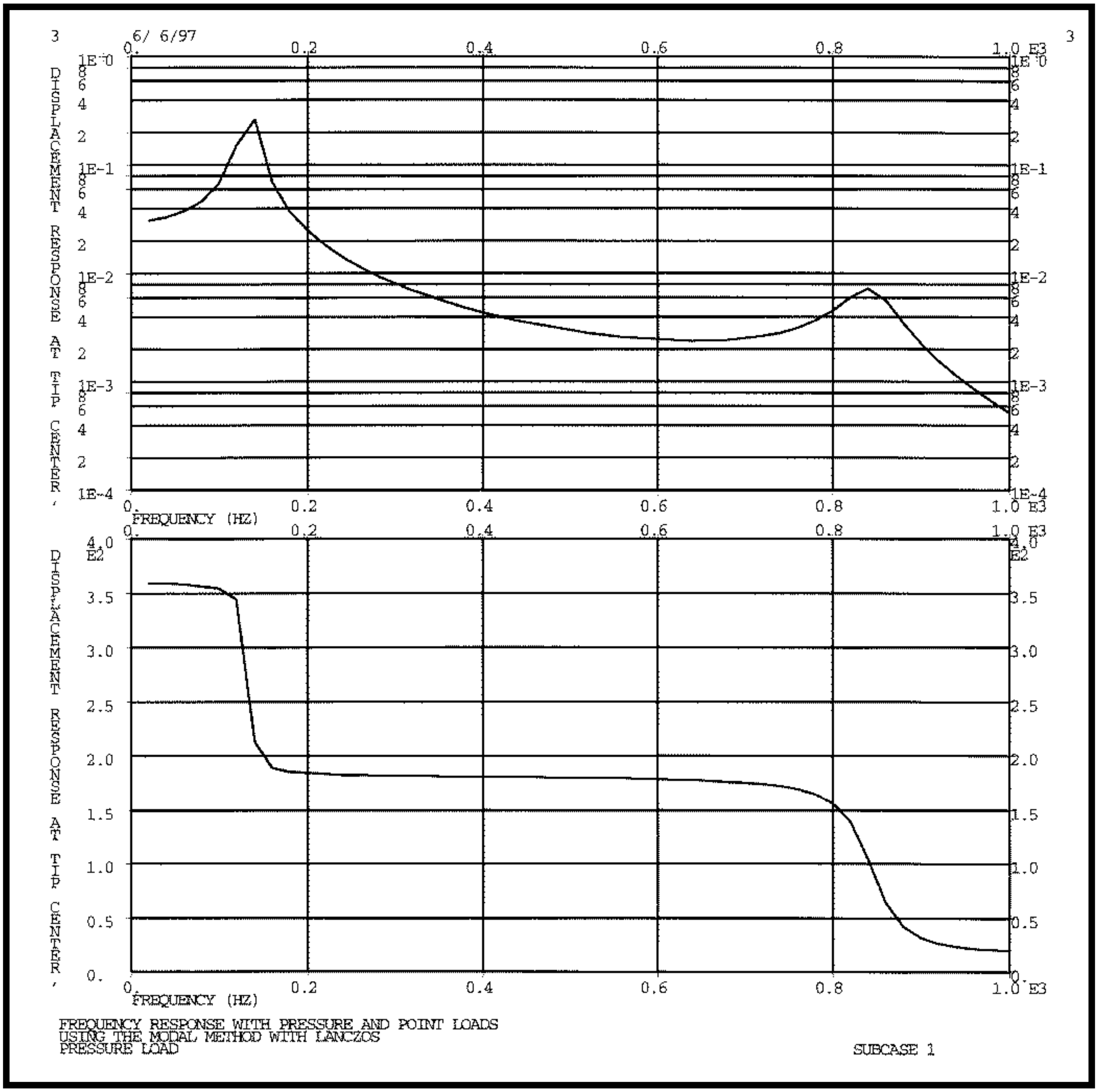

6/ 6/97
DISPLACEMENT RESPONSE AT TIP CENTER,
FREQUENCY (HZ)
FREQUENCY RESPONSE WITH PRESSURE AND POINT LOADS
USING THE MODAL METHOD WITH LANCZOS
PRESSURE LOAD
SUBCASE 1

Figure 11.5

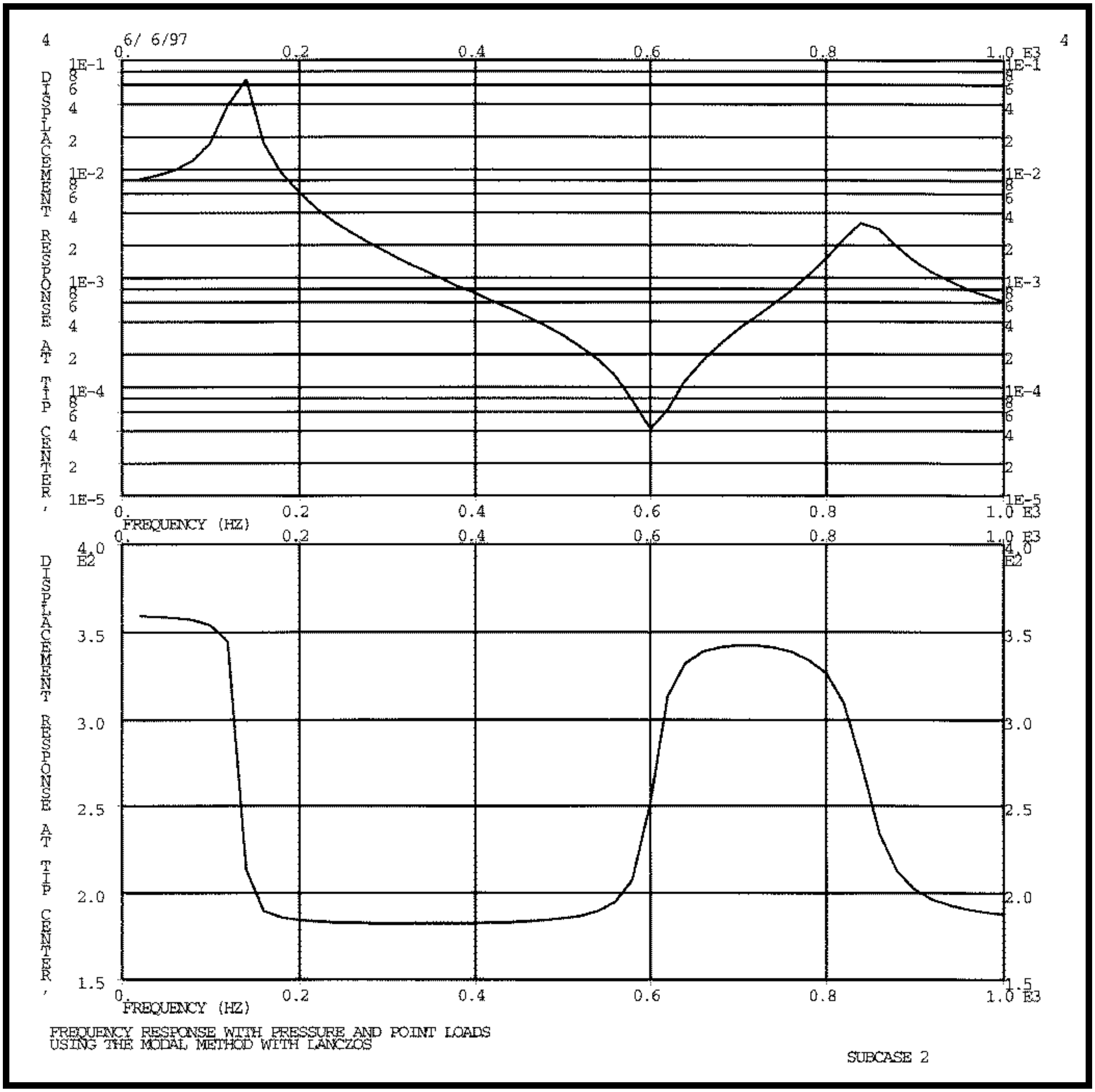

6/ 6/97
DISPLACEMENT RESPONSE AT TIP CENTER,
FREQUENCY (HZ)
FREQUENCY RESPONSE WITH PRESSURE AND POINT LOADS
USING THE MODAL METHOD WITH LANCZOS
SUBCASE 2

Figure 11.6

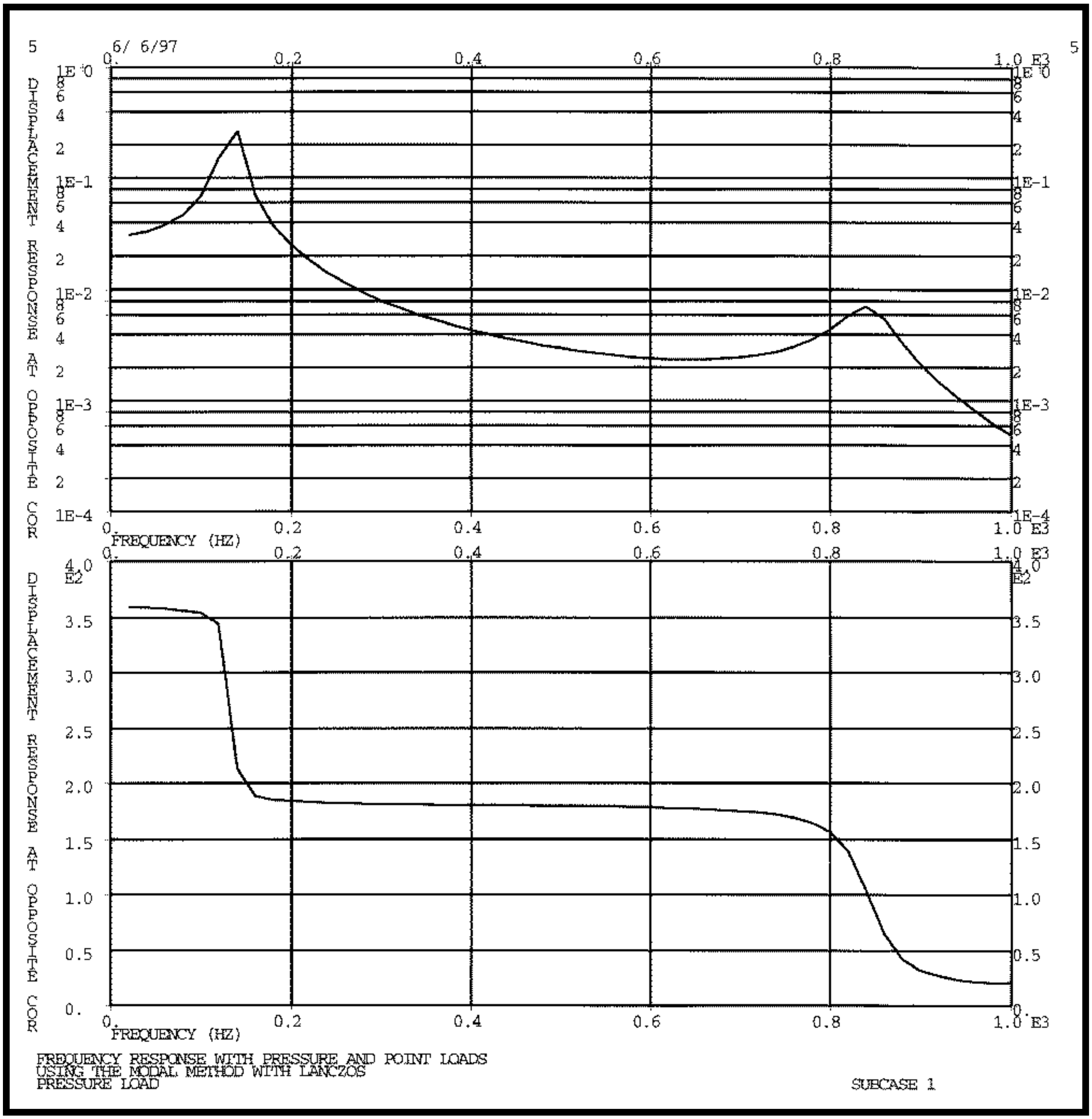

6/ 6/97
DISPLACEMENT RESPONSE AT OPPOSITE COR
FREQUENCY (HZ)
FREQUENCY RESPONSE WITH PRESSURE AND POINT LOADS
USING THE MODAL METHOD WITH LANCZOS
PRESSURE LOAD
SUBCASE 1

Figure 11.7

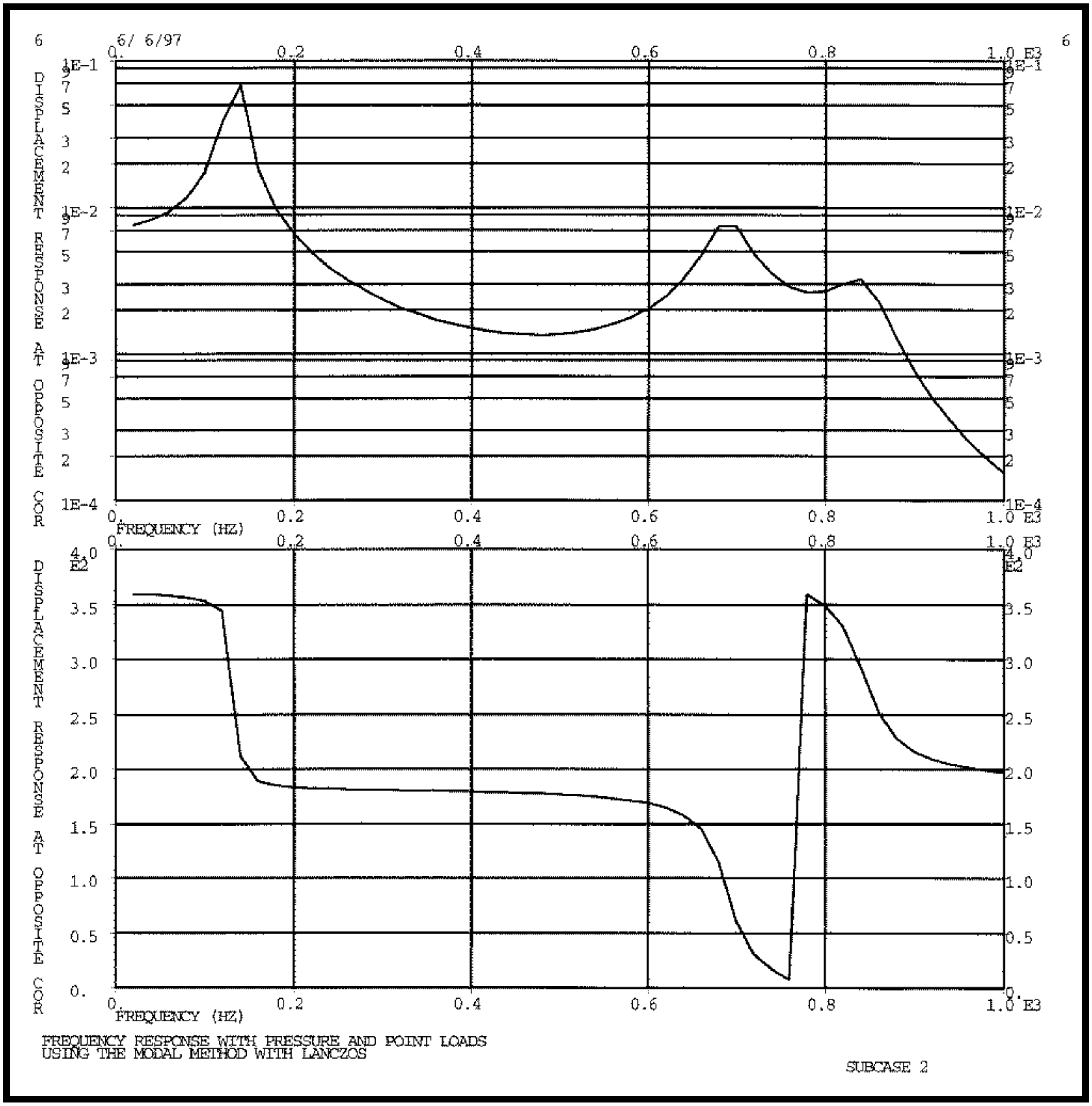

6/ 6/97
DISPLACEMENT RESPONSE AT OPPOSITE COR
FREQUENCY (HZ)
1.0 E3
DISPLACEMENT RESPONSE AT OPPOSITE COR
4.0 E2
FREQUENCY (HZ)
FREQUENCY RESPONSE WITH PRESSURE AND POINT LOADS
USING THE MODAL METHOD WITH LANCZOS
SUBCASE 2

Figure 11.8

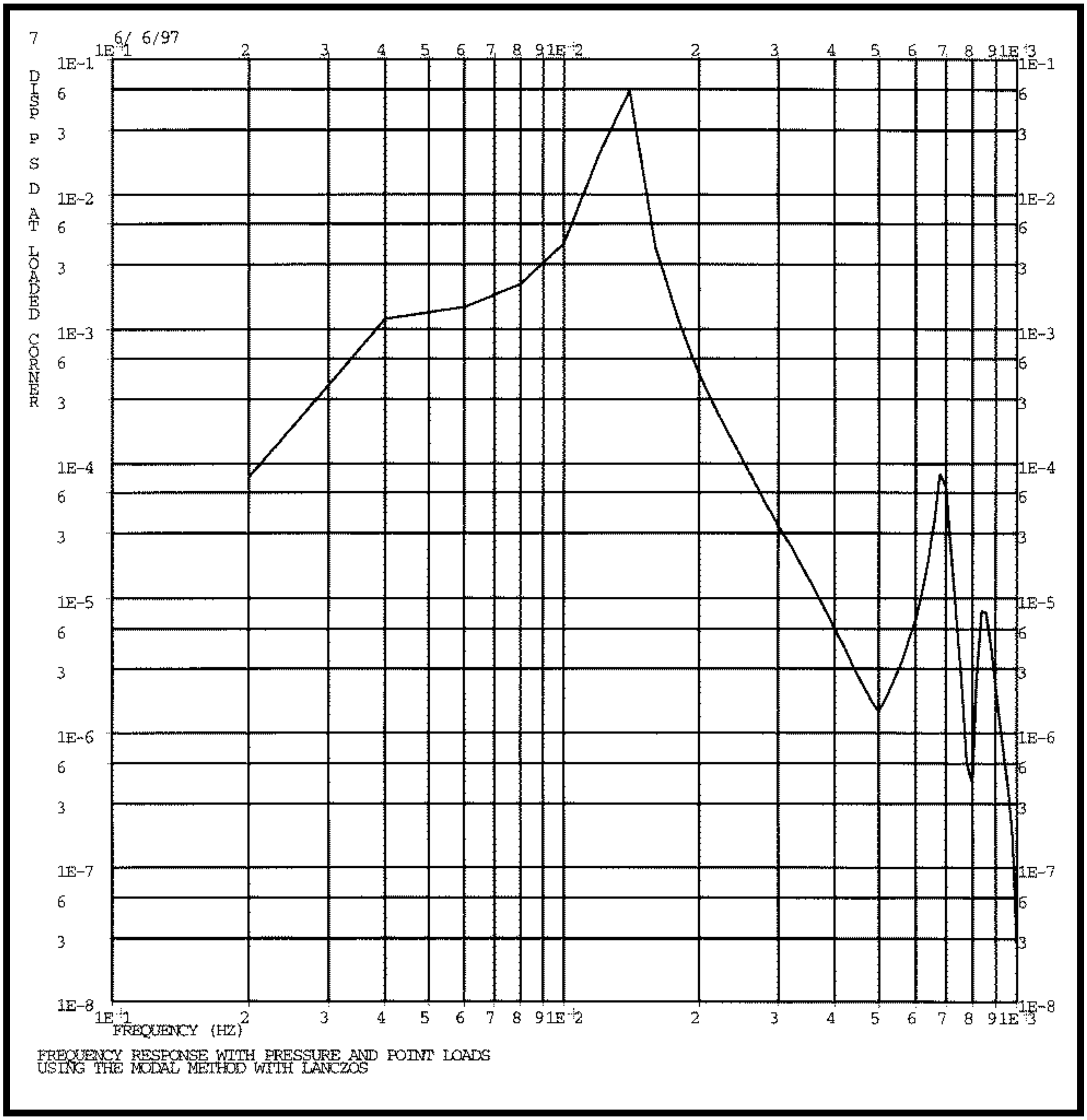

6/ 6/97
DISP P S D AT LOADED CORNER
FREQUENCY (HZ)
FREQUENCY RESPONSE WITH PRESSURE AND POINT LOADS
USING THE MODAL METHOD WITH LANCZOS

Figure 11.9

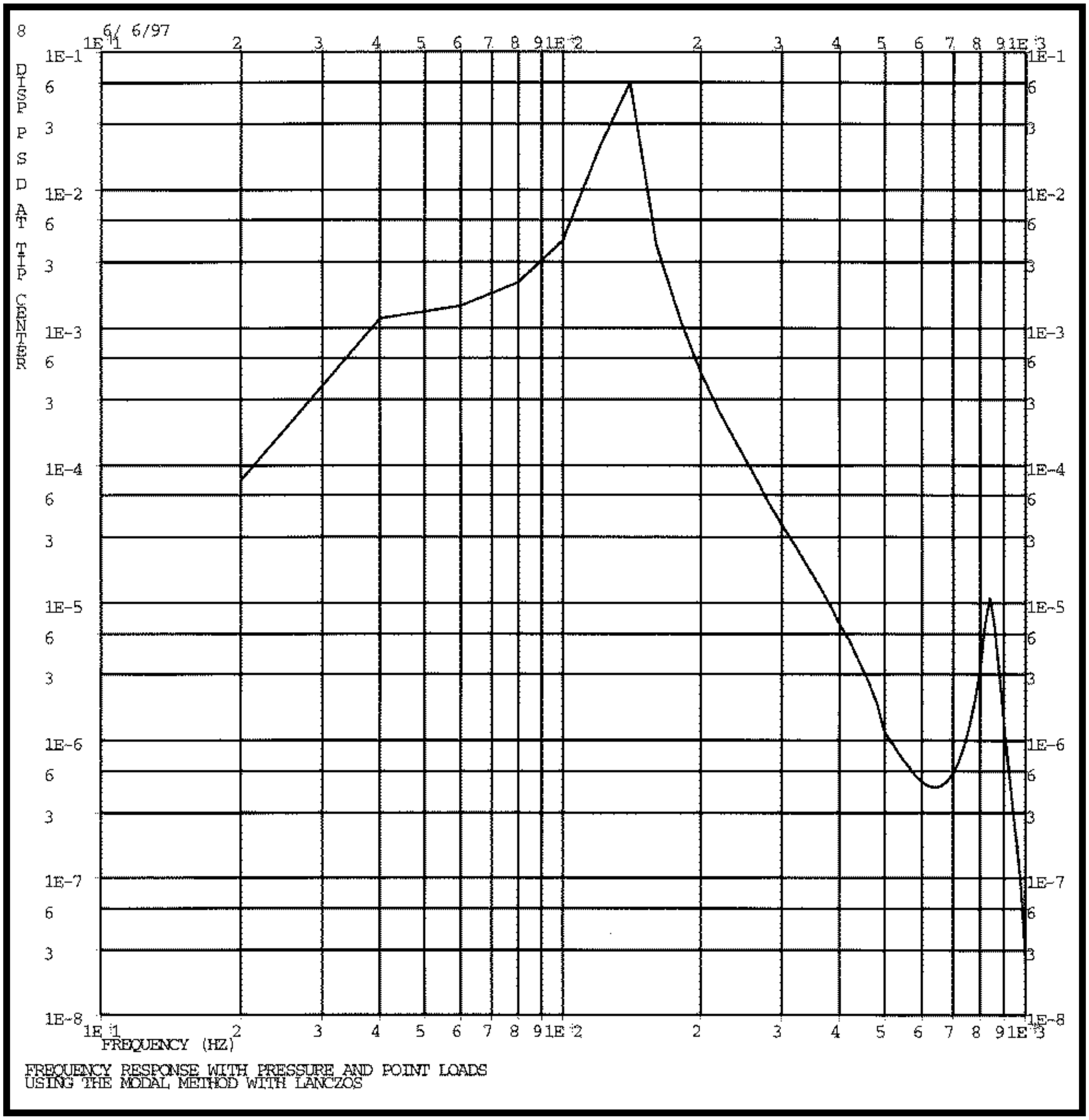
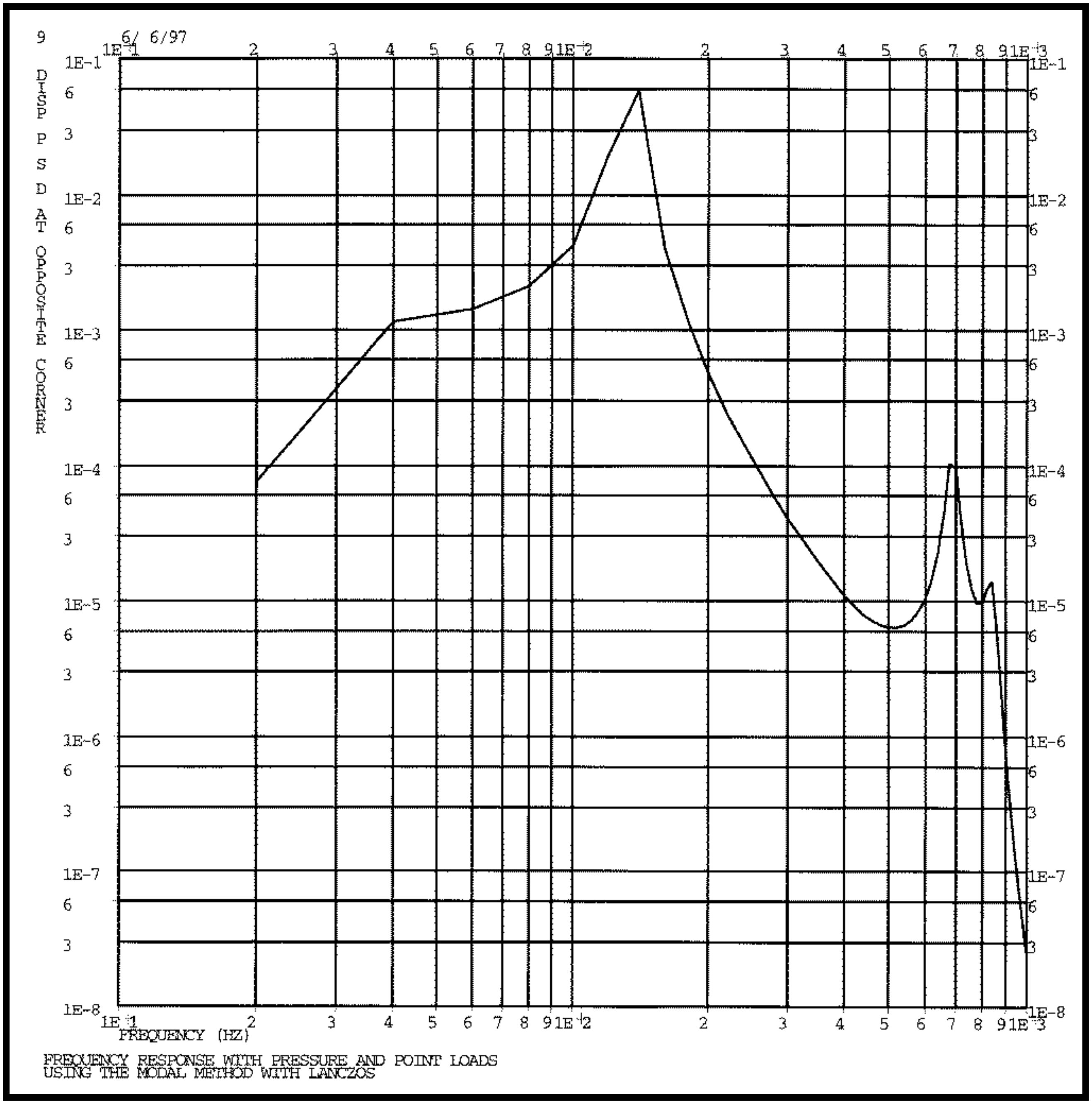
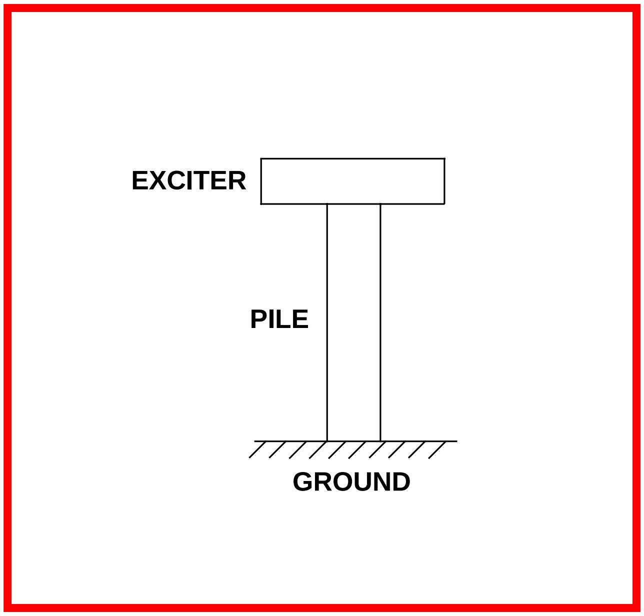

6/ 6/97
DISP PSD AT TIP CENTER
FREQUENCY (HZ)
FREQUENCY RESPONSE WITH PRESSURE AND POINT LOADS
USING THE MODAL METHOD WITH LANCZOS

Figure 11.10


6/ 6/97
DISP PSD AT OPPOSITE CORNER
FREQUENCY (HZ)
FREQUENCY RESPONSE WITH PRESSURE AND POINT LOADS
USING THE MODAL METHOD WITH LANCZOS

WORKSHOP 12

Complex Modes of a Pile Driver



Objectives

- Define complex eigenvalue extraction parameters.
- Submit the file for analysis in MSC.Nastran.
- Compute complex modes.

Model Description:

The model is idealized as shown below in Figure 12.1. (Note that both a spring element and a damper element will be created connected Grid 2 and Grid 3.)

Figure 12.1-Model Description

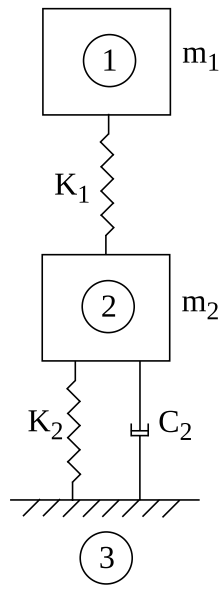


Table 12.1

m_1	**3.0 lb-sec^2/in**
m_2	**1.5 lb-sec^2/in**
K_1	**50,000 lb/in**
K_2	**12,500 lb/in**
C_2	**30 lb-sec/in**

Suggested Exercise Steps

- Generate an input file and submit it to the MSC.Nastran solver for complex eigenvalue analysis.
- Generate a finite element representation of the pile driver using GRID, CONM2, CELAS, and CVISC elements.
- Define material (MAT1), and element (PELAS) and (PVISC) properties.
- Apply x-direction boundary constraint (SPC1).
- Specify complex eigenvalue extraction parameters (CMETHOD) and (EIGC).
- Prepare the model for complex eigenvalue analysis (SOL107).
- Review the results, specifically the complex eigenvalues.

```
ID SEMINAR,PROB12
```

```
CEND
```

```
BEGIN BULK
```

1	2	3	4	5	6	7	8	9	10

1	2	3	4	5	6	7	8	9	10

ENDDATA

Exercise Procedure:

1. Users who are not utilizing MSC.Patran for generating an input file should go to Step 16, otherwise, proceed to step 2.

2. Create a new database and named **prob12.db**

    File/New Database

    New Database Name | **prob12**

    OK

    In the *New Model Preference* form set the following:

    Tolerance | ◆ **Default**

    Analysis code: | **MSC/NASTRAN**

    OK

3. Create the model by the edit method in **Finite Elements**.

    ◆ **Finite Elements**

    Action: | **Create**

    Object: | **Node**

    Method: | **Edit**

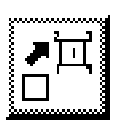
    ❑ **Associate with Geometry**

    Node Location List | **[0 0 0]**

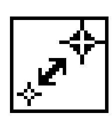
    Apply

    Turn on the label and increase the node size by using the Quick Pick buttons.

    Show Label

    Node Size

4. Similarly, create Nodes 2 and 3.

Node	**Location**
Node 2	[1 0 0]
Node 3	[2 0 0]

5. Create the Bar Element for Node 1 and Node 2.

◆ **Finite Element**

Action:	**Create**
Object:	**Element**
Method:	**Edit**
Shape	**Bar**
Node 1 =	**Node 1**
Node 2 =	**Node 2**

Apply

6. Similarly, create the 2nd bar element by:

Node 1 =	**Node 2**
Node 2 =	**Node 3**

Apply

7. Create the 2 mass elements at Node 1 and Node 2.

◆ **Finite Element**

Action:	**Create**
Object:	**Element**
Method:	**Edit**
Shape:	**Point**
Element ID List	**3**
Node 1 =	**Node 1**

Apply

Element ID List	**4**
Node 1 =	**Node 2**

Apply

8. Create the damper elements connecting Node 2 and Node 3.

◆ **Finite Element**

Action:	**Create**
Object:	**Element**
Method:	**Edit**
Shape	**Bar**
Node 1 =	**Node 2**
Node 2 =	**Node 3**

Apply

9. Create Element Properties, (spring constant).

◆ **Properties**

Action:	**Create**
Dimension:	**1D**
Type:	**Spring**
Property Set Name:	**spring1**

Input Properties ...

Spring Constant:	**50000**
DOF at Node 1:	**UX**
DOF at Node 2:	**UX**

OK

Application Region *(In the select menu, select the* ***Beam Element*** *filter.)*	**Element 1**

Beam Element

Add

Apply

10. Similarly, create the spring constant of 12,500 for the 2nd spring element.

Property Set Name:	**spring2**
Input Properties ...	
Spring Constant:	**12500**
DOF at Node 1:	**UX**
DOF at Node 2:	**UX**
OK	
Application Region	**Element 2**
Add	
Apply	

11. Create Element Properties, (damper coefficient), for the damper element:

◆ **Properties**

Action:	**Create**
Dimension:	**1D**
Type:	**Damper**
Property Set Name:	**damper**
Option(s)...	**Viscous**
Input Properties ...	
[Ext. Viscous Coeff.]	**30**
OK	
Application Region	**Element 5**

Add

Apply

12. Create the mass properties of the mass elements.

◆ **Properties**

Action:	**Create**
Dimension:	**0D**
Type:	**Mass**
Property Set Name:	**mass1**
Option(s):	**Lumped**
Input Properties ...	
Mass:	**3**
OK	
Application Region *(In the select menu, select the **Point Element** filter.)*	**Element 3**

△ **Point element**

Add

Apply

13. Similarly, create the mass property of the 2nd mass element:

◆ **Properties**

Action:	**Create**
Dimension:	**0D**
Type:	**Mass**
Property Set Name:	**mass2**
Option:	**Lumped**

Input Properties ...

Mass: **1.5**

OK

Application Region **Element 4**

Add

Apply

14. Create the constraint at the ground, Node 3.

◆ **Load/BCs**

Action: **Create**

Object: **Displacement**

Type: **Nodal**

New Set Name: **constraint**

Input Data...

Translations < T1 T2 T3 > **< 0, , >**

OK

Select Application Region...

◆ **FEM**

Select Nodes: **Node 3**

Add

OK

Apply

15. Create the analysis deck.

◆ **Analysis**

Action: **Analyze**

Object: **Entire Model**

Method: **Analysis Deck**

Job Name: **prob12**

Solution Type...	
Solution Type:	**◆ COMPLEX EIGENVALUES**
Solution Parameters ...	
Formulation	**Direct**
Complex Eigenvalue...	
Number of Desired Roots=	**4**
OK	
OK	
OK	
Apply	

Generating an input file for MSC.Nastran Users:

MSC.Nastran users can generate an input file using the data previously stated. The result should be similar to the output below.

16. MSC.Nastran input file: **prob12.dat**

```
ID SEMINAR, PROB12
SOL 107
TIME 5
CEND
TITLE= TWO-DOF MODEL (IMAC 8, PG 891)
SUBTITLE= COMPLEX MODES
DISPLACEMENT= ALL $ DEFAULT= REAL, IMAGINARY
SPC= 100
CMETHOD= 99
$
BEGIN BULK
$
$ COMPLEX EIGENVALUE EXTRACTION PARAMETERS
$
EIGC, 99, HESS, , , , , 4
$
$ DEFINE GRIDS, MASSES, AND STIFFNESSES
$  GRID 1 = EXCITER (X=2, MASS=3)  50K STIFFNESS BETWEEN GRIDS 1 AND 2
$  GRID 2 = PILE (X=1, MASS=3)   12.5K STIFFNESS BETWEEN GRIDS 2 AND 3
$  GRID 3 = BASE (X=0, FIX BASE)
$
GRID, 1, , 2., 0., 0.
GRID, 2, , 1., 0., 0.
GRID, 3, , 0., 0., 0.
GRDSET, , , , , , , 23456
CELAS2, 1, 50000., 1, 1, 2, 1
CELAS2, 2, 12500., 2, 1, 3, 1
CONM2, 201, 1, , 3.0
CONM2, 202, 2, , 1.5
SPC, 100, 3, 1
$
$ DEFINE DAMPER OF 30 BETWEEN GRIDS 2 AND 3
$
CVISC, 101, 1, 2, 3
PVISC, 1, 30.
$
ENDDATA
```

Submitting the input file for analysis:

17. Submit the input file to MSC.Nastran for analysis.

    17a. To submit the MSC.Patran **.bdf** file, find an available UNIX shell window. At the command prompt enter **nastran prob12.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

    17b. To submit the MSC.Nastran **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran prob12 scr=yes**. Monitor the run using the UNIX **ps** command.

18. When the run is completed, edit the **prob12.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing WARNING messages indicate modeling errors.

19. While still editing **prob12.f06**, search for the word:

    E I G E N V A L U E (spaces are necessary).

Comparison of Results

20. Compare the results obtained in the **.f06** file with the following results:

```
                                    C O M P L E X   E I G E N V A L U E   S U M M A R Y
             ROOT     EXTRACTION                  EIGENVALUE                        FREQUENCY              DAM
              NO.        ORDER             (REAL)           (IMAG)                  (CYCLES)                 C
FICIENT
                 1           2         -2.660969E+00    -4.983521E+01          7.931520E+00
1.067907E-01
                 2           3         -7.339031E+00    -2.360312E+02          3.756553E+01
6.218695E-02
                 3           1         -2.660969E+00     4.983521E+01          7.931520E+00
1.067907E-01
                 4           4         -7.339031E+00     2.360312E+02          3.756553E+01
6.218695E-02
                                                                                                  SUBCAS
      COMPLEX EIGENVALUE = -2.660969E+00, -4.983521E+01
                                   C O M P L E X   E I G E N V E C T O R   NO.         1
                                                       (REAL/IMAGINARY)

      POINT ID.   TYPE          T1             T2             T3             R1             R2
           1      G      1.000000E+00    .0             .0             .0             .0             .0
                          .0            .0             .0             .0             .0             .0
           2      G      8.514119E-01    .0             .0             .0             .0             .0
                         1.591320E-02    .0             .0             .0             .0             .0
           3      G       .0            .0             .0             .0             .0             .0
                          .0            .0             .0             .0             .0             .0
                                                                                                  SUBCA
     COMPLEX EIGENVALUE = -7.339031E+00, -2.360312E+02
                                  C O M P L E X   E I G E N V E C T O R   NO.         2
                                                      (REAL/IMAGINARY)

     POINT ID.   TYPE          T1             T2             T3             R1             R2
           1      G     -4.241094E-01    .0             .0             .0             .0             .0
                        -3.768431E-02    .0             .0             .0             .0             .0
           2      G      1.000000E+00    .0             .0             .0             .0             .0
                          .0            .0             .0             .0             .0             .0
           3      G       .0            .0             .0             .0             .0             .0
                          .0            .0             .0             .0             .0             .0
                                                                                                  SUBCA
     COMPLEX EIGENVALUE = -2.660969E+00,  4.983521E+01
                                  C O M P L E X   E I G E N V E C T O R   NO.         3
                                                      (REAL/IMAGINARY)

     POINT ID.   TYPE          T1             T2             T3             R1             R2
           1      G      1.000000E+00    .0             .0             .0             .0             .0
                          .0            .0             .0             .0             .0             .0
           2      G      8.514119E-01    .0             .0             .0             .0             .0
                        -1.591320E-02    .0             .0             .0             .0             .0
           3      G       .0            .0             .0             .0             .0             .0
                          .0            .0             .0             .0             .0             .0
                                                                                                  SUBCA
     COMPLEX EIGENVALUE = -7.339031E+00,  2.360312E+02
                                  C O M P L E X   E I G E N V E C T O R   NO.         4
                                                      (REAL/IMAGINARY)

     POINT ID.   TYPE          T1             T2             T3             R1             R2
           1      G     -4.241094E-01    .0             .0             .0             .0             .0
                         3.768431E-02    .0             .0             .0             .0             .0
           2      G      1.000000E+00    .0             .0             .0             .0             .0
                          .0            .0             .0             .0             .0             .0
           3      G       .0            .0             .0             .0             .0             .0
                          .0            .0             .0             .0             .0             .0
```

21. MSC.Nastran Users have finished this exercise. MSC.Patran Users should proceed to the next step.

22. Proceed with the Reverse Translation process, that is importing the **prob12.op2** results file into MSC.Patran. To do this, return to the ***Analysis*** form and proceed as follows.

◆ **Analysis**

Action:	**Read Output2**
Object:	**Result Entities**
Method:	**Translate**
Select Results File...	
Select Available Files	**prob12.op2**
OK	
Apply	

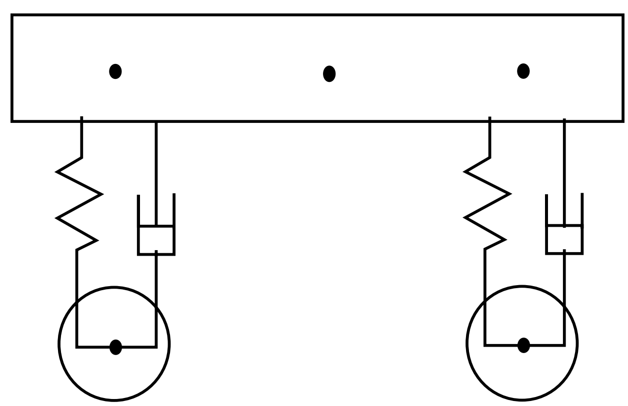
23. View the results.

◆ **Results**

Action:	**Create**
Object:	**Deformation**
Select Results Cases	**Default, Mode 1: freq. = 7.9315**
Select Deformation Result:	**Eigenvectors, Translational**
Plot Options	
Complex No as:	**Imaginary**
Apply	

WORKSHOP 13

Nolins in Linear Transient

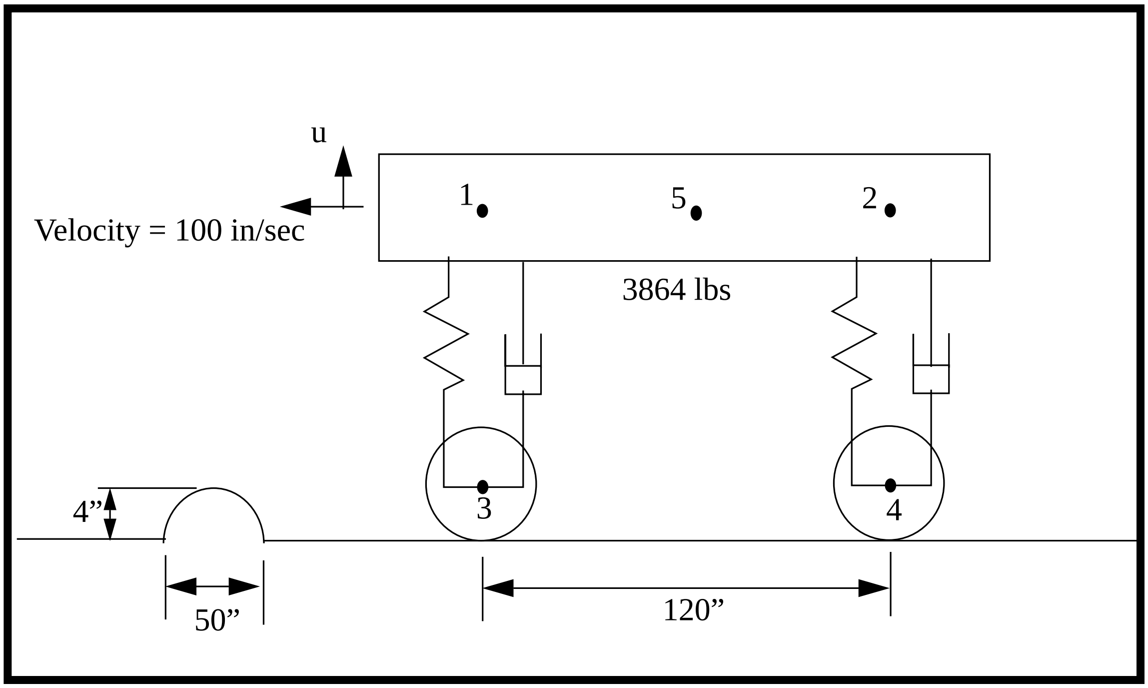


Objectives:

- Represent non-structural variables using non-structural DOFs.
- Define dynamic functions with transfer functions.
- Create a nonlinear transient force.
- Prepare a MSC.Nastran input file for a transient analysis.
- Visualize analysis results.

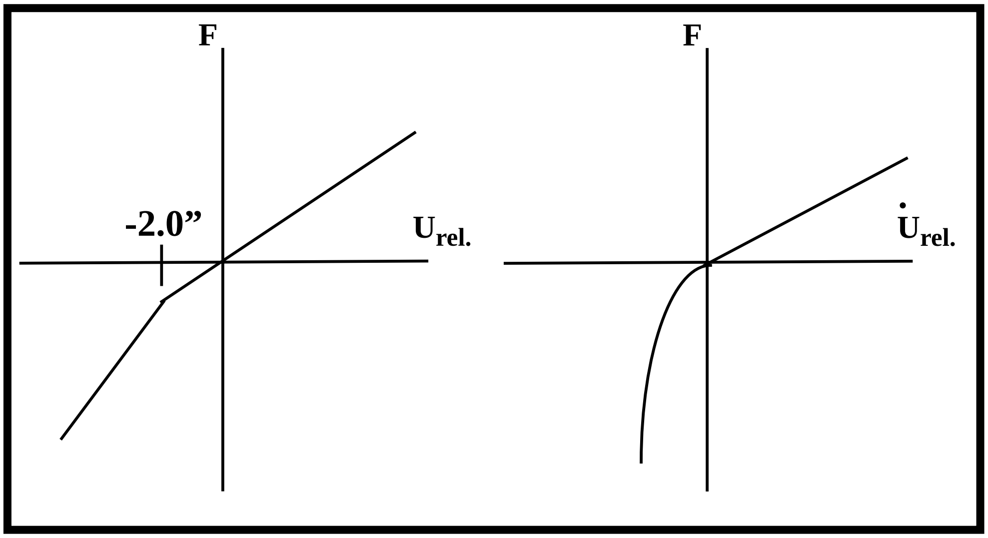
Model Description:

Figure 13.1-Car traveling over a speed bump



k :	$u \geq -2.0in$	**197.4 lb/in**
	$u < -2.0in$	**394.8 lb/in**
c :	$\dot{u} \geq 0$	**1.88 lb/(in/sec)**
	$\dot{u} \leq 0$	**1.88 lb/(in/sec) + 0.3 lb/(in/sec)^2**

Figure 13.2-Force vs Relative Velocity



Suggested Exercise Steps:

- Generate a finite element representation of the model using (GRID), (CBAR), and (CELAS2) elements.
- Define material (MAT1) and element (PBAR) properties.
- Constraints to eliminate rigid-body modes (SPC1).
- Define non-structural variables (CONM2).
- Specify scalar damper property and connection (CDAMP2).
- Define extra points (EPOINT).
- Define dynamic transfer functions (TF).
- Add nonlinear portion of the spring (NOLIN1).
- Add nonlinear portion of the damper (NOLIN4).
- Define the time-varying load (DAREA & TLOAD2).
- Define time delay (DELAY).
- Specify integration time step.
- Prepare the model for a direct transient analysis (SOL109).
- Request response in terms of nodal displacement and nonlinear load output.
- Generate an input file and submit it to the MSC.Nastran solver for direct transient analysis.
- Review the results, specially the xy plot of nodal displacements and nonlinear load.

ID SEMINAR,PROB13

CEND

BEGIN BULK

1	2	3	4	5	6	7	8	9	10

1	2	3	4	5	6	7	8	9	10

ENDDATA

Generating an input file for MSC.Nastran Users:

1. MSC.Nastran users can generate an input file using the data from pages 13-3 and 13-4 (general model description). The result should be similar to the output below (**prob13.dat**):

```
ASSIGN OUTPUT2 = 'prob13.op2', UNIT=12
ID NAS102, WORKSHOP13
SOL 109
TIME 100
CEND
TITLE= SIMPLE CAR MODEL WITH NOLINEAR
SUBTITLE= SPRINGS AND DAMPERS RUNNING OVER A BUMP
LABEL= SOL 109, CONSTANT DELTA TIME
SEALL= ALL
SPC= 100
TFL= 100
NONLINEAR = 100
DLOAD = 100
TSTEP = 100
DISPLACEMENT(PLOT)= ALL
NLLOAD(PLOT)= ALL
$
OUTPUT(XYPLOT)
CSCALE=1.3
XAXIS= YES
YAXIS= YES
XGRID LINES= YES
YGRID LINES= YES
XTITLE= TIME (SEC)
YTITLE= VERTICAL DISPLACEMENT OF POINT 1
XYPLOT DISP/1(T2)
YTITLE= VERTICAL DISPLACEMENT OF POINT 2
XYPLOT DISP/2(T2)
YTITLE= VERTICAL DISPLACEMENT OF POINT 3
XYPLOT DISP/3(T2)
YTITLE= VERTICAL DISPLACEMENT OF POINT 4
XYPLOT DISP/4(T2)
YTITLE= VERTICAL DISPLACEMENT OF POINT 5
XYPLOT DISP/5(T2)
YTITLE= NONLINEAR FORCES AT POINT 1
XYPLOT NONLINEAR/1(T2)
```

```
YTITLE= NONLINEAR FORCES AT POINT 2
XYPLOT NONLINEAR/2(T2)
$
BEGIN BULK
PARAM,POST,-1
PARAM,PATVER,3.0
$
$ CARRIAGE POINTS
$
GRID, 1, , 0., 0., 0.
GRID, 2, , 120., 0., 0.
GRID, 5, , 60., 0., 0.
$
$ WHEEL POINTS
$
GRID, 3, , 0., -10., 0.
GRID, 4, , 120., -10., 0.
$
$ CAR CARRIAGE
$
CBAR, 5, 11, 1, 5, 0., 1., 0.
CBAR, 6, 11, 5, 2, 0., 1., 0.
PBAR, 11, 12, 10., 10., 10.
MAT1, 12, 3.0E+7, , .33
$
$ CONSTRAINTS TO ELIMINATE RIGID-BODY MODES
$
SPC1, 100, 1345, 1, 2, 5
SPC1, 100, 13456, 3, 4
$
$ SYSTEM WILL HAVE A NATURAL FREQUENCY OF 1 HZ
$ WITH CRITICAL DAMPING OF 1 PERCENT
$
CONM2, 10, 1, ,2.5
CONM2, 15, 2, ,2.5
CONM2, 20, 5, ,5.
$
CELAS2, 30, 197.4, 1, 2, 3, 2
CELAS2, 40, 197.4, 2, 2, 4, 2
$
CDAMP2, 50, 1.88, 1, 2, 3, 2
CDAMP2, 60, 1.88, 2, 2, 4, 2
$
$ DEFINE EXTRA POINTS TO HOLD DIFFERENCES
$ BETWEEN WHEELS AND CARRIAGE
```

```
$
EPOINT, 101, 102
$
$ USE TRANSFER FUNCTIONS TO TRACK DIFFERENCES
$ 101= V1 - V3
$ 102= V2 - V4
$
TF, 100, 101, 0, 1., 0., 0.,
, 1, 2, -1., 0., 0.,
, 3, 2, 1., 0., 0.
$
TF, 100, 102, 0, 1., 0., 0.,
, 2, 2, -1., 0., 0.,
, 4, 2, 1., 0., 0.
$
$ ADD NONLINEAR PORTION OF SPRINGS
$
NOLIN1, 100, 1, 2, 197.4, 101, 0, 111
NOLIN1, 100, 2, 2, 197.4, 102, 0, 111
TABLED2, 111, -2.0,
, -1., 1., 0., 0., 1., 0.,ENDT
$
$ ADD NONLINEAR PORTION OF DAMPERS
$
NOLIN4, 100, 1, 2, -0.3, 101, 10, 2.
NOLIN4, 100, 2, 2, -0.3, 102, 10, 2.
$
$ USE LAGRANGE MULTIPLIERS TO IMPOSE WHEEL DISPLACEMENT
$ 103= V3
$ 104= V4
$
EPOINT, 103, 104
$
TF, 100, 103, 0, 0., 0., 0.,
, 3, 2, 1., 0., 0.
TF, 100, 3, 2, 0., 0., 0.,
, 103, 0, 1., 0., 0.
$
TF, 100, 104, 0, 0., 0., 0.,
, 4, 2, 1., 0., 0.
TF, 100, 4, 2, 0., 0., 0.,
, 104, 0, 1., 0., 0.
$
```

```
$ MOVE WHEELS OVER BUMP
$
TLOAD2, 100, 222, 333, 0, 0., 0.5, 1., -90.
DAREA, 222, 103, 0, 4.
DAREA, 222, 104, 0, 4.
DELAY, 333, 104, 0, 1.2
$
$ INTEGRATION INFORMATION
TSTEP, 100, 200, .05, 1
$
ENDDATA
```

Submitting the input file for analysis

2. Submit the input file to MSC.Nastran for analysis.

   To submit the MSC.Nastran **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran prob13 scr=yes**. Monitor the run using the UNIX **ps** command.

3. When the run is completed, use **plotps** utility to create a postscript file, **prob13.ps**, from the binary plot file **prob13.plt**. The nonlinear force and displacement plots are shown in figures 13.3 to 13.9.

4. When the run is completed, edit the **prob13.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing WARNING messages indicate modeling errors.

5. While still editing **prob13.f06**, search for the word:

   X Y - O U T P U T S U M M A R Y (spaces are necessary).

Comparison of Results

6. Compare the results obtained in the **.f06** file with thefollowing results:

X Y - O U T P U T   S U M M A R Y   (R E S P O N S E)

SUBCASE ID	CURVE TYPE	FRAME NO.	CURVE ID.	XMIN-FRAME/ ALL DATA	XMAX-FRAME/ ALL DATA	YMIN-FRAME/ ALL DATA	X FOR YMIN	YMAX-FRAME/ ALL DATA	X Y
1	NONLIN	1	1(4)	0.000000E+00	1.000002E+01	0.000000E+00	0.000000E+00	2.975151E+02	6.00000
				0.000000E+00	1.000002E+01	0.000000E+00	0.000000E+00	2.975151E+02	6.00000
1	NONLIN	2	2(4)	0.000000E+00	1.000002E+01	0.000000E+00	0.000000E+00	4.661461E+02	1.74999
				0.000000E+00	1.000002E+01	0.000000E+00	0.000000E+00	4.661461E+02	1.74999
1	DISP	3	1(4)	0.000000E+00	1.000002E+01	-2.836541E+00	2.199999E+00	5.942877E+00	4.50000
				0.000000E+00	1.000002E+01	-2.836541E+00	2.199999E+00	5.942877E+00	4.50000
1	DISP	4	2(4)	0.000000E+00	1.000002E+01	-2.671688E+00	1.999999E+00	7.464218E+00	1.60000
				0.000000E+00	1.000002E+01	-2.671688E+00	1.999999E+00	7.464218E+00	1.60000
1	DISP	5	3(4)	0.000000E+00	1.000002E+01	-4.440892E-16	5.000001E-01	4.000000E+00	2.50000
				0.000000E+00	1.000002E+01	-4.440892E-16	5.000001E-01	4.000000E+00	2.50000
1	DISP	6	4(4)	0.000000E+00	1.000002E+01	-1.332268E-15	1.749999E+00	4.000000E+00	1.45000
				0.000000E+00	1.000002E+01	-1.332268E-15	1.749999E+00	4.000000E+00	1.45000
1	DISP	7	5(4)	0.000000E+00	1.000002E+01	-2.150819E+00	2.149999E+00	3.963917E+00	1.60000
				0.000000E+00	1.000002E+01	-2.150819E+00	2.149999E+00	3.963917E+00	1.60000

7. **MSC.Nastran Users have finished this exercise. MSC.Patran Users should proceed to the next step.**

8. Proceed with the Reverse Translation process, that is importing the **prob13.op2** results file into a blank MSC.Patran database..

File/New

New database name:	**prob13**
◆ **Analysis**	
Action:	**Read Output2**
Object:	**Both**
Method:	**Translate**
Select Results File...	
Select Available Files	**prob13.op2**
OK	
Apply	

9. Plot the results in xy plots.

The first plot is to make the Displacement versus Frequency plot at Node 1.

◆ **Results**	
Action:	**Create**
Object:	**Graph**
Select Results Cases	**Springs and Dampers....**
Filter Method	**All**
Filter	
Apply	
Close	
Select y Result:	**Nonlinear Applied Load Translational**
Quanity:	**Magnitude**

x: **Global Variable**

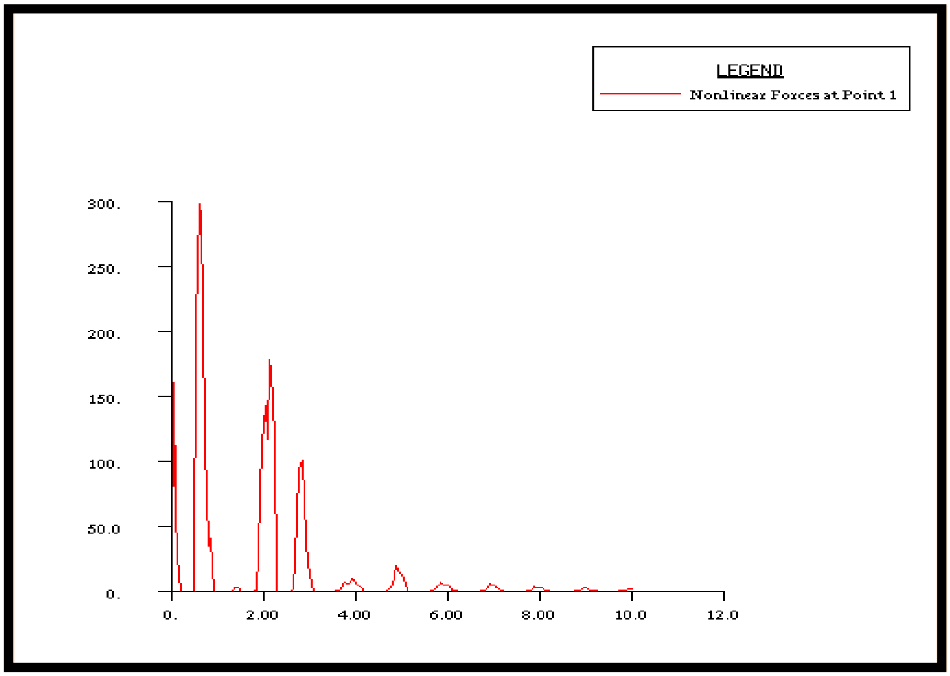
Variable: **Time**

Target Enities

Select Nodes: **Node 1**

Apply

Figure 13.3-Nonlinear Forces at Point 1

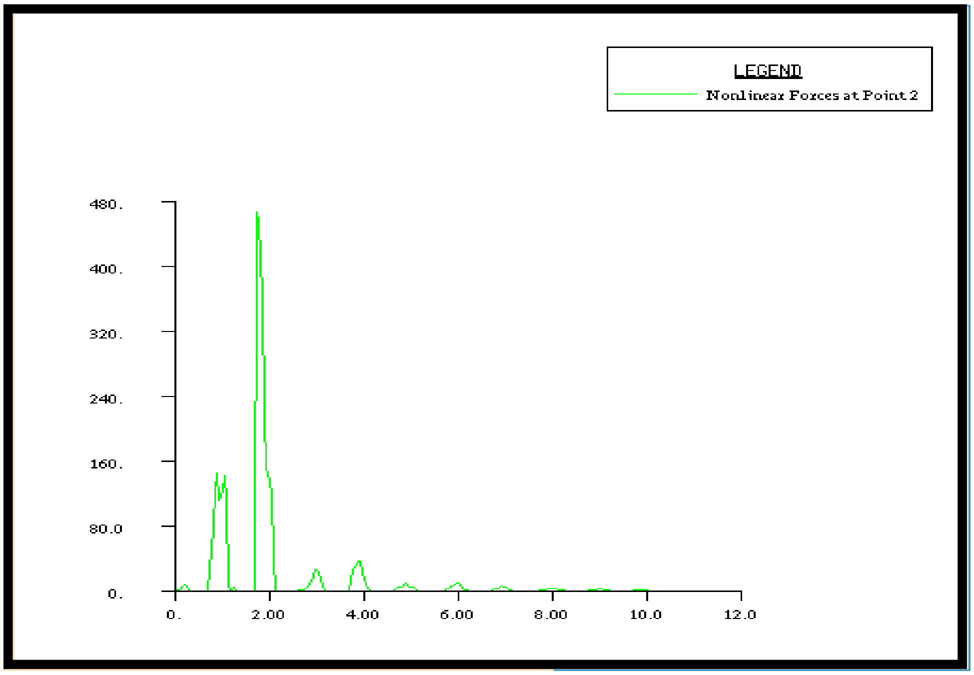


Target Enities

Select Nodes: **Node 2**

Apply

Figure 13.4-Nonlinear Forces at Point 2



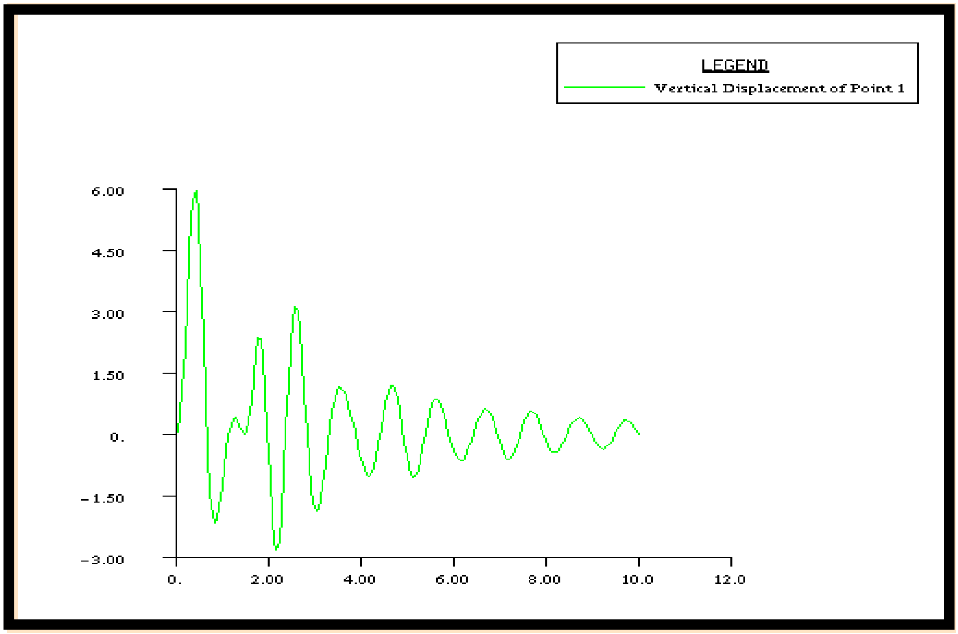
To get the displacement results***:***

Target Enities

Select Nodes:	**Node 1**
Select Results Cases	**Displacement, Translational**
Quanity:	**Y Component**

Apply

Figure 13.5-Vertical Displacement of Point 1

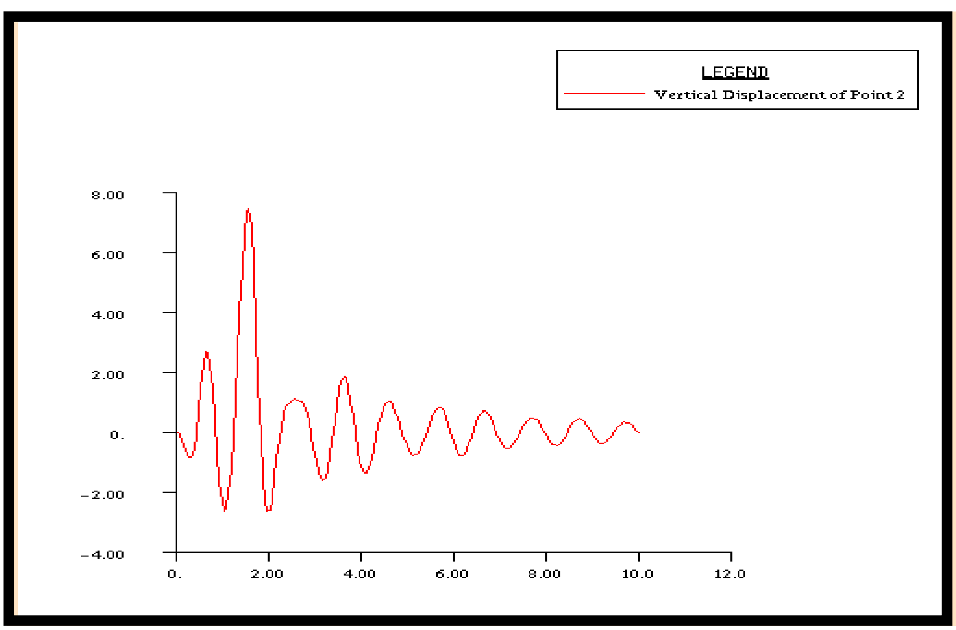


Target Enities

Select Nodes: **Node 2**

Apply

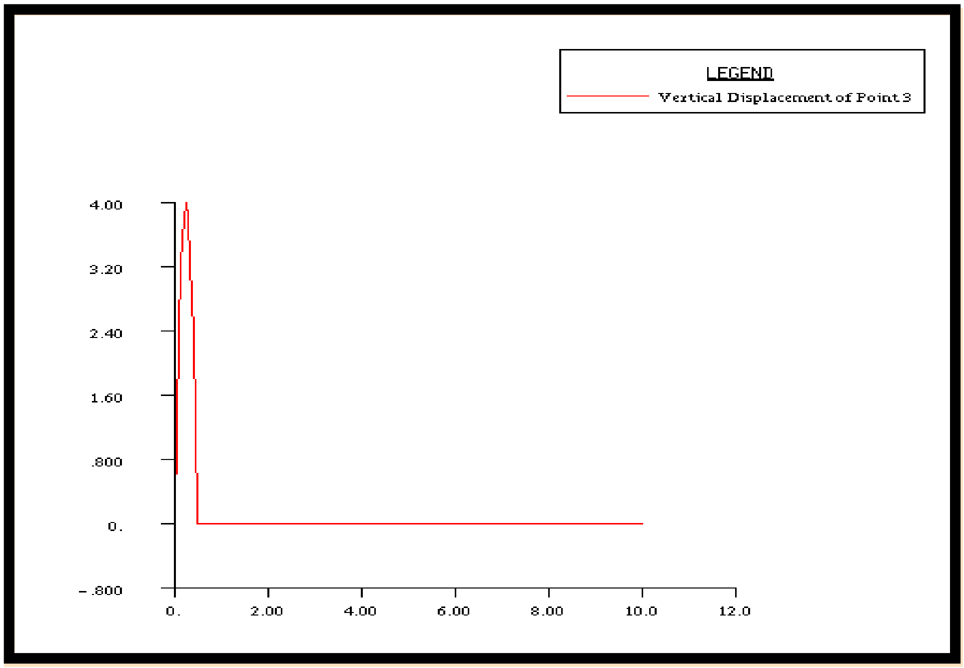
Figure 13.6-Vertical Displacement of Point 2



Select Nodes: **Node 3**

Apply

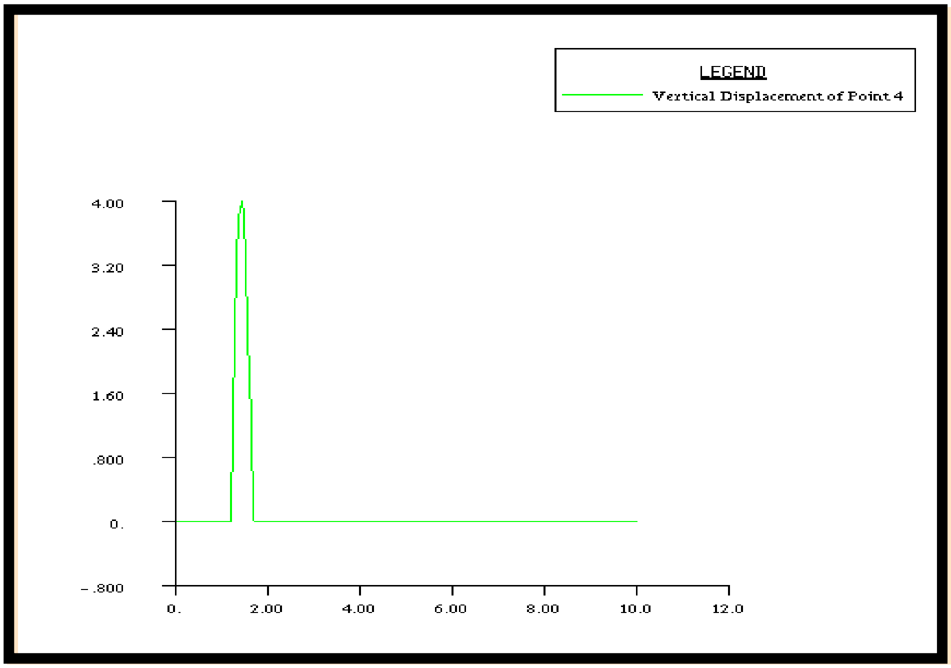
Figure 13.7-Vertical Displacement of Point 3



Select Nodes: **Node 4**

Apply

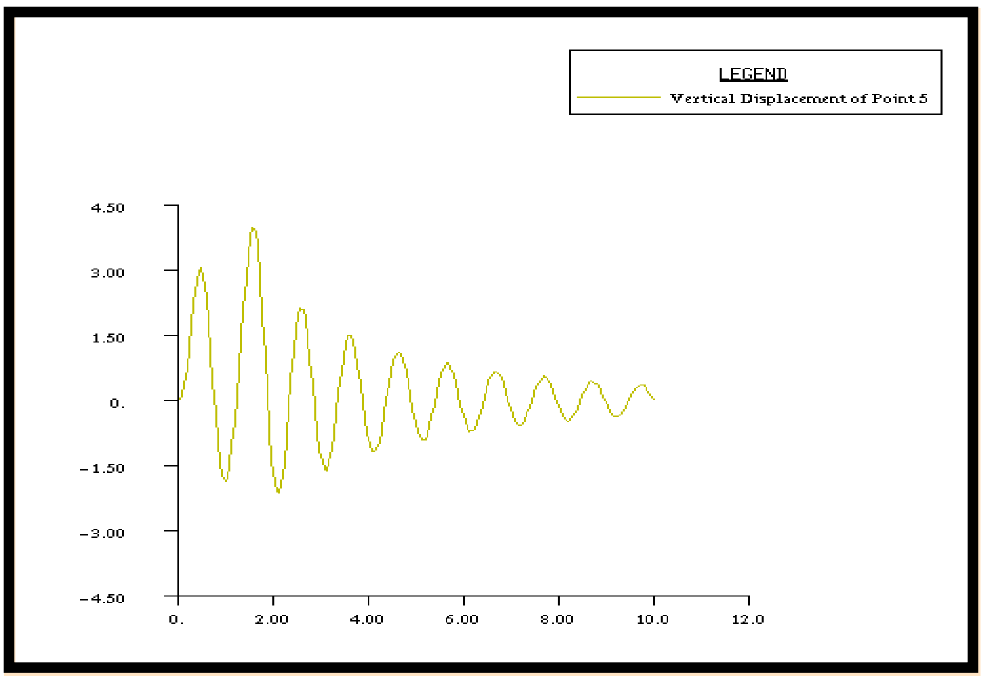
Figure 13.8-Vertical Displacement of Point 4



Select Nodes: **Node 5**

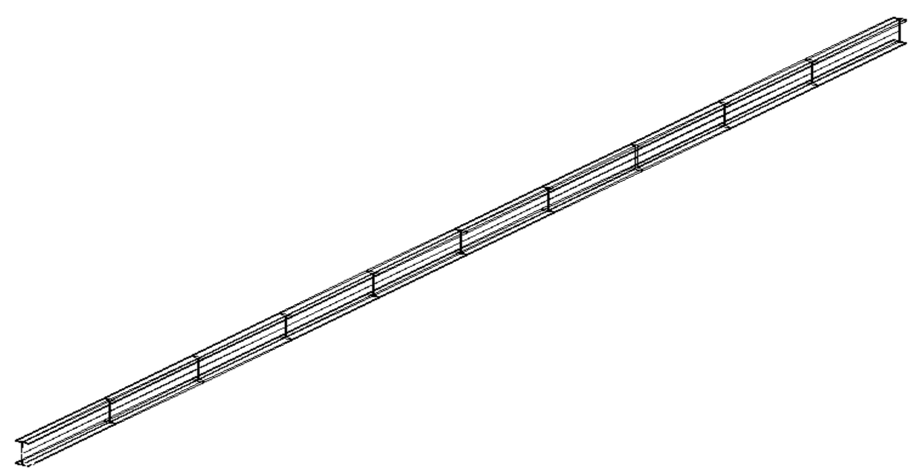
Apply

Figure 13.9-Vertical Displacement of Point 5



WORKSHOP 14a

Modal Analysis of a Beam



Objectives

- Perform normal modes analysis of a cantilever beam.
- Submit the file for analysis in MSC.Nastran.
- Find the first three natural frequencies and mode shapes of the beam.

Model Description:

The goal of this example is to find the first 3 modes of a beam pinned at both ends.

Figure 14a.1 below is a finite element representation of the beam. One end is contrained in all translation and the other is free to move in the X. Both ends are held in the X-rotation.

Figure 14a.1-Grid Coordinates and Element Connectivities

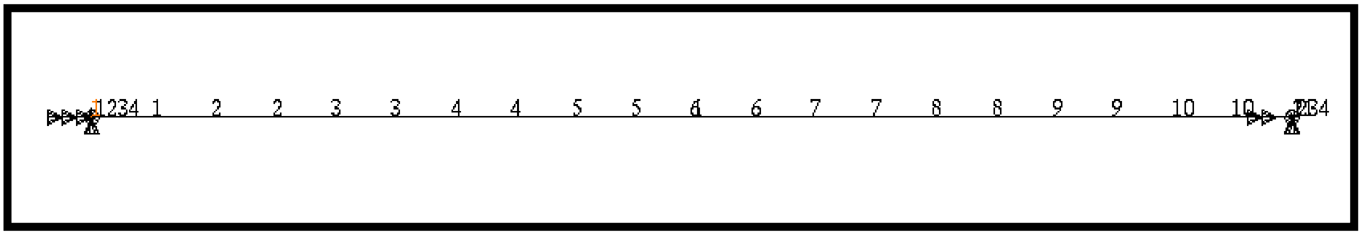


Figure 14a.2-Beam Cross Section

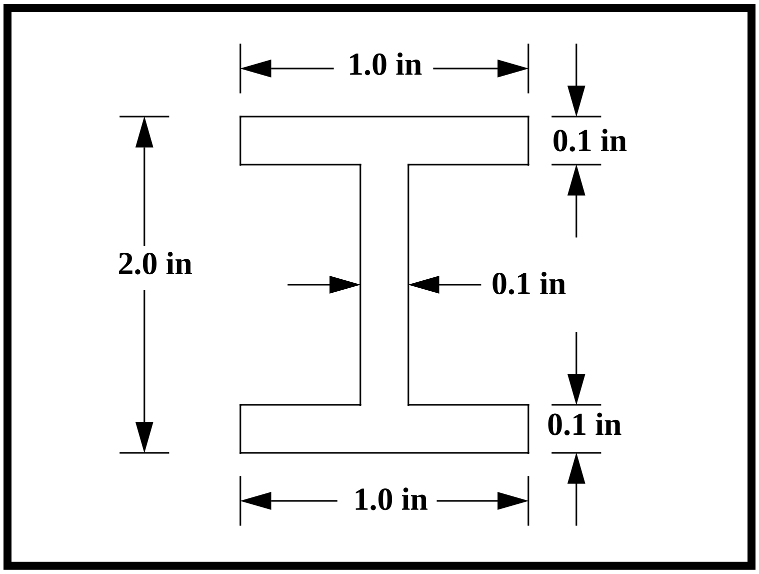


Table 14a.1

Length	**100 in**
Height	**2 in**
Width	**1 in**
Thickness	**0.100 in**
Area	**0.38 in^2**
I_1	**0.229 in^4**
I_2	**0.017 in^4**

Hand Calculations

$$f_n = \frac{K_n}{2\pi}\left[\frac{EIg}{Wl^4}\right]^{1/2}$$

$$f_n = K_n\left(\frac{1}{2\pi}\left[\frac{10\times10^6(0.229)(386.4)}{(0.38)(0.101)(100)^4}\right]^{1/2}\right)$$

$$f_n = K_n(2.417)$$

* I of the strong axis is used since translational Z DOF has been constrained by the permanent constraint.

From Theory

Mode	K_n	f_n
1	**9.87**	**23.85 Hz**
2	**39.5**	**95.46 Hz**
3	**88.8**	**214.59 Hz**

Suggested Exercise Steps

- Explicitly generate a finite element representation of the beam structure. (i.e., the grids (GRID) and element connectivities (CBAR) should be defined manually.)
- Define material (MAT1) and element (PBARL) properties.
- Apply the fixed boundary constraints (SPC1).
- Prepare the model for a normal modes analysis (SOL 103 and PARAMS).
  - PARAM, WTMASS, 0.00259
  - PARAM, COUPMASS, 1
  - EIGRL (To select Lanczos.)
- Generate an input file and submit it to the MSC.Nastran solver for normal modes analysis.
- Review the results, specifically the eigenvalues.

```
ID SEMINAR,PROB1
CEND
BEGIN BULK
```

1	2	3	4	5	6	7	8	9	10

1	2	3	4	5	6	7	8	9	10

ENDDATA

Exercise Procedure:

1. Users who are not utilizing MSC.Patran for generating an input file should go to Step 10, otherwise, proceed to step 2.

2. Create a new database named **prob14a.db**.

   File/New Database

New Database Name	**prob14a**

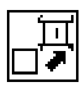
   OK

   In the *New Model Preference* form set the following:

Tolerance	◆ **Default**
Analysis Code:	**MSC/NASTRAN**

   OK

3. Activate the entity labels by selecting the Show Labels icon on the tool-bar.

    **Show Labels**

4. Create a curve.

   ◆ **Geometry**

Action:	**Create**
Object:	**Curve**
Method:	**XYZ**
Vector Coordinates List	**<100, 0, 0>**
Origin Coordinates List	**[0, 0, 0]**

   Apply

5. Create the finite element model and mesh the surface.

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Curve**
Global Edge Length	**10**
Curve List	**Curve 1**

Apply

6. Create nodal displacements.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Displacement**
Type:	**Nodal**
New Set Name	**disp1**

Input Data...

Translations <T1 T2 T3>	**<0 0 0>**
Rotations <R1 R2 R3>	**<0 , , >**

OK

Select Application Region...

■ **Geometry**

Select Geometry Entities	**Point 1**

Add

OK

Apply

New Set Name	**disp2**

Input Data...

Translations <T1 T2 T3>	**< , 0 0>**

Rotations <R1 R2 R3> | **<0 , , >**

OK

Select Application Region...

Select Geometry Entities | **Point 2**

Add

OK

Apply

New Set Name | **permanent_constraint**

Input Data...

Translations <T1 T2 T3> | **< , , 0>**

Rotations <R1 R2 R3> | **<0 , 0 , >**

OK

Select Application Region...

Select Geometry Entities | **Curve 1**

Add

OK

Apply

7. Create a set of material properties for the bar.

◆ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name	**alum**

Input Properties...

Elastic Modulus =	**10.0E6**
Poisson Ratio =	**.3**
Density =	**.101**

Apply

Cancel

8. Define the bar properties.

◆ **Properties**

Action:	**Create**
Dimension:	**1D**
Type:	**Beam**
Property Set Name	**bar**
Input Properties...	
Material Name (*Select from **Material Property Sets** box*)	**m:alum**
■ **Use Beam Section**	*<Click on Beam Library>*
New Section Name	**ibeam**
H	**2**
W1	**1**
W2	**1**
t	**0.1**
t1	**0.1**
t2	**0.1**
OK	
Bar Orientation	**Coord 0.2**
OK	
Select Members	**Curve 1**
Add	
Apply	

9. Now, you will generate the input file for analysis.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method	**Analysis Deck**
Job Name	**prob14a**
Solution Type...	
Solution Type:	◆ **NORMAL MODES**
Solution Parameters ...	
<deselect Automatic Constraints>	❐ **Automatic Constraints**
Mass Calculation:	**Coupled**
Data Deck Echo:	**None**
Wt. -Mass Conversion =	**.00259**
OK	
OK	
Subcase Create...	
Available Subcases	**Default**
Subcase Parameters...	
Number of Desired Roots =	**3**
OK	
Apply	
Cancel	*<to close form>*
Apply	

An MSC.Nastran input file called **prob14a.bdf** will be generated. The process of translating your model into an input file is called Forward Translation. The Forward Translation is complete when the Heartbeat turns green. MSC.Patran Users should proceed to step 11.

Generating an input file for MSC.Nastran Users:

MSC.Nastran users can generate an input file using the data from Table 14a.1. The result should be similar to the output below.

10. MSC.Nastran Input File: **prob14a.dat**

```
SOL 103
TIME 600
CEND
TITLE = Normal Modes Example
SUBCASE 1
   METHOD = 1
   SPC = 1
   VECTOR=ALL
BEGIN BULK
PARAM,WTMASS,.00259
PARAM,COUPMASS,1
EIGRL    1                       3       0
PBARL   1      1             I                                   +      A
+      A 2.      1.      1.      .1      .1      .1
CBAR     1       1       1       2       0.      1.      0.
CBAR     2       1       2       3       0.      1.      0.
CBAR     3       1       3       4       0.      1.      0.
CBAR     4       1       4       5       0.      1.      0.
CBAR     5       1       5       6       0.      1.      0.
CBAR     6       1       6       7       0.      1.      0.
CBAR     7       1       7       8       0.      1.      0.
CBAR     8       1       8       9       0.      1.      0.
CBAR     9       1       9       10      0.      1.      0.
CBAR     10      1       10      11      0.      1.      0.
MAT1     1       1.+7            .3      .101
GRID     1               0.      0.      0.              345
GRID     2               10.     0.      0.              345
GRID     3               20.     0.      0.              345
GRID     4               30.     0.      0.              345
GRID     5               39.9999 0.      0.              345
GRID     6               49.9999 0.      0.              345
GRID     7               60.     0.      0.              345
GRID     8               70.     0.      0.              345
GRID     9               80.     0.      0.              345
GRID     10              90.     0.      0.              345
GRID     11              100.    0.      0.              345
SPC1     1       1234    1
SPC1     1       234     11
ENDDATA
```

Submit the input file for analysis

11. Submit the input file to MSC.Nastran for analysis.

    11a. To submit the MSC.Patran **.bdf** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran prob14a.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

    11b. To submit the MSC.Nastran **.dat** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran prob14a scr=yes**. Monitor the run using the UNIX **ps** command.

12. When the run is completed, edit the **prob14a.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing WARNING messages indicate modeling errors.

13. While still editing **prob14a.f06**, search for the word:

E I G E N (spaces are necessary)

What are the first three modes?

1st = __________Hz

2nd = __________Hz

3rd = __________Hz

Comparison of Results

14. Compare the results obtained in the .f06 file with the following results:

R E A L   E I G E N V A L U E S

MODE NO.	EXTRACTION ORDER	EIGENVALUE	RADIANS	CYCLES	GENERALIZED MASS	GENERALIZED STIFFNESS
1	1	2.239398E+04	1.496462E+02	2.381693E+01	1.000000E+00	2.239398E+04
2	2	3.549898E+05	5.958102E+02	9.482614E+01	1.000000E+00	3.549898E+05
3	3	1.771818E+06	1.331096E+03	2.118506E+02	1.000000E+00	1.771818E+06

15. **MSC.Nastran Users have finished this exercise. MSC.Patran Users should proceed to the next step.**

16. Proceed with the Reverse Translation process, that is importing the **prob14a.op2** results file into MSC.Patran. To do this, return to the Analysis form and proceed as follows:

◆ **Analysis**

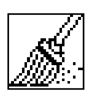
Action:	**Read Output2**
Object:	**Result Entities**
Method	**Translate**
Select Results File...	
Select Results File	**prob14a.op2**
OK	
Apply	

When the translation is complete bring up the ***Results*** form.

◆ **Results**

Action:	**Create**
Object:	**Deformation**
Select Results Cases	**Default, Mode 1:Freq=23.816**
Select Deformation Result	**Eigenvectors, Translational**
Apply	

To reset the graphics, click on this icon:



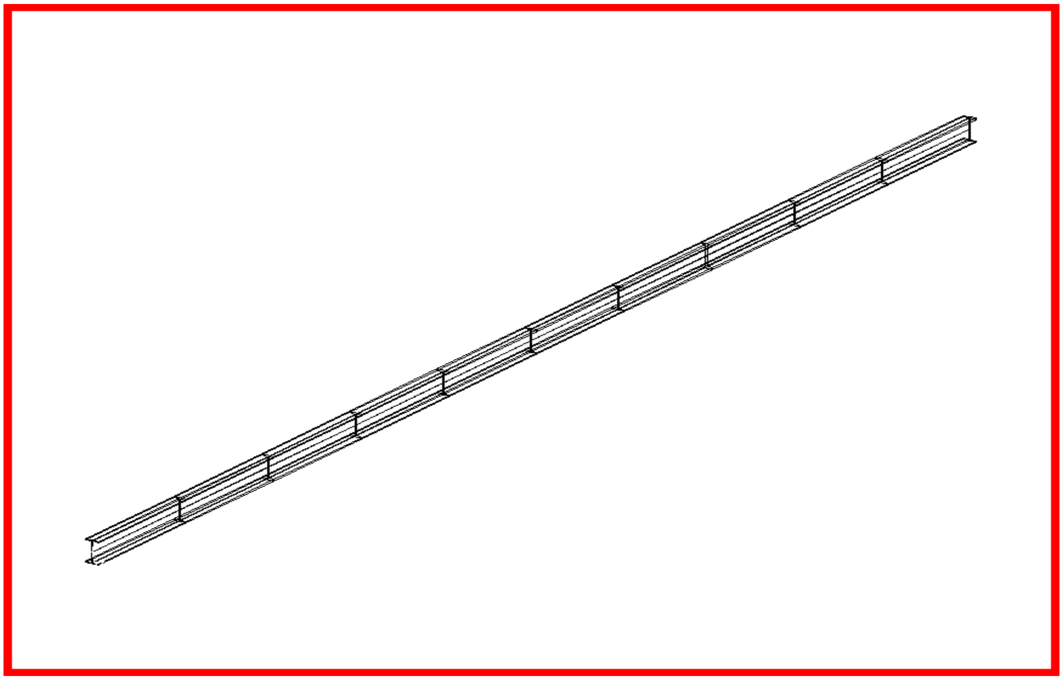
Reset Graphics

You can go back and select any *Results Case, Fringe Results or Deformation Results* you are interested in.

Quit MSC.Patran when you are finished with this exercise.

WORKSHOP 14b

Normal Modes with Differential Stiffness



Objectives

- Analyze a stiffened beam for normal modes.
- Produce an MSC/ NASTRAN input file that represent beam and load.
- Submit for analysis.
- Find normal modes (natural frequencies).

Model Description:

The goal of this example is to analyze a stiffened model. In this case, the beam from Problem 14a. with a 500 lb force applied.

Figure 14b.1 below is a finite element representation of the beam. This is no longer a simple normal modes analysis. Instead we will be using a nonlinear static solution (SOL 106) with (PARAM, NMLOOP and METHOD and EIGRL).

Below is a finite element representation of the beam. One end is pinned in 3 translations and one rotation. The other is pinned in 2 translations and one rotation with a 500 lb force applied.

Figure 14b.1-Grid Coordinates and Element Connectivities

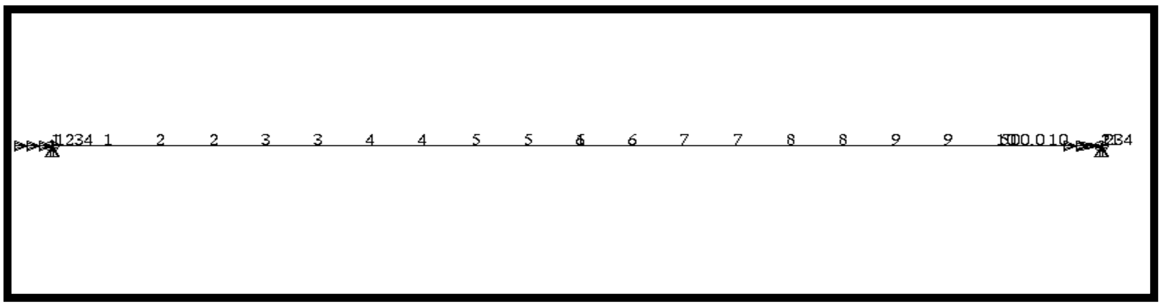


Figure 14b.2-Beam Cross Section

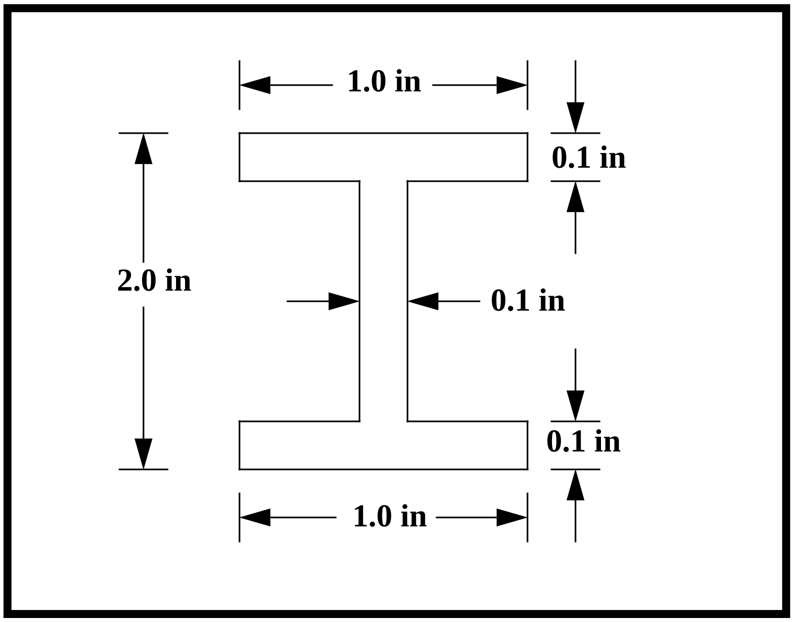


Table 14b.1

Length	**100 in**
Height	**2 in**
Width	**1 in**
Thickness	**0.100 in**
Area	**0.38 in^2**
I_1	**0.229 in^4**
I_2	**0.017 in^4**

Theoretical Solution

$$f_n = \frac{K_n}{2\pi}\left[\frac{EIg}{Wl^4}\left(1 + \frac{1}{Kr}\frac{Pl^2}{EI}\right)\right]^{1/2}$$

For Mode 1, Kr =9.87

$$f_n = \frac{9.87}{2\pi}\left[\frac{10\times10^6(0.229)(386.4)}{(0.38)(0.101)(100)^4} x\left(1 + \frac{1}{9.87}\frac{(500)(100)^2}{(10\times10^6)(0.229)}\right)\right]^{1/2}$$

$$f_n = 26.36Hz$$

For Static Load

$$\Delta = \frac{PL}{AE}$$

$$\Delta = \frac{500(100)}{0.38(10\times10^6)}$$

$$\Delta = 0.0132$$

Suggested Exercise Steps

- Open database created in Problem 1a in order to modify it, adding a load and reanalyze.
- Create 500 lb force applied at one end (FORCE).
- Make sure analysis is set to nonlinear static (SOL 106).
- Prepare nonlinear analysis to also analyze for normal mode (PARAM NMLOOP, EIGRL, LGDISP, NLPARM).
- Review the results, specifically the eigenvectors.

```
ID SEMINAR,PROB1
CEND
BEGIN BULK
```

1	2	3	4	5	6	7	8	9	10

1	2	3	4	5	6	7	8	9	10

ENDDATA

Exercise Procedure:

1. Users who are not utilizing MSC.Patran for generating an input file should go to Step 6, otherwise, proceed to step 2.

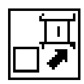
2. Open database created in Problem 14a named **prob14a.db**.

   File/Open Database

   Existing Database Name — **prob14a**

   OK

3. Activate the entity labels by selecting the Show Labels icon on the tool-bar.

    **Show Labels**

4. Create force.

   ◆ **Loads/BCs**

   Action: — **Create**

   Object: — **Force**

   Type: — **Nodal**

   New Set Name — **pull**

   Input Data...

   Force <F1 F2 F3> — **<500, , >**

   OK

   Select Application Region...

   Select Geometry Entities — **Point 2**

   Add

   OK

   Apply

5. Now, you will generate the input file for analysis.

   ◆ **Analysis**

   Action: — **Analyze**

Object:	**Entire Model**
Method	**Analysis Deck**
Job Name	**prob14b**
Solution Type...	
Solution Type:	**◆ NONLINEAR STATIC**
Solution Parameters ...	
<deselect Automatic Constraints>	**❒ Automatic Contraints**
Mass Calculation:	**Coupled**
Data Deck Echo:	**None**
Wt. -Mass Conversion =	**.00259**
OK	
OK	
Direct Text Input...	
◆ Case Control Section	**METHOD = 10**
◆ Bulk Data Section	**PARAM, NMLOOP, 5** **EIGRL, 10, , , 3**
OK	
Subcase Create...	
Available Subcases	**Default**
Subcase Parameters...	
Number of Load Increments =	**5**
OK	
Apply	
Cancel	
Apply	

An MSC.Nastran input file called **prob14b.bdf** will be generated. The process of translating your model into an input file is called Forward Translation. The Forward Translation is complete when the Heartbeat turns green. MSC.Patran Users should proceed to step 7.

Generating an input file for MSC.Nastran Users:

MSC.Nastran users can generate an input file using the data from Table 14b.1. The result should be similar to the output below.

6. MSC.Nastran Input File: **prob14b.dat**

```
SOL 106
TIME 600
CEND
$
TITLE = Normal Modes with Differential Stiffness
METHOD = 10
SUBCASE 1
   NLPARM = 1
   SPC = 1
   LOAD = 1
   DISPLACEMENT=ALL
$
BEGIN BULK
PARAM   COUPMASS 1
PARAM    WTMASS  .00259
PARAM    LGDISP  1
NLPARM   1      5              AUTO    5       25      PW      NO      +       A
+      A         .001    1.-7
PARAM,NMLOOP,5
$
EIGRL,10,,,3
PBARL    1      1              I                                       +       B
+      B 2.      1.      1.      .1      .1      .1
CBAR     1       1       1       2       0.      1.      0.
CBAR     2       1       2       3       0.      1.      0.
CBAR     3       1       3       4       0.      1.      0.
CBAR     4       1       4       5       0.      1.      0.
CBAR     5       1       5       6       0.      1.      0.
CBAR     6       1       6       7       0.      1.      0.
CBAR     7       1       7       8       0.      1.      0.
CBAR     8       1       8       9       0.      1.      0.
CBAR     9       1       9       10      0.      1.      0.
CBAR     10      1       10      11      0.      1.      0.
$
MAT1     1       1.+7            .3      .101
GRID     1               0.      0.      0.         345
GRID     2               10.     0.      0.         345
GRID     3               20.     0.      0.              345
GRID     4               30.     0.      0.              345
GRID     5               39.9999 0.      0.              345
GRID     6               49.9999 0.      0.              345
GRID     7               60.     0.      0.         345
GRID     8               70.     0.      0.         345
GRID     9               80.     0.      0.         345
GRID     10              90.     0.      0.              345
```

```
GRID     11               100.     0.       0.                345
LOAD     2       1.       1.       1
SPC1     1       1234     1
SPC1     1       234      11
FORCE    1       11       0        500.     1.       0.       0.
ENDDATA
```

Submit the input file for analysis

7. Submit the input file to MSC.Nastran for analysis.

   7a. To submit the MSC.Patran **.bdf** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran prob14b.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

   7b. To submit the MSC.Nastran **.dat** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran prob14b scr=yes**. Monitor the run using the UNIX **ps** command.

8. When the run is completed, edit the **prob14b.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing WARNING messages indicate modeling errors.

9. While still editing **prob14b.f06**, search for the word:

E I G E N (spaces are necessary)

What are the first three natural frequencies?

1st = ___________Hz

2nd = ___________Hz

3rd = ___________Hz

Comparison of Results

10. Compare the results obtained in the **.f06** file with the following results:

MODE NO.	EXTRACTION ORDER	EIGENVALUE	RADIANS	CYCLES	GENERALIZED MASS	GENE STI
1	1	2.735837E+04	1.654037E+02	2.632481E+01	1.000000E+00	2.735837
2	2	3.748482E+05	6.122484E+02	9.744236E+01	1.000000E+00	3.748482
3	3	1.816509E+06	1.347779E+03	2.145057E+02	1.000000E+00	1.81650

11. **MSC.Nastran Users should have finished this exercise. MSC.Patran Users should proceed to the next step.**

12. Proceed with the Reverse Translation process, that is importing the **prob14b.op2** results file into MSC.Patran. To do this, return to the Analysis form and proceed as follows:

◆ **Analysis**

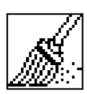
Action:	**Read Output2**
Object:	**Result Entities**
Method	**Translate**
Select Results File...	
Select Results File	**prob14b.op2**
OK	
Apply	

When the translation is complete bring up the **Results** form.

◆ **Results**

Action:	**Create**
Object:	**Deformation**
Select Results Cases	**Default, Mode 1:Freq=26.325**
Select Deformation Result	**Eigenvectors, Translational**
Apply	

To reset the graphics, click on this icon:



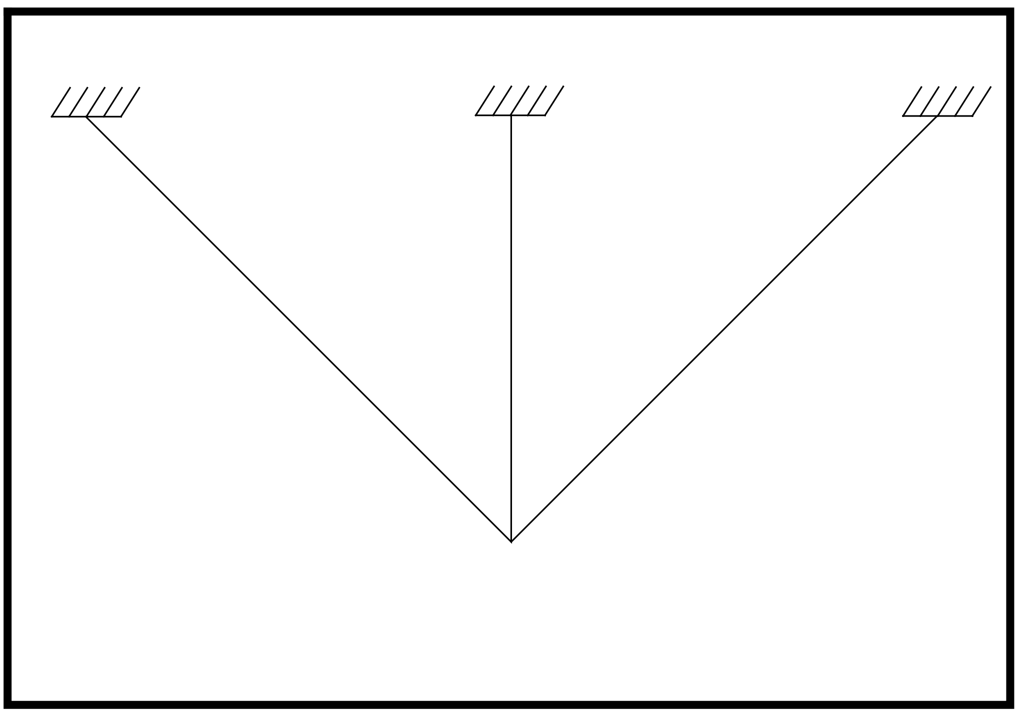
Reset Graphics

You can go back and select any *Results Case, Fringe Results or Deformation Results* you are interested in.

Quit MSC.Patran when you are finished with this exercise.

WORKSHOP 15

Weight Minimization of a Three Bar Truss



Objectives:

- Minimize the weight of the truss.
- First mode must be between 1500-1550 Hz.
- Submit the file for analysis in MSC.Nastran.
- Recover the desired objective while satisfying the frequency requirement.

Model Description:

You must minimize the weight of the following three bar truss problem. The first mode must be between 1500-1550 hz. The model will have different areas for the inside and outside beams. The structure must remain symetric.

Below is a Geometric representation of the truss. It also contains the loads and boundary constraints.

Figure 15.1-Loads and Boundary Conditions

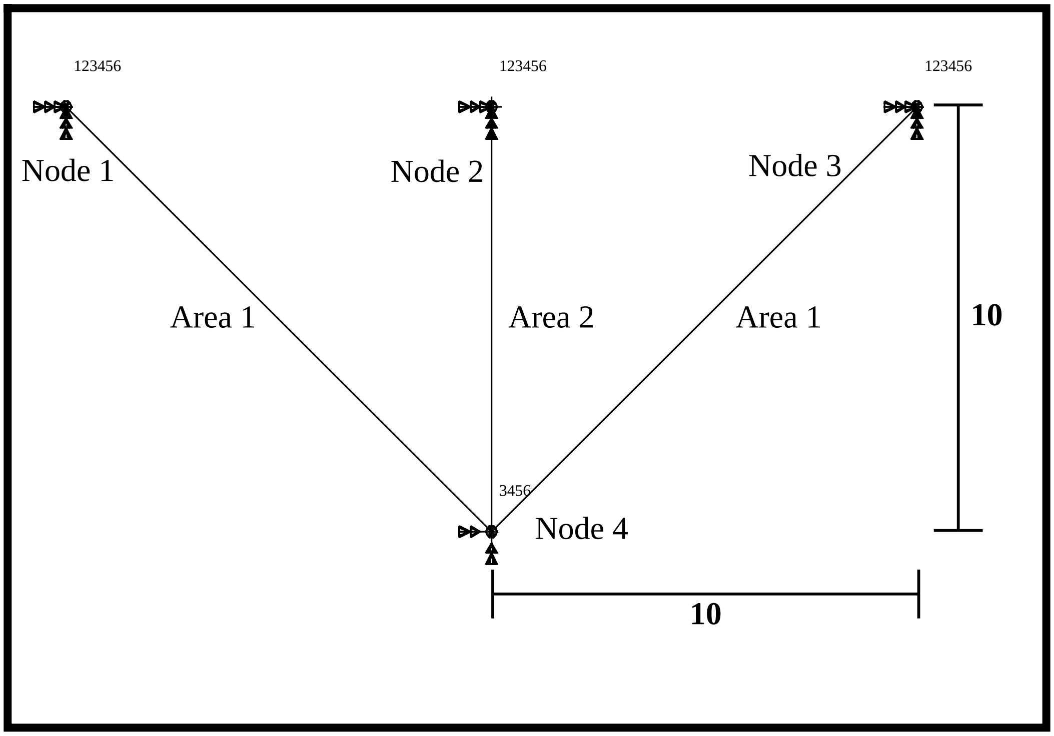


Table 15.1

Elastic Modulus	**10E6**
Poisson's Ratio	**.33**
Density	**.1**
Wt.-Mass Conversion	**.00259**
Area 1	**1.0**
Area 2	**2.0**

Optimization Statement

- **Design Variables**
  - These are the areas of the three rod elements (A1, A2, A3).
- **Objective**
  - Minimize the weight of the truss.
- **Subject to the follow constraints**
  - The first mode must be between 1500-1550 Hz.
  - A1=A3 to impose symmetry.

Suggested Exercise Steps:

- Generate the analysis model. The nodes (GRID) and element connectivities (CROD) should be defined manually.
- Define material (MAT1) and element (PROD) properties.
- Apply fixed boundary constraints (SPC1) to the upper nodes.
- Create the appropriate design optimization model.
- Define the design variables (DESVAR).
- Relate one design variable to another design variable (DLINK).
- Define design variable to analysis model parameter relations (DVPREL).
- Specify design sensitivity response quantities (DRESP1).
- Define constraints (DCONSTR).
- Define optimization control parameters (DOPTPRM).
- Prepare the model for linear static analysis and normal modes analysis using Lanczos Method.
    - PARAM, WTMASS, 0.00259
- Generate an input file and submit it to MSC.Nastran for structural optimization analysis.
- Review the results, specifically the eigenvalues and the design variable history.

```
ID SEMINAR,PROB15
```

```
CEND
```

```
BEGIN BULK
```

1	2	3	4	5	6	7	8	9	10

1	2	3	4	5	6	7	8	9	10

ENDDATA

Exercise Procedure:

1. Users who are not utilitizing MSC.Patran for generating an input file should go to Step 14, otherwise, proceed to step 2.

2. Create a new database called **prob15.db**.

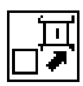
File/New...

New Database Name:	**prob15**
OK	

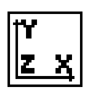
In the **New Model Preferences** form set the following:

Tolerance:	● **Default**
Analysis Code:	**MSC/NASTRAN**
Analysis Type:	**Structural**
OK	

3. Activate the entity labels by selecting the **Show Labels** button on the toolbar.

 **Show Labels**

4. Change to a front view by selecting the **Front View** button on the toolbar.

 **Front View**

Whenever possible click ❑ **Auto Execute** (turn off).

5. Create nodes.

◆ **Finite Elements**

Action:	**Create**
Object:	**Node**
Method:	**Edit**

❑ **Associate with Geometry**

Node Location List: **[-10 0 0]**

Apply

Node Location List: **[0 0 0]**

Apply

Node Location List: **[10 0 0]**

Apply

Node Location List: **[0 -10 0]**

Apply

6. Create bars.

◆ **Finite Elements**

Action: **Create**

Object: **Element**

Method: **Edit**

Shape: **Bar**

Node 1 = **Node 1**

Node 2 = **Node 4**

Apply

Node 1 = **Node 2**

Node 2 = **Node 4**

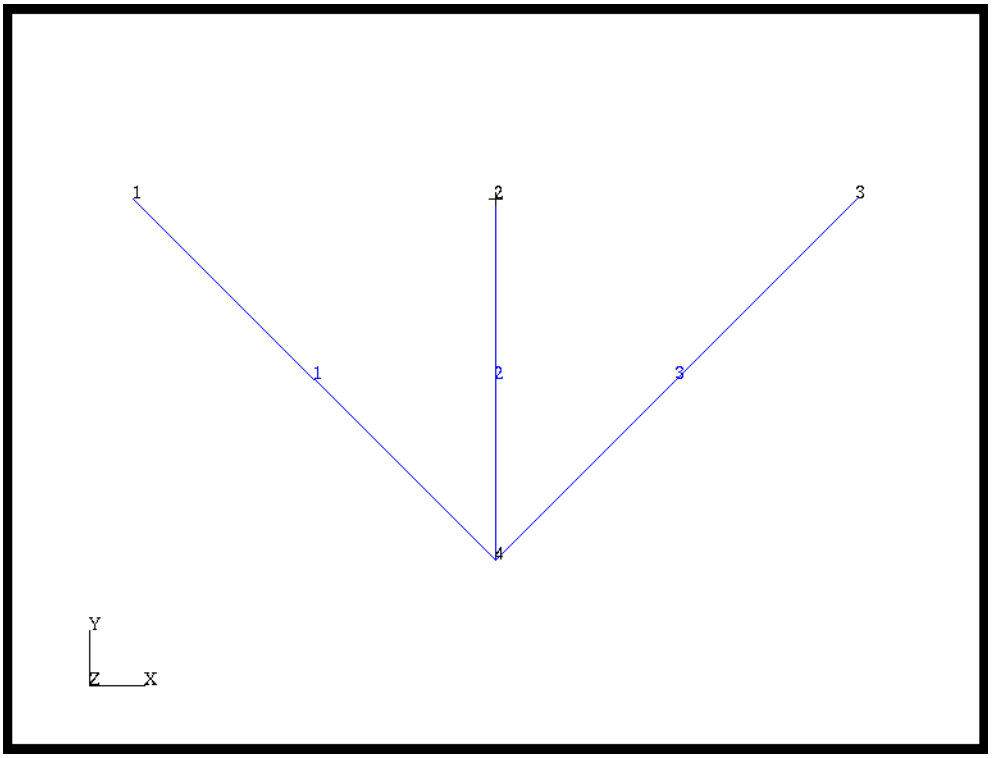
Apply

Node 1 = **Node 3**

Node 2 = **Node 4**

Apply

Figure 15.2-Nodes and Element Labels



7. Next, define a material using the specified modulus of elasticity and allowable stresses.

◆ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name:	**alum**
Input Properties...	
Elastic Modulus =	**10e6**
Poisson Ratio=	**0.33**
Density=	**0.1**
OK	

Apply

8. Next, reference the material that was created in the previous step.

◆ **Properties**

Action: **Create**

Dimension: **1D**

Type: **Rod**

Property Set Name: prop_1

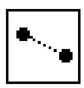
Input Properties...

Material Name: m:alum

Area: 1

OK

Pick **Select Members** and then click on the **Beam element** icon from the small menu window.



Beam element

Select Members: Elm 1 3

Add

Apply

Property Set Name: prop_2

Input Properties...

Area: 2

OK

Select Members: Elm 2

Add

Apply

9. Create nodal constraints.

◆ **Loads/BCs**

Action: **Create**

Object: **Displacement**

Type: **Nodal**

New Set Name: **disp_1**

Input Data...

Translations < T1 T2 T3 >: **< 0, 0, 0 >**

Rotations < R1 R2 R3 >: **< 0, 0, 0 >**

OK

Select Application Region...

Geometry Filter: ● **FEM**

Select Nodes: **Node 1:3**

Add

OK

Apply

New Set Name: **disp_2**

Input Data...

Translations < T1 T2 T3 >: **< , , 0 >**

Rotations < R1 R2 R3 >: **< 0, 0, 0 >**

OK

Select Application Region...

Geometry Filter: ● **FEM**

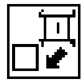
Select Nodes: **Node 4**

Add

OK

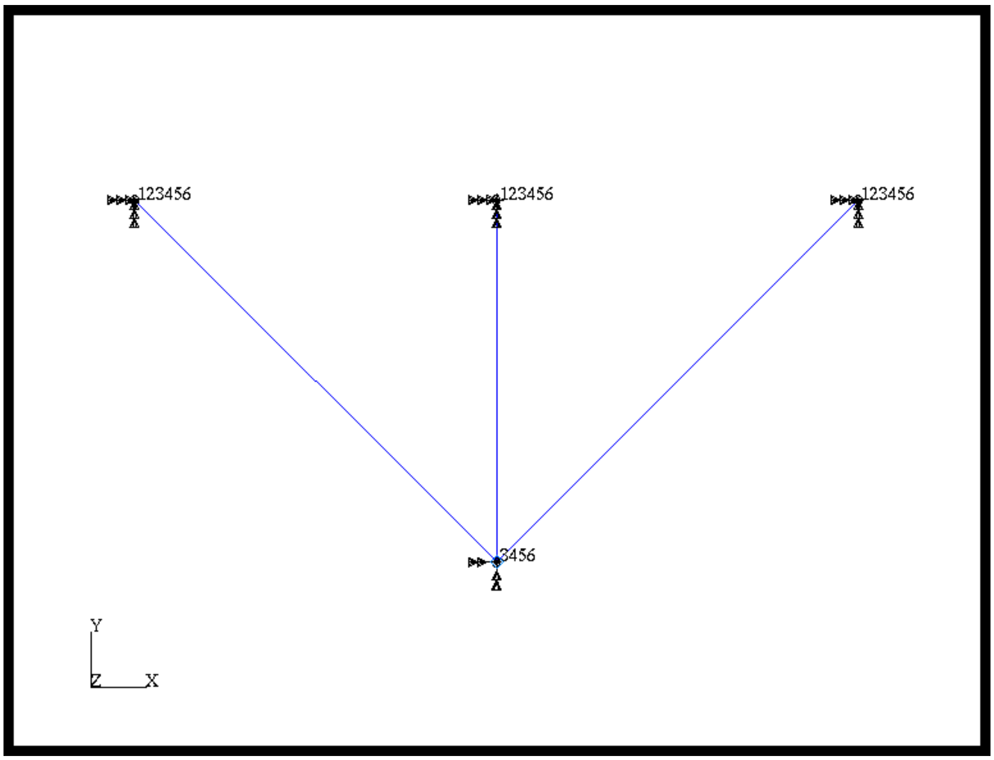
Apply

10. Activate the entity labels by selecting the **Hide Labels** button on the toolbar.



Hide Labels

Figure 15.3-Nodes, Element Labels, and Constraints



11. Create Load Cases.

◆ **Load Cases**

Action: **Create**

Load Case Name: **case_1**

Assign/Prioritize Loads/BCs

Select Individual Loads/BCs **Displ_disp_1**

Displ_disp_2

OK

Apply

12. Using **Tools** for **Model Variables**.

Tools/Model Variables...

Action:	**Create**
Object:	**Variable**
Method:	**Property**
Dimension:	**1D**
Type:	**Rod**
Select Property Set:	**prop_1**
Select Property Name:	**Area**

Apply

Select Property Set:	**prop_2**
Select Property Name:	**Area**

Apply

Close

13. Using **Tools** for **Design Study**.

Tools/Design Study...

Action:	**Create**
Object:	**Design Study**
Design Study Name:	**opt_1**

Design Variables...

	Lower Bound	*Upper Bound*
prop_1_Area:	**0.1**	**100**
prop_2_Area:	**0.1**	**100**

OK

Design Objective...

Existing Objectives: **Total_Weight**

Min/Max: **minimize**

OK

Design Constraints...

Action: **Create**

Solution: **Normal Modes**

Response: **Frequency**

Constraint Name: **FREQ_1**

Frequency Mode Number: **1**

Lower Bound (Hz): **1500**

Upper Bound (Hz): **1550**

Apply

Close

Apply

Close

Now you are ready to generate an input file for analysis.

14. Click on the **Analysis** radio button on the **Top Menu Bar** and complete the entries as shown here.

◆ **Analysis**

Action: **Optimize**

Object: **Entire Model**

Method: **Analysis Deck**

Job Name: **prob15**

Translation Parameters...

Data Output:	**OP2 and Print**
OK	
Optimization Parameters...	
Mass Calculation	**Coupled**
Wt. -Mass Conversion =	**.00259**
Maximum Number of Design cycles (DESMAX) =	**30**
Print Design Data (P1) every n-th cycle where n =	**1**
Print Analysis Results (NASPRT) every n-th cycle where n =	**1**
OK	
Subcase Create...	
Solution Type:	**103 Normal Modes**
Available Subcases:	**case_1**
Constraints in Current Subcase:	**FREQ_1**
Apply	
Cancel	
Subcase Select...	
Solution Type:	**103 Normal Modes**
Subcases Available:	**case_1**
OK	
Apply	

An MSC.Nastran input file called **prob15.bdf** will be generated. This process of translating your model into an input file is called the Forward Translation. The Forward Translation is complete when the Heartbeat turns green. MSC/NATRAN users should proceed to **Step 15**.

Generating an input file for MSC.Nastran Users:

MSC.Nastran users can generate an input file using the data from page 15-3 (Model Description). The result should be similar to the output below.

15. MSC.Nastran input file: **prob15.dat**

```
ASSIGN OUTPUT2='prob15.op2', UNIT=12
ID NAS102, WORKSHOP 15
TIME 10
SOL 200            $ OPTIMIZATION
CEND
TITLE= SYMMETRIC THREE BAR TRUSS DESIGN OPTIMIZATION - VARIATION OF
D200X1
SUBTITLE= GOAL IS TO MIN WT WHILE KEEPING THE 1ST MODE BETWEEN 1500-1550 HZ
ECHO= SORT
SPC= 100
DISP(PLOT) ALL
DESOBJ(MIN)= 100   $ (DESIGN OBJECTIVE = DRESP ID)
DESSUB= 200        $  DEFINE CONSTRAINT SET FOR BOTH SUBCASES
SUBCASE 1
  ANALYSIS= MODES
  METHOD= 10
BEGIN BULK
$
$---------------------------------------------------------------------
--
$ ANALYSIS MODEL
$---------------------------------------------------------------------
--
$
EIGRL, 10, , , 2
PARAM, POST, -1
PARAM, PATVER, 3.0
$
$ GRID DATA
$      2       3       4      5       6       7      8       9       10
GRID,   1,       ,      -10.0,    0.0,  0.0
GRID,   2,       ,        0.0,    0.0,  0.0
GRID,   3,       ,       10.0,    0.0,  0.0
GRID,   4,       ,        0.0,  -10.0,  0.0
$ SUPPORT DATA
SPC,    100,    1,      123456,  ,      2,      123456
SPC,    100,    3,      123456,  ,      4,      3456
$ ELEMENT DATA
CROD,   1,      11,     1,       4
CROD,   2,      12,     2,       4
```

```
CROD,   3,      13,     3,       4
$ PROPERTY DATA
PROD,   11,     1,      1.0
PROD,   12,     1,      2.0
PROD,   13,     1,      1.0
MAT1,   1,      1.0E+7, ,       0.33,   0.1
$
PARAM, WTMASS, .00259
$
$----------------------------------------------------------------------
$ DESIGN MODEL
$----------------------------------------------------------------------
$
$...DESIGN VARIABLE DEFINITION
$
$DESVAR,ID,     LABEL,  XINIT,  XLB,    XUB,    DELXV(OPTIONAL)
DESVAR, 1,      A1,     1.0,    0.1,    100.0
DESVAR, 2,      A2,     2.0,    0.1,    100.0
DESVAR, 3,      A3,     1.0,    0.1,    100.0
$
$...IMPOSE X3=X1 (LEADS TO A3=A1)
$
$DLINK, ID,     DDVID,  CO,     CMULT,  IDV1,   C1,     IDV2,   C2,     +
$+,     IDV3,   C3,     ...
DLINK,  1,      3,      0.0,    1.0,    1       1.00
$
$...DEFINITION OF DESIGN VARIABLE TO ANALYSIS MODEL PARAMETER RELATIONS
$
$DVPREL1,ID,    TYPE,   PID,    FID,    PMIN,   PMAX,   CO,     ,       +
$+,     DVID1,  COEF1,  DVID2,  COEF2,  ...
DVPREL1, 10,    PROD,   11,     4,      ,       ,       ,       ,       +DP1
+DP1,   1,      1.0
DVPREL1, 20,    PROD,   12,     4,      ,       ,       ,       ,       +DP2
+DP2,   2,      1.0
DVPREL1, 30,    PROD,   13,     4,      ,       ,       ,       ,       +DP3
+DP3,   3,      1.0
$
$...STRUCTURAL RESPONSE INDENTIFICATION
$
$DRESP1 ID      LABEL   RTYPE   PTYPE   REGION  ATTA    ATTB    ATT1    +
$+      ATT2    ...
DRESP1  100     W       WEIGHT
DRESP1  210     MODE1   EIGN                    1
```

```
$
$...CONSTRAINTS
$
$DCONSTR,DCID,  RID,    LALLOW, UALLOW
DCONSTR, 200,   210,    8.883E7, 9.485E7
$
$...OPTIMIZATION CONTROL
$
DOPTPRM, DESMAX, 30
$
$.......2.......3.......4.......5.......6.......7.......8.......9.......0
ENDDATA
```

Submitting the input file for analysis:

16. Submit the input file to MSC.Nastran for analysis.

    16a. To submit the MSC.Patran **.bdf** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran prob15.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

    16b. To submit the MSC.Nastran **.dat** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran prob15 scr=yes**. Monitor the run using the UNIX **ps** command.

17. Edit the **prob15.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing WARNING messages indicate modeling errors.

18. While still editing **prob15.f06**, search for the word:

DESIGN VARIABLE HISTORY

Design Variable	Initial Value	Optimization Value	Iteration Value
1	________	____________	__________
2	________	____________	__________
3	________	____________	__________

Comparison of Results

19. Compare the results obtained in the .f06 file with the results on the following page::

```
**************************************************************
  S U M M A R Y   O F   D E S I G N   C Y C L E   H I S T O R Y
**************************************************************
```

(HARD CONVERGENCE ACHIEVED)

(SOFT CONVERGENCE ACHIEVED)

NUMBER OF FINITE ELEMENT ANALYSES COMPLETED 13
NUMBER OF OPTIMIZATIONS W.R.T. APPROXIMATE MODELS 12

OBJECTIVE AND MAXIMUM CONSTRAINT HISTORY

CYCLE NUMBER	OBJECTIVE FROM APPROXIMATE OPTIMIZATION	OBJECTIVE FROM EXACT ANALYSIS	FRACTIONAL ERROR OF APPROXIMATION	MAXIMUM VALUE OF CONSTRAINT
INITIAL		4.828427E+00	1.195946E-01	
1	4.601859E+00	4.602146E+00	-6.247799E-05	7.284999E-03
2	3.726671E+00	3.726556E+00	3.090150E-05	1.419008E-03
3	2.981427E+00	2.981173E+00	8.501315E-05	1.431373E-03
4	2.390100E+00	2.390151E+00	-2.144634E-05	2.942288E-04
5	1.917196E+00	1.917195E+00	4.974322E-07	6.048387E-05
6	1.532210E+00	1.532388E+00	-1.162230E-04	-4.593781E-04
7	1.227819E+00	1.227710E+00	8.884553E-05	-1.306546E-04
8	9.821384E-01	9.821677E-01	-2.985792E-05	-1.306546E-04
9	7.867956E-01	7.866402E-01	1.976112E-04	5.399446E-04
10	6.290510E-01	6.290196E-01	4.984271E-05	2.694998E-04
11	5.992411E-01	5.992462E-01	-8.554079E-06	1.614478E-05
12	5.992462E-01	5.992462E-01	0.000000E+00	1.614478E-05

DESIGN VARIABLE HISTORY

INTERNAL DV. ID.	EXTERNAL DV. ID.	LABEL	INITIAL :	1 :	2 :	3 :	4 :	5 :
1	1	PROP_1:1	1.0000E+00 :	7.7875E-01 :	6.2355E-01 :	4.9884E-01 :	3.9907E-01 :	3.1996E-01 :
2	2	PROP_2:2	2.0000E+00 :	2.3995E+00 :	1.9629E+00 :	1.5702E+00 :	1.2614E+00 :	1.0122E+00 :

INTERNAL DV. ID.	EXTERNAL DV. ID.	LABEL	6 :	7 :	8 :	9 :	10 :	11 :
1	1	PROP_1:1	2.5549E-01 :	2.0482E-01 :	1.6385E-01 :	1.3140E-01 :	1.0502E-01 :	1.0000E-01 :
2	2	PROP_2:2	8.0977E-01 :	6.4840E-01 :	5.1872E-01 :	4.1497E-01 :	3.3198E-01 :	3.1640E-01 :

INTERNAL DV. ID.	EXTERNAL DV. ID.	LABEL	12 :	13 :	14 :	15 :	16 :	17 :
1	1	PROP_1:1	2.5549E-01 :					
2	2	PROP_2:2	8.0977E-01 :	:				

20. MSC.Nastran Users have finished this exercise. MSC.Patran Users should proceed to the next step.

21. Proceed with the Reverse Translation process, that is, importing the **prob15.op2** results file into MSC.Patran. To do this, return to the **Analysis** form and proceed as follows:

◆ **Analysis**

Action:	**Read Output 2**
Object:	**Result Entities**
Method:	**Translate**

Select Results File...

Selected Results File:	**prob15.op2**

OK

Apply

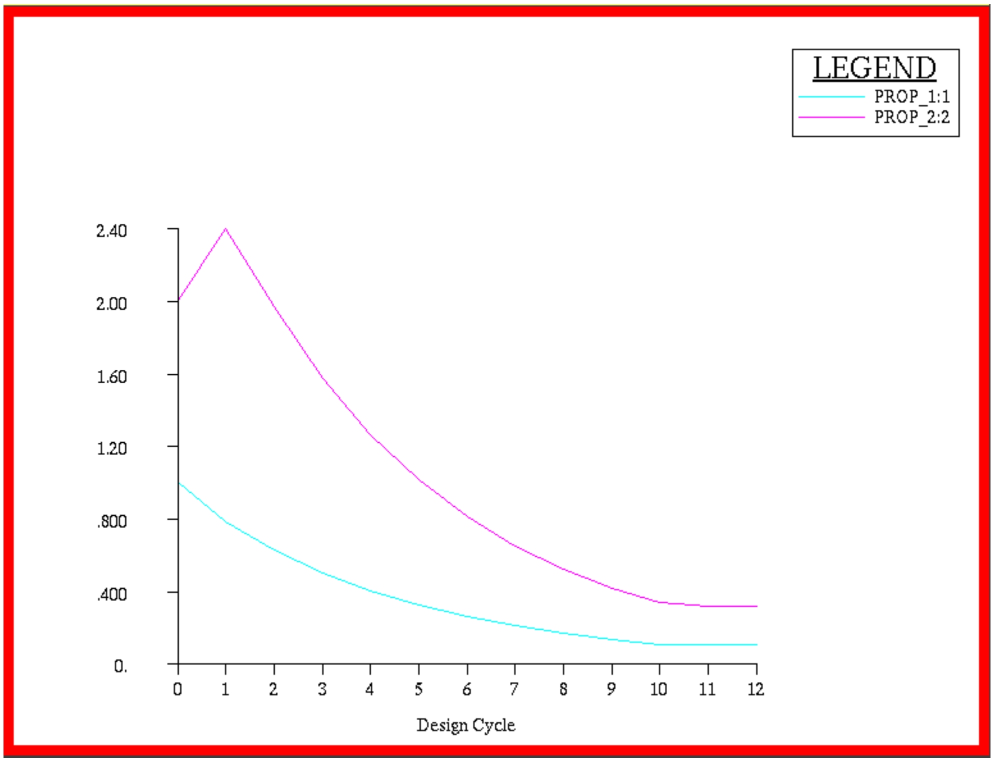
22. Plot **Select Result Graph**.

◆ **XY Plot**

Action:	**Post**
Object:	**XY Window**
Select Current XY Window:	**DesignVariableHistory**
Post/Unpost XY Windows:	**DesignVariableHistory**

Apply

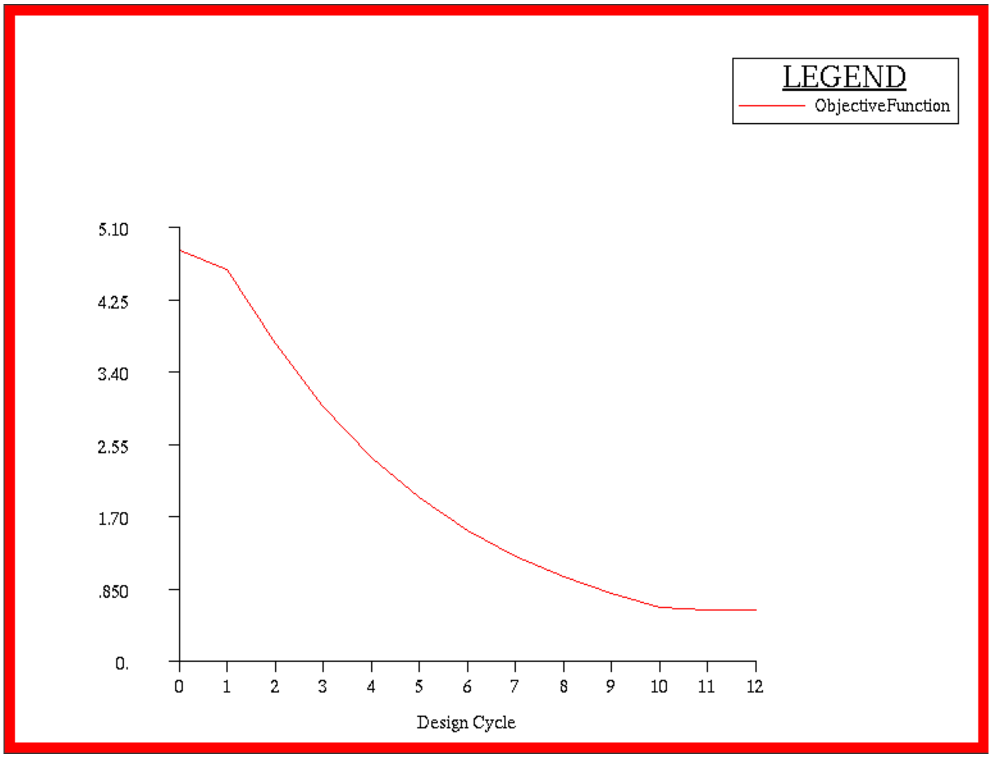
Figure 2.1 - Design Variable History



Action:	**Post**
Object:	**XY Window**
Select Current XY Window:	**ObjectiveFunctionHistory**
Post/Unpost XY Windows:	**ObjectiveFunctionHistory**

Apply

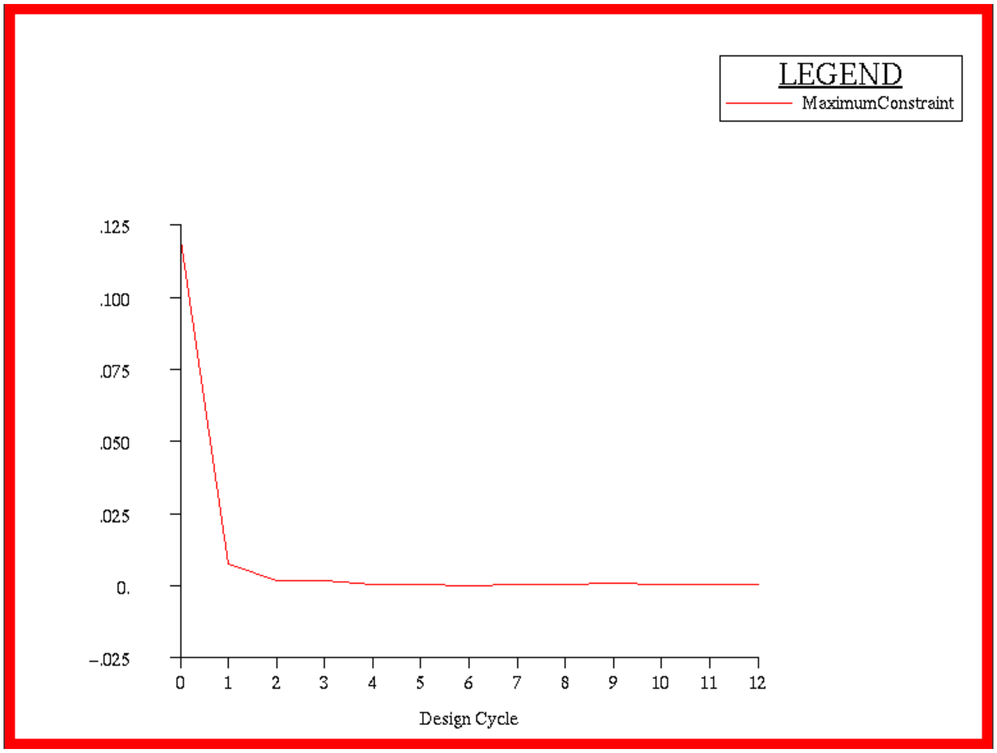
Figure 2.2 - Objective Function History



Action:	**Post**
Object:	**XY Window**
Select Current XY Window:	**MaximumConstraintHistory**
Post/Unpost XY Windows:	**MaximumConstraintHistory**

Apply

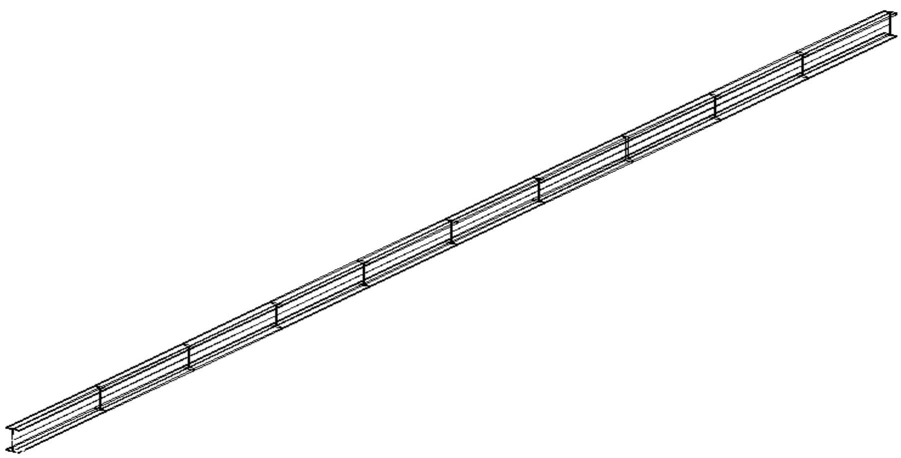
Figure 2.3 - Maximum Constraint History



Quit MSC.Patran when you have completed this exercise.

APPENDIX 1a

Modal Analysis of a Beam (SI Units)



Objectives

- Perform normal modes analysis of a cantilever beam.
- Submit the file for analysis in MSC.Nastran.
- Find the first three natural frequencies and mode shapes of the beam.

Model Description:

The goal of this example is to find the first 3 modes of a beam pinned at both ends.

Figure A-1a.1 below is a finite element representation of the beam. One end is contrained in all translations and the other is free to move in the X. Both ends are held in the X-rotation.

Figure A-1a.1-Grid Coordinates and Element Connectivities

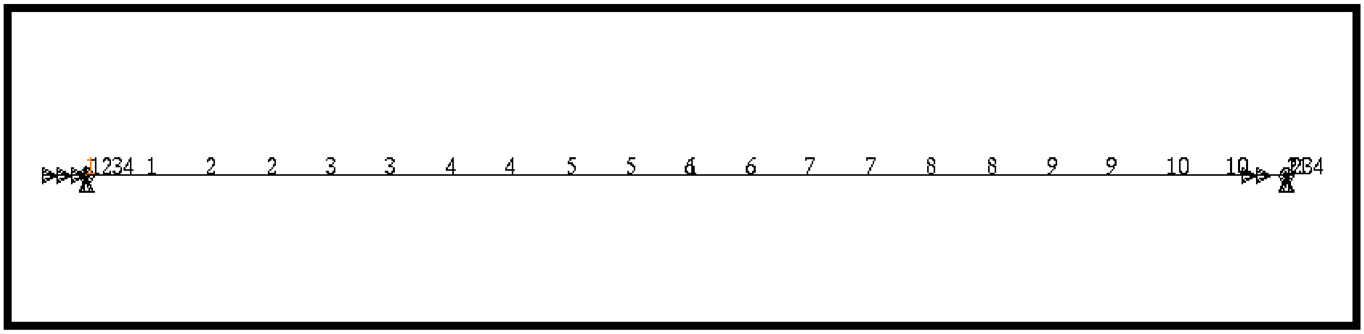


Table A-1a.1

Length	**1.0×10^3 mm**
Elastic Modulus	**2.0684×10^5 MPa**
Density	**7.8334×10^{-9} N-sec^2/mm^4**
Poisson's Ratio	**0.32**
Area	**5×10^3 mm^2**
I_1	**1.0417×10^6 mm^4**
I_2	**1.0 mm^4**
Force	**1×10^7 N**

Hand Calculations

$$f_n = \frac{K_n}{2\pi}\left[\frac{EIg}{Wl^4}\right]^{1/2}$$

$$f_n = K_n\left(\frac{1}{2\pi}\left[\frac{2.0684\times10^5(1.0417\times10^6)}{7.8334\times10^{-9}(5\times10^3)(1.0\times10^3)^4}\right]^{1/2}\right)$$

$$f_n = K_n(11.805)$$

From Theory

Mode	**K_n**	**f_n**
1	**9.87**	**116.51 Hz**
2	**39.5**	**466.28 Hz**
3	**88.8**	**1048.28 Hz**

Suggested Exercise Steps

- Explicitly generate a finite element representation of the beam structure. (i.e., the grids (GRID) and element connectivities (CBAR) should be defined manually.)
- Define material (MAT1) and element (PBAR) properties.
- Apply the fixed boundary constraints (SPC1).
- Prepare the model for a normal modes analysis (SOL 103 and PARAMS).
    - PARAM, COUPMASS, 1
    - EIGRL (To select Lanczos)
- Generate an input file and submit it to the MSC.Nastran solver for normal modes analysis.
- Review the results, specifically the eigenvalues.

```
ID SEMINAR,PROB1
```

```
CEND
```

```
BEGIN BULK
```

1	2	3	4	5	6	7	8	9	10

1	2	3	4	5	6	7	8	9	10

ENDDATA

Exercise Procedure:

1. Users who are not utilizing MSC.Patran for generating an input file should go to Step 10, otherwise, proceed to step 2.

2. Create a new database named **probap1.db**.

   File/New Database

   New Database Name **probap1**

   OK

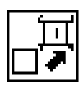
   In the *New Model Preference* form set the following:

   Tolerance ◆ **Default**

   Analysis Code: **MSC/NASTRAN**

   OK

3. Activate the entity labels by selecting the Show Labels icon on the tool-bar.

   Show Labels

4. Create a curve.

   ◆ **Geometry**

Action:	**Create**
Object:	**Curve**
Method:	**XYZ**
Vector Coordinates List	**<1000, 0, 0>**
Origin Coordinates List	**[0, 0, 0]**

   Apply

5. Create the finite element model and mesh the surface.

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Curve**
Global Edge Length	**100**
Curve List	**Curve 1**

Apply

6. Create nodal displacements.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Displacement**
Type:	**Nodal**
New Set Name	**disp1**

Input Data...

Translations <T1 T2 T3>	**<0 0 0>**
Rotations <R1 R2 R3>	**<0 , , >**

OK

Select Application Region...

■ **Geometry**

Select Geometry Entities	**Point 1**

Add

OK

Apply

New Set Name	**disp2**

Input Data...

Translations <T1 T2 T3>	**< , 0 0>**

Rotations <R1 R2 R3> **<0 , , >**

OK

Select Application Region...

◆ **Geometry**

Select Geometry Entities **Point 2**

Add

OK

Apply

New Set Name **permanent_constraint**

Input Data...

Translations <T1 T2 T3> **< , , 0>**

Rotations <R1 R2 R3> **<0 , 0 , >**

OK

Select Application Region...

Select Geometry Entities **Curve 1**

Add

OK

Apply

7. Create a set of material properties for the bar.

◆ **Materials**

Action: **Create**

Object: **Isotropic**

Method: **Manual Input**

Material Name **mat_1**

Input Properties...

Elastic Modulus = **2.0684E5**

Poisson Ratio = **0.32**

Density =	**7.8334E-9**
Apply	
Cancel	

8. Define the bar properties.

◆ **Properties**

Action:	**Create**
Dimension:	**1D**
Type:	**Beam**
Property Set Name	**bar**
Input Properties...	
Material Name *(Select from **Material Property Sets** box)*	**m:mat_1**
Bar Orientation	**Coord 0.2**
Area	**5E3**
[Inertia 1,1]	**1.0417E6**
[Inertia 1,2]	**1.0**
OK	
Select Members	**Curve 1**
Add	
Apply	

9. Now, you will generate the input file for analysis.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method	**Analysis Deck**
Job Name	**probap1**
Solution Type...	

Solution Type:	**◆ NORMAL MODES**
Solution Parameters ...	
<deselect Automatic Constraints>	**❒ Automatic Constraints**
Mass Calculation:	**Coupled**
Data Deck Echo:	**None**
OK	
OK	
Subcase Create...	
Available Subcases	**Default**
Subcase Parameters...	
Number of Desired Roots =	**3**
OK	
Apply	
Cancel	*<to close form>*
Apply	

An MSC.Nastran input file called **probap1.bdf** will be generated. The process of translating your model into an input file is called Forward Translation. The Forward Translation is complete when the Heartbeat turns green. MSC.Patran Users should proceed to step 11.

Generating an input file for MSC.Nastran Users:

MSC.Nastran users can generate an input file using the data from Table A-1a.1. The result should be similar to the output below.

10. MSC.Nastran Input File: **probap1.dat**

```
SOL 103
TIME 600
CEND
TITLE = Normal Modes Example (SI UNITS)
SUBCASE 1
   METHOD = 1
   SPC = 1
   VECTOR(SORT1,REAL)=ALL
BEGIN BULK
PARAM    COUPMASS1
EIGRL    1                        3       0
PBAR     1       1       5000.    1.04+6
CBAR     1       1       1        2       0.      1.      0.
CBAR     2       1       2        3       0.      1.      0.
CBAR     3       1       3        4       0.      1.      0.
CBAR     4       1       4        5       0.      1.      0.
CBAR     5       1       5        6       0.      1.      0.
CBAR     6       1       6        7       0.      1.      0.
CBAR     7       1       7        8       0.      1.      0.
CBAR     8       1       8        9       0.      1.      0.
CBAR     9       1       9        10      0.      1.      0.
CBAR     10      1       10       11      0.      1.      0.
MAT1     1       206840.          .32     7.83-9
GRID     1               0.       0.      0.              345
GRID     2               100.000 0.       0.              345
GRID     3               200.000 0.       0.              345
GRID     4               300.000 0.       0.              345
GRID     5               400.000 0.       0.              345
GRID     6               500.     0.      0.              345
GRID     7               600.000 0.       0.              345
GRID     8               700.000 0.       0.              345
GRID     9               800.000 0.       0.              345
GRID     10              900.000 0.       0.              345
GRID     11              1000.    0.      0.              345
SPC1     1       1234    1
SPC1     1       234     11
ENDDATA
```

Submit the input file for analysis

11. Submit the input file to MSC.Nastran for analysis.

    11a. To submit the MSC.Patran **.bdf** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran probap1.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

    11b. To submit the MSC.Nastran **.dat** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran probap1 scr=yes**. Monitor the run using the UNIX **ps** command.

12. When the run is completed, edit the **probap1.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing WARNING messages indicate modeling errors.

13. While still editing **probap1.f06**, search for the word:

E I G E N (spaces are necessary)

What are the first three modes?

1st = ___________Hz

2nd = ___________Hz

3rd = ___________Hz

Comparison of Results

14. Compare the results obtained in the **.f06** file with the following results:

R E A L   E I G E N V A L U E S

MODE NO.	EXTRACTION ORDER	EIGENVALUE	RADIANS	CYCLES	GENERALIZED MASS	GENERALIZED STIFFNESS
1	1	5.352166E+05	7.315850E+02	1.164354E+02	1.000000E+00	5.352166E+05
2	2	8.561551E+06	2.926013E+03	4.656894E+0	1.000000E+00	8.561551E+06
3	3	4.329484E+07	6.579882E+03	1.047221E+03	1.000000E+00	4.329484E+07

15. MSC.Nastran Users have finished this exercise. MSC.Patran Users should proceed to the next step.

16. Proceed with the Reverse Translation process, that is importing the **probap1.op2** results file into MSC.Patran. To do this, return to the Analysis form and proceed as follows:

◆ **Analysis**

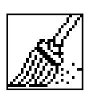
Action:	**Read Output2**
Object:	**Result Entities**
Method	**Translate**
Select Results File...	
Select Results File	**probap1.op2**
OK	
Apply	

When the translation is complete bring up the **Results** form.

◆ **Results**

Action:	**Create**
Object:	**Deformation**
Select Results Cases	**Default, Mode 1:Freq=116.44**
Select Deformation Result	**Eigenvectors, Translational**
Apply	

To reset the graphics, click on this icon:

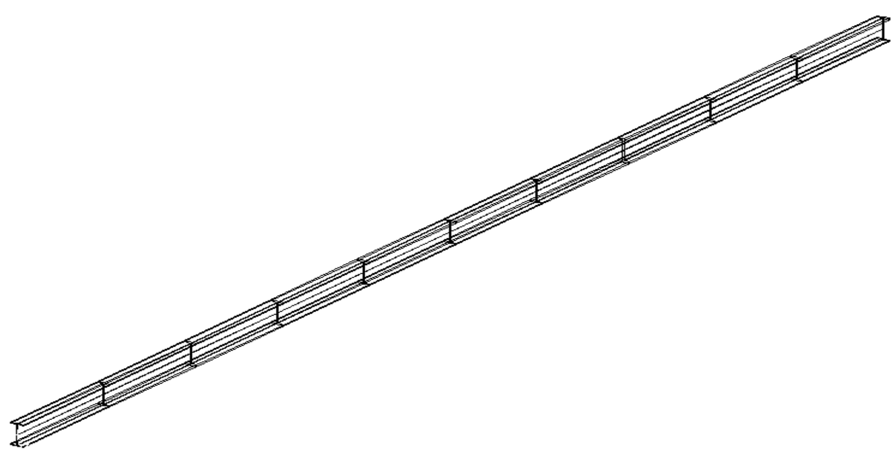
 **Reset Graphics**

You can go back and select any *Results Case*, *Fringe Results or Deformation Results* you are interested in.

Quit MSC.Patran when you are finished with this exercise.

Appendix 1b

Normal Modes with Differential Stiffnes (SI Units)



Objectives

- Analyze a stiffened beam for normal modes.
- Produce a MSC.Nastran input file that represent beam and load
- Submit for analysis.
- Find normal modes (natural frequencies).

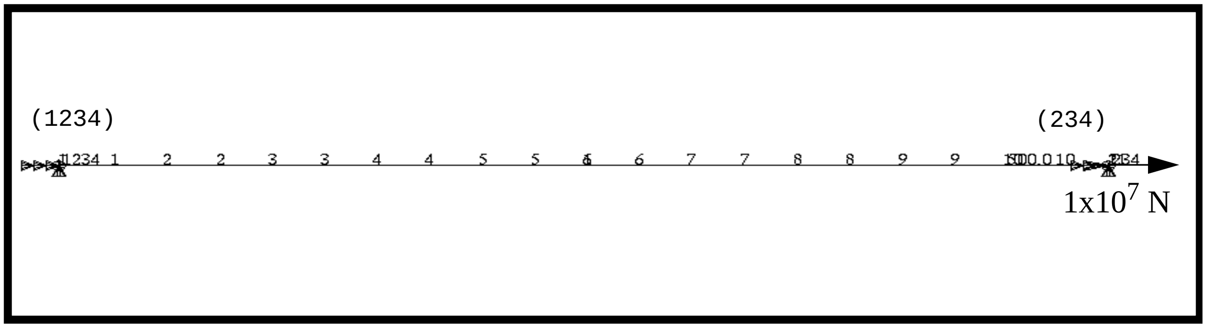
Model Description:

The goal of this example is to analyze a stiffened model. In this case, the beam from Appendix 1a. with a $1x10^7$ N force applied.

This is no longer a simple normal modes analysis. Instead we will be using a nonlinear static solution (SOL 106) with (PARAM, NMLOOP and METHOD and EIGRL).

Figure A-1b.1 below is a finite element representation of the beam. One end is pinned in 3 translations and one rotation. The other is pinned in 2 translations and one rotation with a $1x10^7$ N force applied.

Figure A-1b.1-Grid Coordinates and Element Connectivities



heoretical Solution

$$f_n = \frac{K_n}{2\pi}\left[\frac{EIg}{Wl^4}\left(1 + \frac{1}{Kr}\frac{Pl^2}{EI}\right)\right]^{1/2}$$

For Mode 1, Kr =9.87

$$fn = \frac{9.87}{2\pi}\left[\frac{(2.0684\times10^{5})(1.0417\times10^{6})}{(7.8334\times10^{-9})(5\times10^{3})(1.0\times10^{3})^{4}}\mathrm{x}\left(1 + \frac{1}{9.87}\frac{(1\times10^{7})(1\times10^{3})^{2}}{(2.0684\times10^{5})(1.0417\times10^{6})}\right)\right]^{1/2}$$

$$f_n = 278.22Hz$$

For Static Load

$$\Delta = \frac{PL}{AE}$$

$$\Delta = \frac{(1\times10^{7})(1\times10^{3})}{(5\times10^{3})(2.0684\times10^{5})}$$

$$\Delta = 9.67mm$$

Table A-1b.1

Length	**1.0×10^3 mm**
Elastic Modulus	**2.0684×10^5 MPa**
Density	**7.8334×10^{-9}N-sec^2/mm^4**
Poisson's Ratio	**0.32**
Area	**5×10^3 mm^2**
I_1	**1.0417×10^6 mm^4**
I_2	**1.0 mm^4**
Force	**1×10^7 N**

Suggested Exercise Steps

- Open database created in Problem 1a in order to modify it, adding a load and reanalyze.
- Create $1x10^7$ N force applied at one end (FORCE).
- Make sure analysis is set to nonlinear static (SOL 106).
- Prepare nonlinear analysis to also analyze for normal mode (PARAM NMLOOP, EIGRL, PARAM LGDISP, NLPARM).
- Review the results, specifically the eigenvectors.

```
ID SEMINAR,PROB1
```

```
CEND
```

```
BEGIN BULK
```

1	2	3	4	5	6	7	8	9	10

1	2	3	4	5	6	7	8	9	10

ENDDATA

Exercise Procedure:

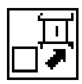
1. Users who are not utilizing MSC.Patran for generating an input file should go to Step 6, otherwise, proceed to step 2.

2. Open database created in Appendix Problem 1a named **probap1.db**.

   File/Open Database

Existing Database Name	**probap1**
OK	

3. Activate the entity labels by selecting the Show Labels icon on the tool-bar.

   
   Show Labels

4. Create force.

   ◆ **Loads/BCs**

Action:	**Create**
Object:	**Force**
Type:	**Nodal**
New Set Name	**pull**
Input Data...	
Force <F1 F2 F3>	**<1e7, , >**
OK	
Select Application Region...	
Select Geometry Entities	**Point 2**
Add	
OK	
Apply	

5. Now, you will generate the input file for analysis.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method	**Analysis Deck**
Job Name	**probap1b**
Solution Type...	
Solution Type:	◆ **NONLINEAR STATIC**
Solution Parameters ...	
<deselect Automatic Constraints>	❐ **Automatic Contraints**
Mass Calculation:	**Coupled**
Data Deck Echo:	**None**
OK	
OK	
Direct Text Input...	
◆ **Case Control Section**	**METHOD = 10**
◆ **Bulk Data Section**	**PARAM, NMLOOP, 5** **EIGRL, 10, , , 3**
OK	
Subcase Create...	
Available Subcases	**Default**
Subcase Parameters...	
Number of Load Increments =	**5**
OK	
Apply	
Cancel	
Apply	

An MSC.Nastran input file called **probap1b.bdf** will be generated. The process of translating your model into an input file is called Forward Translation. The Forward Translation is complete when the Heartbeat turns green. MSC.Patran Users should proceed to step 7.

Generating an input file for MSC.Nastran Users:

MSC.Nastran users can generate an input file using the data from Table A-1b.1. The result should be similar to the output below.

6. MSC.Nastran Input File: **probap1b.dat**

```
SOL 106
TIME 600
CEND
$
TITLT = NORMAL MODES WITH DIFFERENTIAL STIFFNESS
METHOD = 10
SUBCASE 1
   NLPARM = 1
   SPC = 1
   LOAD = 1
   DISPLACEMENT(SORT1,REAL)=ALL
$
BEGIN BULK
PARAM   COUPMASS 1
PARAM    LGDISP  1
NLPARM   1       5               AUTO    5       25      PW      NO      +       A
+      A         .001    1.-7
PARAM,NMLOOP,5
$
EIGRL,10,,,3
PBAR     1       1       5000.   1.04+6
CBAR     1       1       1       2       0.      1.      0.
CBAR     2       1       2       3       0.      1.      0.
CBAR     3       1       3       4       0.      1.      0.
CBAR     4       1       4       5       0.      1.      0.
CBAR     5       1       5       6       0.      1.      0.
CBAR     6       1       6       7       0.      1.      0.
CBAR     7       1       7       8       0.      1.      0.
CBAR     8       1       8       9       0.      1.      0.
CBAR     9       1       9       10      0.      1.      0.
CBAR     10      1       10      11      0.      1.      0.
$
MAT1     1       206840.         .32     7.83-9
GRID     1               0.      0.      0.              345
GRID     2               100.000 0.      0.              345
```

```
GRID    3               200.000 0.      0.              345
GRID    4               300.000 0.      0.              345
GRID    5               400.000 0.      0.              345
GRID    6               500.    0.      0.              345
GRID    7               600.000 0.      0.              345
GRID    8               700.000 0.      0.              345
GRID    9               800.000 0.      0.              345
GRID    10              900.000 0.      0.              345
GRID    11              1000.   0.      0.              345
SPC1    1       1234    1
SPC1    1       234     11
FORCE   1       11      0       1.+7    1.      0.      0.
ENDDATA
```

Submit the input file for analysis

7. Submit the input file to MSC.Nastran for analysis.

   7a. To submit the MSC.Patran **.bdf** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran probap1b.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

   7b. To submit the MSC.Nastran **.dat** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran probap1b scr=yes**. Monitor the run using the UNIX **ps** command.

8. When the run is completed, edit the **probap1b.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing WARNING messages indicate modeling errors.

9. While still editing **probap1b.f06**, search for the word:

E I G E N (spaces are necessary)

What are the first three natural frequencies?

1st = __________Hz

2nd = __________Hz

3rd = __________Hz

Comparison of Results

10. Compare the results obtained in the **.f06** file with the following results:

R E A L   E I G E N V A L U E S

MODE NO.	EXTRACTION ORDER	EIGENVALUE	RADIANS	CYCLES	GENERALIZED MASS	GENERAL STIFFN
1	1	3.056203E+06	1.748200E+03	2.782346E+02	1.000000E+00	3.05620
2	2	1.864932E+07	4.318486E+03	6.873084E+02	1.000000E+00	1.86493
3	3	6.517956E+07	8.073386E+03	1.284919E+03	1.000000E+00	6.51795

11. MSC.Nastran Users have finished this exercise. MSC.Patran Users should proceed to the next step.

12. Proceed with the Reverse Translation process, that is importing the **probap1b.op2** results file into MSC.Patran. To do this, return to the Analysis form and proceed as follows:

◆ **Analysis**

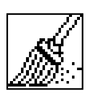
Action:	**Read Output2**
Object:	**Result Entities**
Method	**Translate**
Select Results File...	
Select Results File	**probap1b.op2**
OK	
Apply	

When the translation is complete bring up the ***Results*** form.

◆ **Results**

Action:	**Create**
Object:	**Deformation**
Select Results Cases	**Default, Mode 1:Freq=278.23**
Select Deformation Result	**Eigenvectors, Translational**
Apply	

To reset the graphics, click on this icon:



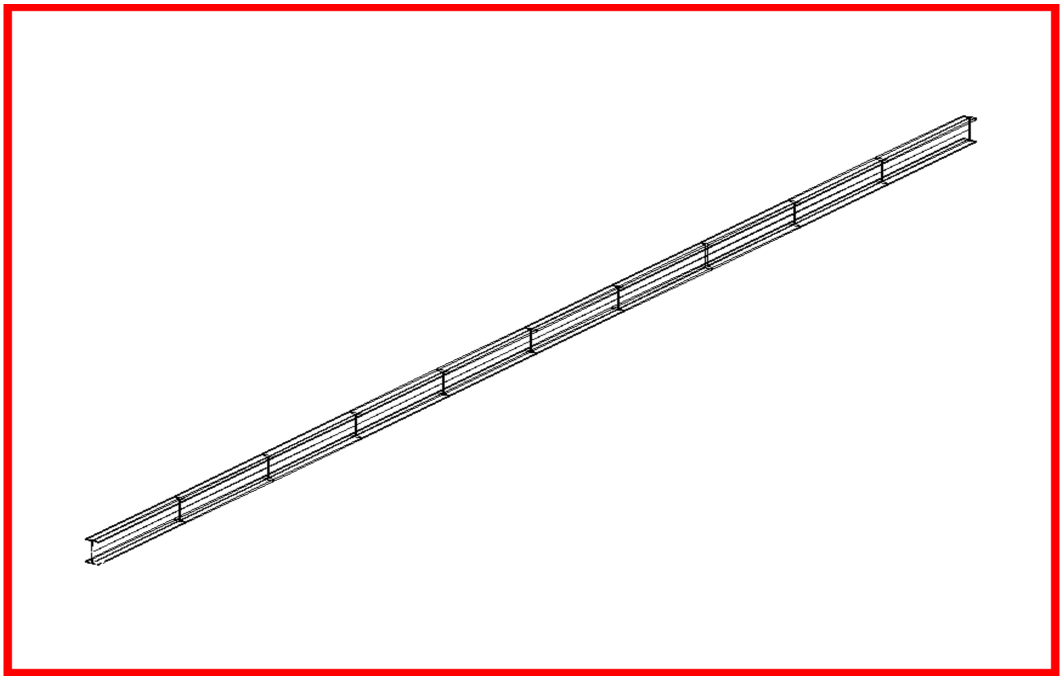
Reset Graphics

You can go back and select any *Results Case, Fringe Results or Deformation Results* you are interested in.

Quit MSC.Patran when you are finished with this exercise.

APPENDIX 1c

Normal Modes with Differential Stiffness, Using STATSUB



Objectives:

- Perform normal modes analysis of a simply supported beam with differential stiffness.
- Utilize the NASTRAN STATSUB feature.
- Find the first three natural frequencies and mode shapes of the beam.

Model Description:

The goal of this example is to determine normal modes with differential stiffness by using the NASTRAN STATSUB feature.

Figure A-1c.1 below is a finite element representation of the beam. One end is constrained in all translation and the other is free to move in the X. Both ends are held in the X-rotation.

Figure A-1c.1 - Grid Coordinates and Element Connectivities

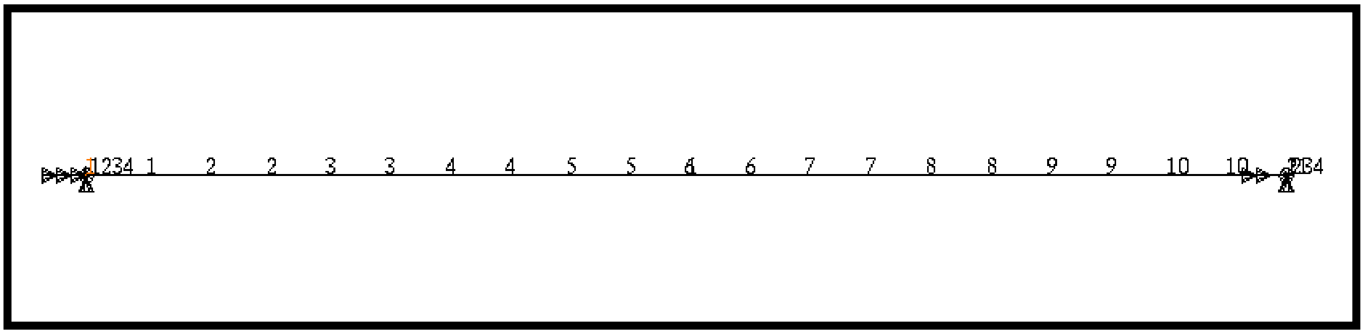


Table A-1c.1

Length	**1.0×10^3 mm**
Elastic Modulus	**2.0684×10^5 MPa**
Density	**7.8334×10^{-9}N-sec^2/mm^4**
Poisson's Ratio	**0.32**
Area	**5×10^3 mm^2**
I_1	**1.0417×10^6 mm^4**
I_2	**1.0 mm^4**
Force	**1×10^7 N**

Theoretical Solution

$$f_n = \frac{K_n}{2\pi}\left[\frac{EIg}{Wl^4}\left(1 + \frac{1}{Kr}\frac{Pl^2}{EI}\right)\right]^{1/2}$$

For Mode 1, Kr =9.87

$$fn = \frac{9.87}{2\pi}\left[\frac{(2.0684\times10^5)(1.0417\times10^6)}{(7.8334\times10^{-9})(5\times10^3)(1.0\times10^3)^4}\,\mathrm{x}\left(1 + \frac{1}{9.87}\frac{(1\times10^7)(1\times10^3)^2}{(2.0684\times10^5)(1.0417\times10^6)}\right)\right]^{1/2}$$

$$f_n = 278.22Hz$$

For Static Load

$$\Delta = \frac{PL}{AE}$$

$$\Delta = \frac{(1\times10^7)(1\times10^3)}{(5\times10^3)(2.0684\times10^5)}$$

$$\Delta = 9.67nm$$

Suggested Exercise Steps

- Explicitly generate a finite element representation of the beam structure (i.e., the grids (GRID) and element connectivities (CBEAM) should be defined manually).
- Define material (MAT1) and element (PBAR) properties.
- Apply the fixed boundary constraints (SPC1).
- Prepare the model for a normal modes analysis (SOL 103 and PARAMs).
  - PARAM WTMASS 0.00259
  - PARAM COUPMASS 1
  - EIGRL (To select Lanczos)
- Generate an input file and submit it to the MSC.Nastran solver for normal modes analysis.
- Review the results, specifically the eigenvalues.

```
ID SEMINAR,PROB1
CEND
BEGIN BULK
```

1	2	3	4	5	6	7	8	9	10

1	2	3	4	5	6	7	8	9	10

ENDDATA

Exercise Procedure:

1.	**Users who are not utilizing MSC.Patran for generating an input file should go to Step 10, otherwise, proceed to Step 2.**

2. Create a new database named **probap1.db**.

File/Open Database

New Database Name	**probap1**
OK	

3. Now, you will generate the initial input file with tensile load.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method	**Analysis Deck**
Job Name	**probap1c**
Solution Type...	
Solution Type:	◆ **Linear Static**
Solution Parameters...	
(Deselect ***Automatic Constraints****.)*	❐ **Automatic Constraints**
Mass Calculation:	**Coupled**
Data Deck Echo:	**None**
OK	
OK	
Direct Text Input...	
◆ **Case Control Section**	
Clear	
◆ **Bulk Data Section**	

Clear

OK

Apply

An MSC.Nastran input file called **probap1c.bdf** will be generated.

Generating an Input File for MSC.Nastran Users:

MSC.Nastran users can generate an input file using the data from Table A-1c.1. The result should be similar to the output below. This is only the starting point.

4. MSC.Nastran input file: **probap1c.bdf**

probap1c.bdf

```
SOL 101
CEND
ECHO = NONE
SUBCASE 1
   SPC = 2
   LOAD = 2
   DISPLACEMENT(SORT1,REAL)=ALL
   SPCFORCES(SORT1,REAL)=ALL
   STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
BEGIN BULK
PARAM    POST    -1
PARAM    PATVER  3.
PARAM    AUTOSPC NO
PBAR     1       1       5000.   1.041+6 1.
CBAR     1       1       1       2       0.      1.      0.
CBAR     2       1       2       3       0.      1.      0.
CBAR     3       1       3       4       0.      1.      0.
CBAR     4       1       4       5       0.      1.      0.
CBAR     5       1       5       6       0.      1.      0.
CBAR     6       1       6       7       0.      1.      0.
CBAR     7       1       7       8       0.      1.      0.
CBAR     8       1       8       9       0.      1.      0.
CBAR     9       1       9       10      0.      1.      0.
CBAR     10      1       10      11      0.      1.      0.
MAT1     1       206840.         .32     7.833-9
GRID     1               0.      0.      0.
GRID     2               100.000 0.      0.
GRID     3               200.000 0.      0.
GRID     4               300.000 0.      0.
GRID     5               400.000 0.      0.
GRID     6               500.    0.      0.
GRID     7               600.000 0.      0.
GRID     8               700.000 0.      0.
GRID     9               800.000 0.      0.
GRID     10              900.000 0.      0.
GRID     11              1000.   0.      0.
SPCADD   2       1       3       4
LOAD     2       1.      1.      1
SPC1     1       1234    1
SPC1     3       234     11
SPC1     4       345     1       THRU    11
FORCE    1       11      0       1.+7    1.      0.      0.
ENDDATA 052ec265
```

Modification of the Input File:

Before submitting the input file for analysis, modifications must be made to it in order to perform normal mode analysis with differential stiffness. Add in the appropriate commands within the **.bdf** file by doing the following:

5. Open the input file, **probap1c.bdf** with the use of a text editor or any means available.

   5a. Locate and **REPLACE** the text SOL 101 with the following:

   SOL 103

   5b. **ADD** a new subcase after the SUBCASE 1 section and before BEGIN BULK text line:

   ```
   SUBCASE 2
       STATSUB = 1
       METHOD = 1
       SPC = 2
       VECTOR = ALL
   ```

   5c. **ADD** the following after the BEGIN BULK text line:

   EIGRL, 1, , , 3

Save the changes to the **.bdf** file and close the text editor. You are now ready to proceed with the submitting the input file for analysis.

Submitting the Input File for Analysis:

6. Submit the input file to MSC.Nastran for analysis.

   6a. To submit the MSC.Patran **.bdf** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran probap1c.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

   6b. To submit the MSC.Nastran **.dat** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran probap1c.dat scr=yes**. Monitor the run using the UNIX **ps** command.

7. When the run is completed, edit the **probap1c.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing WARNING messages indicate modeling errors.

8. While still editing **probap1c.f06**, search for the word:

R E A L (spaces are necessary)

What are the first three modes?

1st = ___________Hz

2nd = ___________Hz

3rd = ___________Hz

Comparison of Results:

9. Compare the results obtained in the **.f06** file with the following results:

MODE NO.	EXTRACTION ORDER	EIGENVALUE	RADIANS	CYCLES	GENERALIZED MASS	GENERALI STIFFN
1	1	3.055547E+06	1.748012E+03	2.782048E+02	1.000000E+00	3.055547E
2	2	1.865041E+07	4.318612E+03	6.873285E+02	1.000000E+00	1.865041E
3	3	6.515460E+07	8.071840E+03	1.284673E+03	1.000000E+00	6.515460E

Quit MSC.Patran when you are finished with this exercise.

Mode 1	**278.20 Hz**
Mode 2	**687.33 Hz**
Mode 3	**12847 Hz**