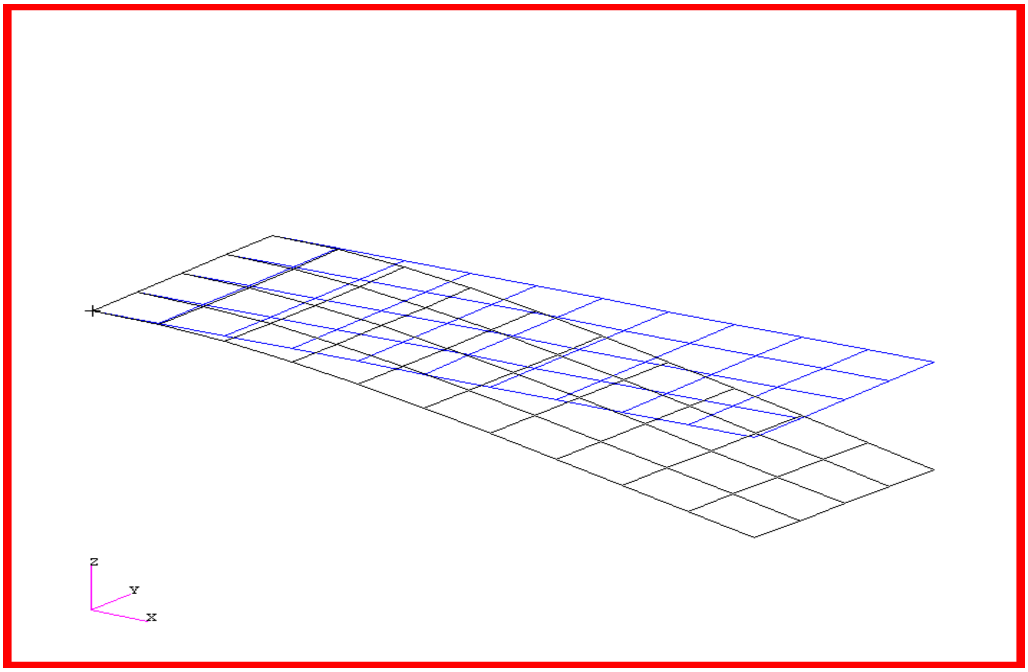
WORKSHOP PROBLEM 6

Modal Frequency Response Analysis



Objectives:

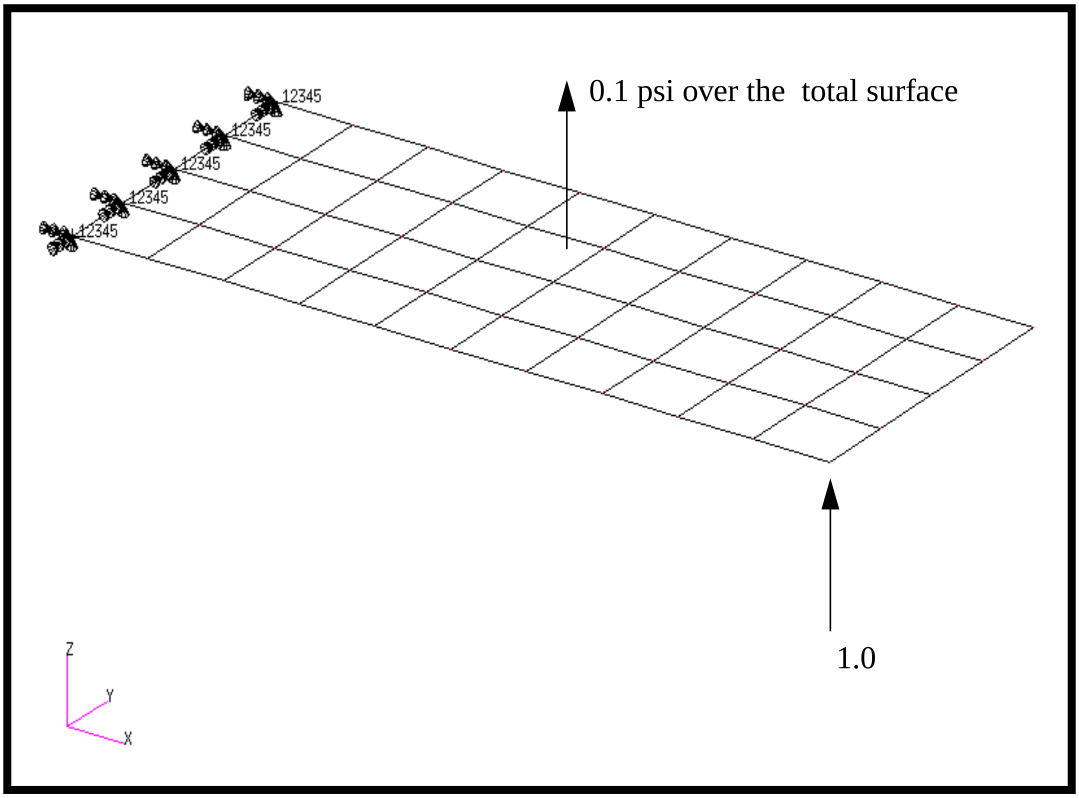
- Define a frequency-varying excitation.
- Produce a MSC/NASTRAN input file from a dynamic math model created in Workshop 1.
- Submit the file for analysis in MSC/NASTRAN.
- Compute nodal displacements for desired frequency domain.

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 direct transient 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 Database

New Database Name **prob6**

OK

In the *New Model Preference* 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...

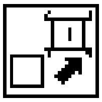
Select File **plate.bdf**

OK

Apply

OK

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



Show Labels

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

◆ **Load Cases**

Action:	**Create**
Load Case Name:	**frequency_dependent**
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 transient response.

◆ **Fields**

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

■ **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 Dependence *(Select from the **Time Dependent Fields** box.)*	**f:frequency_dependent_load**

OK

Select Application Region...

■ **FEM**

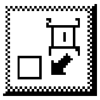
Select 2D Elements or Edge:	**Elem 1:40**

Add

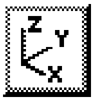
OK

Apply

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



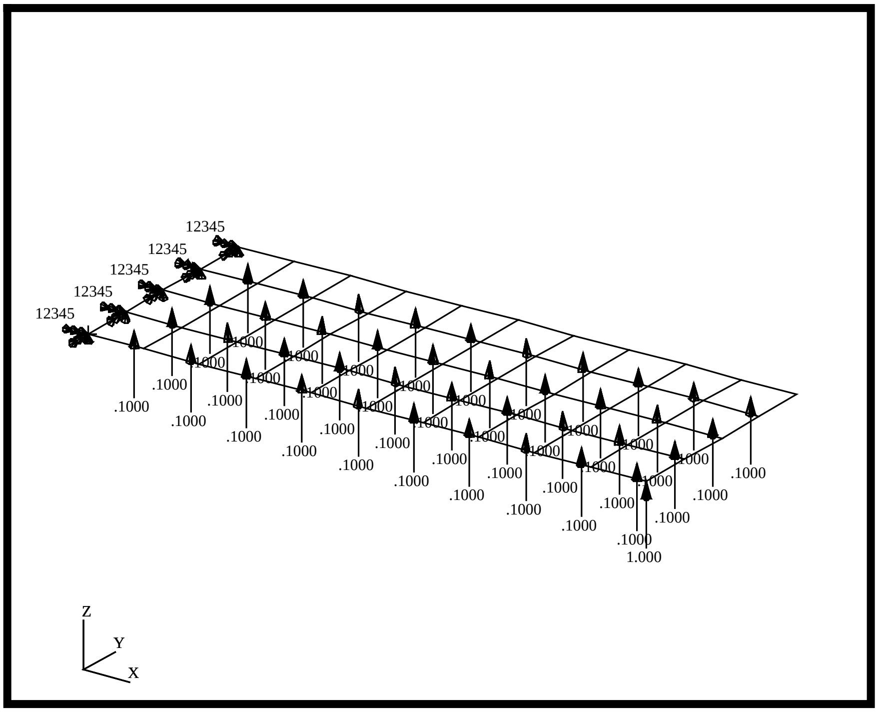
Hide Labels



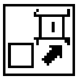
Iso 3 View

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**

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 Dependence: (*Select from the **Time Dependent Fields** box.*)	**f:frequency_dependent_load**
OK	

Select Application Region...

■ **FEM**

Select Nodes **Node 11**

Add

OK

Apply

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**

Solution Type...

Solution Type: ◆ **FREQUENCY RESPONSE**

Solution Parameters...

Formulation: **Modal**

Mass Calculation: **Coupled**

Wt.-Mass Conversion = **0.00259**

Eigenvalue Extraction...

Frequency Range of Interest:

Lower = **10.**

Upper = **2000.**

OK

OK

OK

Direct Text Input...

◆ **Case Control Section**	**SDAMPING = 100**
◆ **Bulk Data Section** *(Each line in the box is a separate line to input!)*	**TABDMP1, 100, CRIT,** **+, 0., .03, 10., .03, ENDT** **FREQ4, 2, 20., 1000., .03, 5**
OK	
Subcase Create...	
Available Subcases	**frequency_dependent**
Subcase Parameters...	
Starting Frequency =	**20**
Ending Frequency =	**1000**
# of Freq. Increments =	**49**
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	

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 to insert the appropriate phase for the point load.

Search for:

```
RLOAD1  5       6                               1
```

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

```
RLOAD1  5       6               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 results on the following page:

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 FOR YMAX
1	DISP	1	11(5,--)	2.000000E+01	1.000000E+03	3.481836E-04	4.400000E+02	1.699121E-01	1.336996E+02
				2.000000E+01	1.000000E+03	3.481836E-04	4.400000E+02	1.699121E-01	1.336996E+02
1	DISP	1	11(--, 11)	2.000000E+01	1.000000E+03	1.390213E+02	1.000000E+03	3.258276E+02	2.000000E+01
				2.000000E+01	1.000000E+03	1.390213E+02	1.000000E+03	3.258276E+02	2.000000E+01
1	DISP	2	33(5,--)	2.000000E+01	1.000000E+03	2.271459E-04	6.600000E+02	1.700317E-01	1.336996E+02
				2.000000E+01	1.000000E+03	2.271459E-04	6.600000E+02	1.700317E-01	1.336996E+02
1	DISP	2	33(--, 11)	2.000000E+01	1.000000E+03	1.385571E+02	1.000000E+03	3.263339E+02	2.000000E+01
				2.000000E+01	1.000000E+03	1.385571E+02	1.000000E+03	3.263339E+02	2.000000E+01
1	DISP	3	55(5,--)	2.000000E+01	1.000000E+03	1.278678E-04	1.000000E+03	1.696787E-01	1.336996E+02
				2.000000E+01	1.000000E+03	1.278678E-04	1.000000E+03	1.696787E-01	1.336996E+02
1	DISP	3	55(--, 11)	2.000000E+01	1.000000E+03	1.687413E+01	7.001384E+02	3.573561E+02	7.104853E+02
				2.000000E+01	1.000000E+03	1.687413E+01	7.001384E+02	3.573561E+02	7.104853E+02

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 importing the **prob6.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 File	**prob6.op2**
OK	
Apply	

19. Plot the results in XY Plots.

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

◆ **Results**

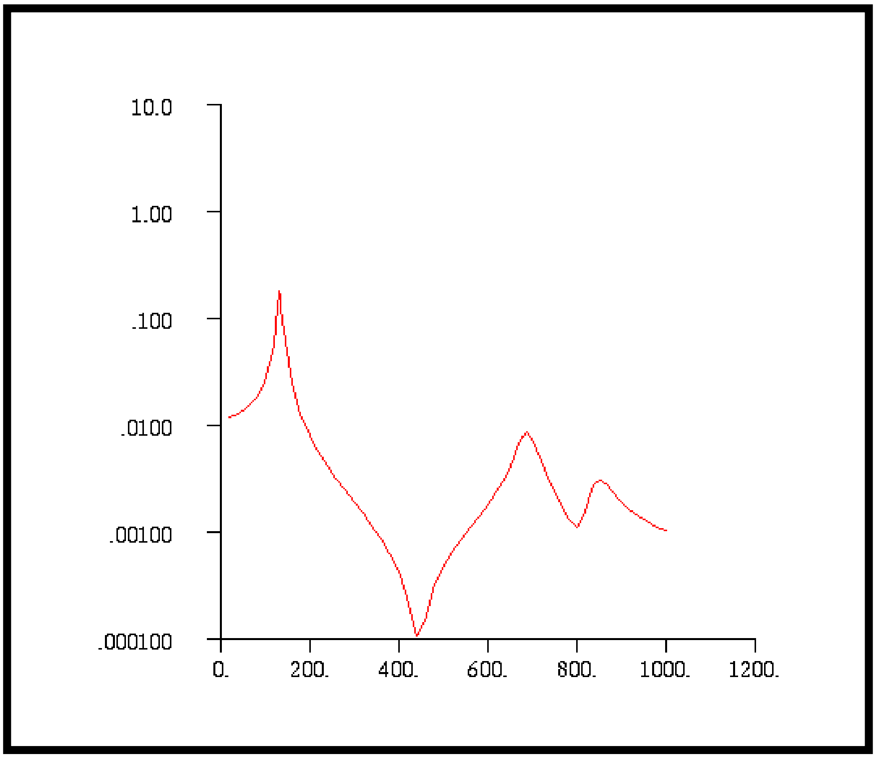
Form Type:	**Advanced**
Select Result Cases *(Highlight all cases.)*	
Get Results	
Select Result	**1.1-Displacements, Translational**
Plot Type:	**XY Plot**
Plot Type Options...	
Result XY Plot Types	**Results Versus Global Variables**
Global Var...	
Global Variable:	**1-Frequency**
Apply	
Result (Y)...	
Results:	**1.1-Displacements, Translational**

Vector Component	❐ **X** ❐ **Y** ■ **Z**
Numerical Form for Complex Results	■ **Mag.**
OK	
Node IDs	**Node 11**
Apply...	
New Title or Title Filter	**Displacement Response at Loaded Corner**
Rename	
Apply	

◆ **XY Plot**

Action:	**Modify**
Object:	**Axis**
Active Axis:	❐ **X** ■ **Y**
Scale...	
Scale:	■ **Logarithmic**
Apply	
Cancel	

Figure 6.3-Displacement Response at Loaded Corner



The second plot is the Displacement versus Frequency at Node 33. Repeat the above steps of plotting the XY plots of Node 11 for Nodes 33 and 55. Push **Cancel** to remove any miscellaneous forms until the ***Results Display*** form.

◆ **Results**

Plot Type Options...

Result (Y)...

Numerical Form for Complex Results ■ **Mag.**

OK

Node IDs **Node 33**

Apply...

Result XY Window Name: **XYWindow2**

New Title or Title Filter: **Displacement Response at Tip Center**

Rename

Apply

◆ **XY Plot**

Action:	**Modify**
Object:	**Axis**
Active Axis:	❐ **X** ■ **Y**

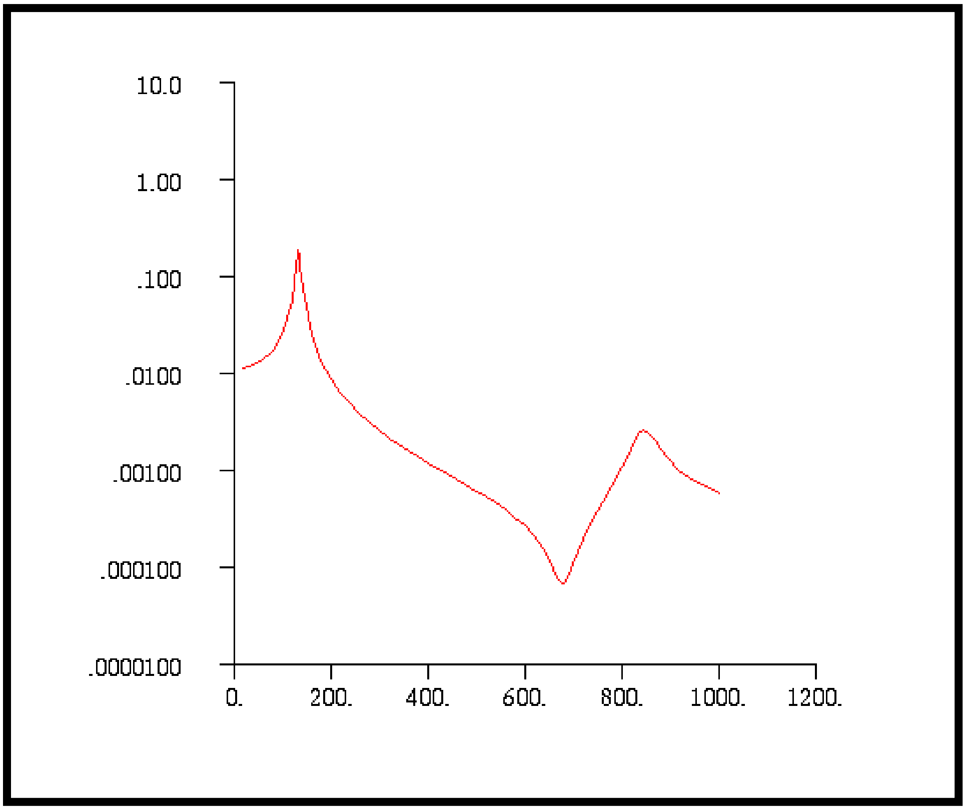
Scale...

Scale:	■ **Logarithmic**

Apply

Cancel

Figure 6.4-Displacement Response at Tip Center



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

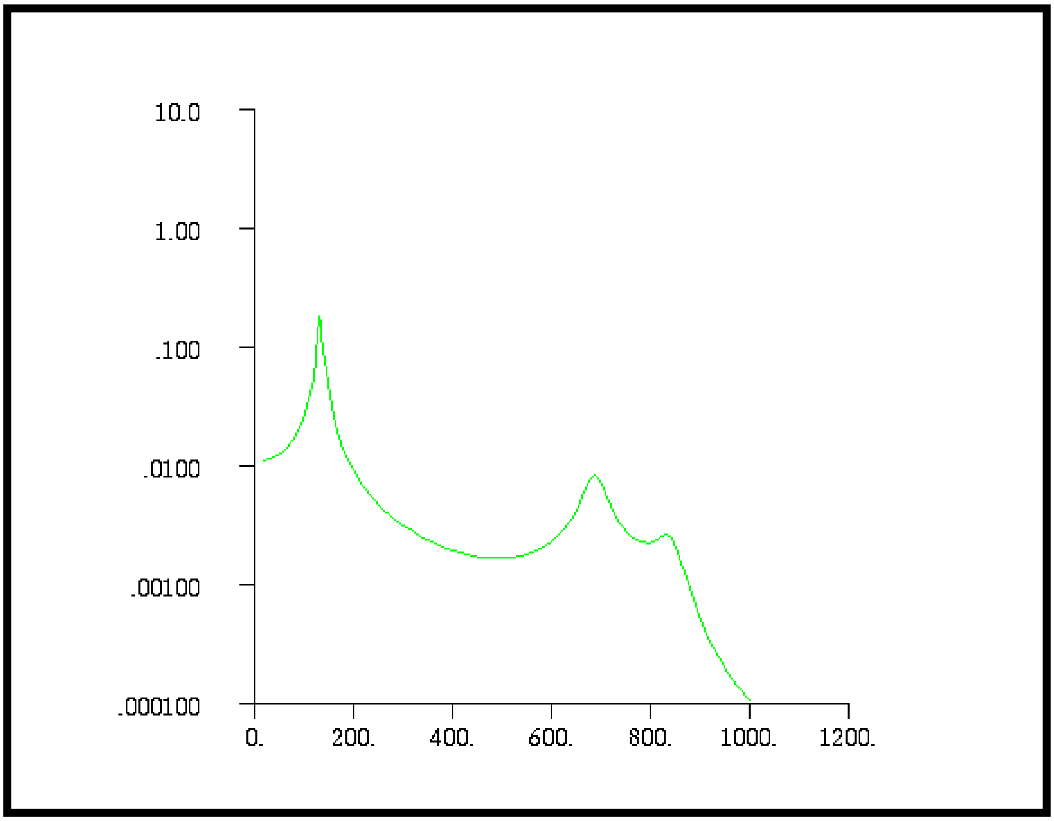
◆ **Results**

Plot Type Options...	
Result (Y)...	
Numerical Form for Complex Results	■ **Mag.**
OK	
Node IDs	**Node 55**
Apply...	
Result XY Window Name:	**XYWindow3**
New Title or Title Filter	**Displacement Response at Opposite Corner**
Rename	
Apply	

◆ **XY Plot**

Action:	**Modify**
Object:	**Axis**
Active Axis:	❐ **X** ■ **Y**
Scale...	
Scale:	■ **Logarithmic**
Apply	
Cancel	

Figure 6.5-Displacement Response at Opposite Corner



Quit MSC/PATRAN when you have completed this exercise.