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.
### Direct Frequency Response Analysis

<table>
<thead>
<tr>
<th></th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**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):
  
  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**
  
  [Options ... ]
  
  **Frequency (f)**
  
  **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.

<table>
<thead>
<tr>
<th>Freq (f)</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
</tr>
</tbody>
</table>
7. Create the frequency dependent unit force.

◆ **Load/BCs**

<table>
<thead>
<tr>
<th>Action:</th>
<th>Create</th>
</tr>
</thead>
<tbody>
<tr>
<td>Object:</td>
<td>Force</td>
</tr>
<tr>
<td>Type:</td>
<td>Nodal</td>
</tr>
<tr>
<td>New Set Name:</td>
<td>unit_force</td>
</tr>
</tbody>
</table>

**Input Data...**

<table>
<thead>
<tr>
<th>Spatial Dependence/Force:</th>
<th>&lt;0 0 1&gt;</th>
</tr>
</thead>
<tbody>
<tr>
<td>* Time/Freq. Dependence:</td>
<td>f:frequency_dependent_load</td>
</tr>
</tbody>
</table>

(Select from the Time Dependent Fields box)

**OK**

**Select Application Region...**

**FEM**

<table>
<thead>
<tr>
<th>Select Nodes:</th>
<th>Node 11</th>
</tr>
</thead>
</table>

**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

<table>
<thead>
<tr>
<th>Action:</th>
<th>Analyze</th>
</tr>
</thead>
<tbody>
<tr>
<td>Object:</td>
<td>Entire Model</td>
</tr>
<tr>
<td>Method:</td>
<td>Analysis Deck</td>
</tr>
</tbody>
</table>

MSC.Nastran 102 Exercise Workbook 5-11
Job Name

**Translation Parameters...**

*Data Output:*

**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**
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
YTITLE= YES
YBLOG= NO

XTITLE= FREQUENCY (HZ)

YTITLE= DISPLACEMENT RESPONSE AT LOADED CORNER, MAGNITUDE

YBTITLE= DISPLACEMENT RESPONSE AT LOADED CORNER, PHASE

XYPLOT DISP RESPONSE / 11 (T3RM, T3IP)

YTITLE= DISPLACEMENT RESPONSE AT TIP CENTER, MAGNITUDE

YBTITLE= DISPLACEMENT RESPONSE AT TIP CENTER, PHASE

XYPLOT DISP RESPONSE / 33 (T3RM, T3IP)

YTITLE= 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 =  __________
### Comparison of Results

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

For POINT-ID = 11:

<table>
<thead>
<tr>
<th>FREQUENCY</th>
<th>TYPE</th>
<th>T1</th>
<th>T2</th>
<th>T3</th>
<th>R1</th>
<th>R2</th>
<th>R3</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.000000E+01</td>
<td>G</td>
<td>0.0</td>
<td>0.0</td>
<td>8.817999E-03</td>
<td>6.435859E-04</td>
<td>2.632016E-03</td>
<td>0.0</td>
</tr>
<tr>
<td>4.000000E+01</td>
<td>G</td>
<td>0.0</td>
<td>0.0</td>
<td>356.4954</td>
<td>176.5664</td>
<td>176.5000</td>
<td>0.0</td>
</tr>
<tr>
<td>9.799999E+02</td>
<td>G</td>
<td>0.0</td>
<td>0.0</td>
<td>356.2596</td>
<td>176.5677</td>
<td>176.2785</td>
<td>0.0</td>
</tr>
<tr>
<td>1.000000E+03</td>
<td>G</td>
<td>0.0</td>
<td>0.0</td>
<td>187.6832</td>
<td>7.8008</td>
<td>15.1581</td>
<td>0.0</td>
</tr>
</tbody>
</table>

For POINT-ID = 33:

<table>
<thead>
<tr>
<th>FREQUENCY</th>
<th>TYPE</th>
<th>T1</th>
<th>T2</th>
<th>T3</th>
<th>R1</th>
<th>R2</th>
<th>R3</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.000000E+01</td>
<td>G</td>
<td>0.0</td>
<td>0.0</td>
<td>8.183126E-03</td>
<td>5.993295E-04</td>
<td>2.443290E-03</td>
<td>0.0</td>
</tr>
<tr>
<td>4.000000E+01</td>
<td>G</td>
<td>0.0</td>
<td>0.0</td>
<td>356.4899</td>
<td>176.5639</td>
<td>176.4950</td>
<td>0.0</td>
</tr>
<tr>
<td>9.799999E+02</td>
<td>G</td>
<td>0.0</td>
<td>0.0</td>
<td>356.2376</td>
<td>176.5565</td>
<td>176.2581</td>
<td>0.0</td>
</tr>
<tr>
<td>1.000000E+03</td>
<td>G</td>
<td>0.0</td>
<td>0.0</td>
<td>188.0180</td>
<td>5.5597</td>
<td>10.0794</td>
<td>0.0</td>
</tr>
</tbody>
</table>

For POINT-ID = 55:

<table>
<thead>
<tr>
<th>FREQUENCY</th>
<th>TYPE</th>
<th>T1</th>
<th>T2</th>
<th>T3</th>
<th>R1</th>
<th>R2</th>
<th>R3</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.000000E+01</td>
<td>G</td>
<td>0.0</td>
<td>0.0</td>
<td>7.606255E-03</td>
<td>5.587703E-04</td>
<td>2.371172E-03</td>
<td>0.0</td>
</tr>
<tr>
<td>4.000000E+01</td>
<td>G</td>
<td>0.0</td>
<td>0.0</td>
<td>356.4844</td>
<td>176.5612</td>
<td>176.4928</td>
<td>0.0</td>
</tr>
<tr>
<td>9.799999E+02</td>
<td>G</td>
<td>0.0</td>
<td>0.0</td>
<td>356.2155</td>
<td>176.5442</td>
<td>176.2492</td>
<td>0.0</td>
</tr>
<tr>
<td>1.000000E+03</td>
<td>G</td>
<td>0.0</td>
<td>0.0</td>
<td>193.1958</td>
<td>4.6290</td>
<td>9.0143</td>
<td>0.0</td>
</tr>
</tbody>
</table>
15. MSC.Nastran MSC.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

<table>
<thead>
<tr>
<th>Action:</th>
<th>Attach XDB</th>
</tr>
</thead>
<tbody>
<tr>
<td>Object:</td>
<td>Result Entities</td>
</tr>
<tr>
<td>Method:</td>
<td>Local</td>
</tr>
</tbody>
</table>

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

<table>
<thead>
<tr>
<th>Action:</th>
<th>Create</th>
</tr>
</thead>
<tbody>
<tr>
<td>Object:</td>
<td>Graph</td>
</tr>
<tr>
<td>Method:</td>
<td>y vs x</td>
</tr>
</tbody>
</table>

Select Results Cases

- Frequency_response, 0 of 50 subcases

Filter Method

<table>
<thead>
<tr>
<th>Filter</th>
</tr>
</thead>
</table>

Apply

Close

Select Y result: Displacement, Translational

Quantity: Z Component
The next step is to make the plot of Phase versus Frequency.

**Plot Options**

*Complex No as:*  
Phase:  
Apply
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.