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.
BEGIN BULK
<table>
<thead>
<tr>
<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>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<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 13, otherwise, proceed to step 2.

2. Open a new database named prob3.db.

   File/New Database
   
   New Database Name
   prob3
   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 toolbar.

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

◆ Load Cases

Action: 
Load Case Name: 
Load Case Type: 
Assign/Prioritize Loads/BCs

Select Load/BCs to Add to Spreadsheet
(Select from menu.)

OK

Apply

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

◆ Fields

Action:
Object: 
Method: 
Field Name: 
[Options ...]

Maximum Number of \( \tau \)

OK

Input Data ...

Map Function to Table...

\( PCL \) Expression \( f'(t) \):

\( \text{sind}(90000.^*t) \)

Start Time

End Time

Number of Points

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

<table>
<thead>
<tr>
<th>Time(t)</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>21</td>
<td>0.04</td>
</tr>
</tbody>
</table>

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`
  - **Maximum Number of t:** 21
    - **OK**
  - **Input Data ...**
  - **Map Function to Table...**
    - **PCL Expression f'(t):** `-sind(90000.*'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:

<table>
<thead>
<tr>
<th>Time(t)</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>21</td>
<td>0.04</td>
</tr>
<tr>
<td>OK</td>
<td></td>
</tr>
<tr>
<td>Apply</td>
<td></td>
</tr>
</tbody>
</table>

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 Dependence**
(Select from the *Time Dependent Fields* box)
- **f**:time_dependent_pressure

**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:**

**Object:**

**Type:**

*New Set Name*

**Input Data...**

**Spatial Dependence**

**Force** <F1 F2 F3>

**Time Dependence:**

(Select from the *Time Dependent Fields* box)

```
<0 0 50>

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]

The result should be similar to **Figure 3.2**.
10. Create the analysis.

◆ **Analysis**

*Action:*

*Object:*

*Method:*

*Job Name*

**Solution Type...**

*Solution Type:*

◆ **TRANSIENT RESPONSE**

**Solution Parameters...**

*Formulation:*

*Mass Calculation:*

*Wt.-Mass Conversion =*

*Struct. Damping Coeff. =*

*W3, Damping Factor =*

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

**SPCFORCES(SORT1,Real)=All FEM**

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

**Transient_response**

**Subcase Parameters...**

*Ending Time* = 0.04

*Number of Time Steps* = 100

**OK**

**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*: Default

*Subcases for Solution*

*Sequence*: 109

(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:

   **DISPL** (spaces are necessary)

   Displacement at Grid 11

<table>
<thead>
<tr>
<th>Time</th>
<th>T3</th>
</tr>
</thead>
</table>
   | .0024| _______
   | .0052| _______
   | .02  | _______

   Displacement at Grid 33

<table>
<thead>
<tr>
<th>Time</th>
<th>T3</th>
</tr>
</thead>
</table>
   | .0024| _______
   | .0052| _______
   | .02  | _______
Displacement at Grid 55

<table>
<thead>
<tr>
<th>Time</th>
<th>T3</th>
</tr>
</thead>
<tbody>
<tr>
<td>.0024</td>
<td></td>
</tr>
<tr>
<td>.0052</td>
<td></td>
</tr>
<tr>
<td>.02</td>
<td></td>
</tr>
</tbody>
</table>
Comparison of Results

16. Compare the results obtained in the .f06 file with the results on the following page:
### POINT-ID = 11

**DISPLACEMENT VECTOR**

<table>
<thead>
<tr>
<th>TIME</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>0.0</td>
<td>G</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>4.000000E-04</td>
<td>G</td>
<td>0.0</td>
<td>0.0</td>
<td>-2.173625E-02</td>
<td>1.104167E-02</td>
<td>1.050818E-02</td>
<td>0.0</td>
</tr>
<tr>
<td>8.000000E-04</td>
<td>G</td>
<td>0.0</td>
<td>0.0</td>
<td>-7.204904E-02</td>
<td>2.847414E-02</td>
<td>2.852519E-02</td>
<td>0.0</td>
</tr>
<tr>
<td>1.200000E-03</td>
<td>G</td>
<td>0.0</td>
<td>0.0</td>
<td>-1.433462E-01</td>
<td>4.082027E-02</td>
<td>4.915178E-02</td>
<td>0.0</td>
</tr>
</tbody>
</table>

### POINT-ID = 33

**DISPLACEMENT VECTOR**

<table>
<thead>
<tr>
<th>TIME</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>0.0</td>
<td>G</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>4.000000E-04</td>
<td>G</td>
<td>0.0</td>
<td>0.0</td>
<td>-1.122398E-02</td>
<td>9.220218E-03</td>
<td>6.138594E-03</td>
<td>0.0</td>
</tr>
<tr>
<td>8.000000E-04</td>
<td>G</td>
<td>0.0</td>
<td>0.0</td>
<td>-4.424753E-02</td>
<td>2.576699E-02</td>
<td>2.014980E-02</td>
<td>0.0</td>
</tr>
<tr>
<td>1.200000E-03</td>
<td>G</td>
<td>0.0</td>
<td>0.0</td>
<td>-1.030773E-01</td>
<td>3.819036E-02</td>
<td>3.922388E-02</td>
<td>0.0</td>
</tr>
</tbody>
</table>

### POINT-ID = 55

**DISPLACEMENT VECTOR**

<table>
<thead>
<tr>
<th>TIME</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>0.0</td>
<td>G</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>4.000000E-04</td>
<td>G</td>
<td>0.0</td>
<td>0.0</td>
<td>-2.849185E-03</td>
<td>7.791447E-03</td>
<td>4.611430E-03</td>
<td>0.0</td>
</tr>
<tr>
<td>8.000000E-04</td>
<td>G</td>
<td>0.0</td>
<td>0.0</td>
<td>-1.992890E-02</td>
<td>2.322436E-02</td>
<td>1.681028E-02</td>
<td>0.0</td>
</tr>
<tr>
<td>1.200000E-03</td>
<td>G</td>
<td>0.0</td>
<td>0.0</td>
<td>-6.643156E-02</td>
<td>3.540079E-02</td>
<td>3.501805E-02</td>
<td>0.0</td>
</tr>
</tbody>
</table>

### POINT-ID = 33

**DISPLACEMENT VECTOR**

<table>
<thead>
<tr>
<th>TIME</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>0.0</td>
<td>G</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>4.000000E-04</td>
<td>G</td>
<td>0.0</td>
<td>0.0</td>
<td>-2.115454E-02</td>
<td>8.268487E-05</td>
<td>5.912832E-03</td>
<td>0.0</td>
</tr>
<tr>
<td>8.000000E-04</td>
<td>G</td>
<td>0.0</td>
<td>0.0</td>
<td>-2.998628E-03</td>
<td>6.654292E-06</td>
<td>8.371378E-04</td>
<td>0.0</td>
</tr>
<tr>
<td>1.200000E-03</td>
<td>G</td>
<td>0.0</td>
<td>0.0</td>
<td>-1.529953E-02</td>
<td>-6.482315E-05</td>
<td>-4.277684E-03</td>
<td>0.0</td>
</tr>
</tbody>
</table>
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 `prob3.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 prob3.op2

OK

Apply

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

◆ **Results**

*Form Type:* Advanced

*Select Results Cases* (Highlight all.)

Get Results

Select Result 1.1 Displacements, Translational

Plot Type XY Plot

Plot Type Options...

Global Var...

Global Variables 1-Time

Apply

Result(Y)... 1.1-Displacements, Translational

Vector Component □ X □ Y ■ Z

OK

Node IDs Node 11
You may reset the graphics by clicking on this icon:

![Reset Graphics](image)

**Figure 3.3**-Displacement Response at Node 11
Repeat the above steps for plotting the xy plots of Node 11, Node 33 and Node 55. Return to the Results Display form. If the Curves for XY Plot form and the Result XY Plot Options form are still open, close them by pushing the Cancel button.

**Plot Type Options...**

**Global Var...**

*Global Variables*

Apply

Result(Y)...

*Results*

1.1-Displacements, Translational

*Vector Component*

X   Y   Z

OK

Node IDs

Apply

New Title or Title Filter

Displacement Response at Tip Center

Rename

Apply

**Figure 3.4-Displacement Response at Node 33**
Return to the *Results Display* form. If the *Curves for XY Plot* form and the *Result XY Plot Options* form are still open, close them by pushing the *Cancel* button.

**Plot Type Option...**

**Global Var...**

*Global Variables*

1. **Time**

**Apply**

**Result(Y)...**

*Results*

1.1-**Displacements, Translational**

**Vector Component**

☐ X  ☐ Y  ■ Z

**OK**

**Node IDs**

**Apply**

**New Title or Title Filter**

*Displacement Response at Opposite Corner*

**Rename**

**Apply**

**Figure 3.5-Displacement Response at Node 55**

Quit MSC/PATRAN when you are finished with this exercise.