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**

![Diagram of model with masses and springs](image)

**Table 12.1**

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>$m_1$</td>
<td>3.0 lb-sec$^2$/in</td>
</tr>
<tr>
<td>$m_2$</td>
<td>1.5 lb-sec$^2$/in</td>
</tr>
<tr>
<td>$K_1$</td>
<td>50,000 lb/in</td>
</tr>
<tr>
<td>$K_2$</td>
<td>12,500 lb/in</td>
</tr>
<tr>
<td>$C_2$</td>
<td>30 lb-sec/in</td>
</tr>
</tbody>
</table>
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.
WORKSHOP 12  Complex Modes of a Pile Driver

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>
</tbody>
</table>

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.

<table>
<thead>
<tr>
<th>Node</th>
<th>Location</th>
</tr>
</thead>
<tbody>
<tr>
<td>Node 2</td>
<td>[1 0 0]</td>
</tr>
<tr>
<td>Node 3</td>
<td>[2 0 0]</td>
</tr>
</tbody>
</table>

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
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


**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.)
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

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
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.)

- **Point element**

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

- **Properties**
  - **Action:** Create
  - **Dimension:** 0D
  - **Type:** Mass
  - **Property Set Name:** mass2
  - **Option:** Lumped
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, , >

Select Application Region...

◆ FEM

Select Nodes: Node 3

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
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:

<table>
<thead>
<tr>
<th>ROOT NO.</th>
<th>ORDER</th>
<th>COMPLEX EIGENVALUE (REAL)</th>
<th>IMAG</th>
<th>FREQUENCY (CYCLES)</th>
<th>DAM</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.067907E-01</td>
<td>1</td>
<td>-2.660969E+00</td>
<td>-4.983521E+01</td>
<td>7.931520E+00</td>
<td></td>
</tr>
<tr>
<td>6.218695E-02</td>
<td>2</td>
<td>-7.339031E+00</td>
<td>-2.360312E+02</td>
<td>3.756553E+01</td>
<td></td>
</tr>
<tr>
<td>1.067907E-01</td>
<td>3</td>
<td>-2.660969E+00</td>
<td>4.983521E+01</td>
<td>7.931520E+00</td>
<td></td>
</tr>
<tr>
<td>6.218695E-02</td>
<td>4</td>
<td>-7.339031E+00</td>
<td>2.360312E+02</td>
<td>3.756553E+01</td>
<td></td>
</tr>
</tbody>
</table>

**SUBCASE 1**

COMPLEX EIGENVALUE = -2.660969E+00, -4.983521E+01

<table>
<thead>
<tr>
<th>POINT ID.</th>
<th>TYPE</th>
<th>T1 (REAL)</th>
<th>T2 (REAL)</th>
<th>T3 (REAL)</th>
<th>R1 (REAL)</th>
<th>R2 (REAL)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>G</td>
<td>1.000000E+00</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>G</td>
<td>8.514119E-01</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>3</td>
<td>G</td>
<td>1.591320E-02</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

**SUBCASE 2**

COMPLEX EIGENVALUE = -7.339031E+00, -2.360312E+02

<table>
<thead>
<tr>
<th>POINT ID.</th>
<th>TYPE</th>
<th>T1 (REAL)</th>
<th>T2 (REAL)</th>
<th>T3 (REAL)</th>
<th>R1 (REAL)</th>
<th>R2 (REAL)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>G</td>
<td>-4.241094E-01</td>
<td>-3.768431E-02</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>G</td>
<td>1.000000E+00</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>3</td>
<td>G</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

**SUBCASE 3**

COMPLEX EIGENVALUE = -2.660969E+00, 4.983521E+01

<table>
<thead>
<tr>
<th>POINT ID.</th>
<th>TYPE</th>
<th>T1 (REAL)</th>
<th>T2 (REAL)</th>
<th>T3 (REAL)</th>
<th>R1 (REAL)</th>
<th>R2 (REAL)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>G</td>
<td>1.000000E+00</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>G</td>
<td>8.514119E-01</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>3</td>
<td>G</td>
<td>1.591320E-02</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

**SUBCASE 4**

COMPLEX EIGENVALUE = -7.339031E+00, 2.360312E+02

<table>
<thead>
<tr>
<th>POINT ID.</th>
<th>TYPE</th>
<th>T1 (REAL)</th>
<th>T2 (REAL)</th>
<th>T3 (REAL)</th>
<th>R1 (REAL)</th>
<th>R2 (REAL)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>G</td>
<td>-4.241094E-01</td>
<td>3.768431E-02</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>G</td>
<td>1.000000E+00</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>3</td>
<td>G</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>
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