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

<table>
<thead>
<tr>
<th>Quantity</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length</td>
<td>100 in</td>
</tr>
<tr>
<td>Height</td>
<td>2 in</td>
</tr>
<tr>
<td>Width</td>
<td>1 in</td>
</tr>
<tr>
<td>Thickness</td>
<td>0.100 in</td>
</tr>
<tr>
<td>Area</td>
<td>0.38 in²</td>
</tr>
<tr>
<td>$I_1$</td>
<td>0.229 in⁴</td>
</tr>
<tr>
<td>$I_2$</td>
<td>0.017 in⁴</td>
</tr>
</tbody>
</table>
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, \( Kr = 9.87 \)

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

\[ f_n = 26.36 \text{Hz} \]

For Static Load

\[ \Delta = \frac{PL}{AE} \]

\[ \Delta = \frac{500(100)}{0.38(10 \times 10^6)} \]

\[ \Delta = 0.0132 \]
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.
Normal Modes with Differential Stiffness

ID SEMINAR, PROB1

CEND

BEGIN BULK
<p>| | | | | | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></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 6, otherwise, proceed to step 2.

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

   **File/Open Database**

   *Existing Database Name*
   
   ![prob14a]

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

   ![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 >
Mass Calculation: Coupled
Data Deck Echo: None
Wt. - Mass Conversion = .00259

Direct Text Input...
◆ Case Control Section
◆ Bulk Data Section

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 1.47 .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 39.999 0. 0. 345
GRID 6 49.999 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
```
<table>
<thead>
<tr>
<th>Command</th>
<th>ID</th>
<th>Value 1</th>
<th>Value 2</th>
<th>Value 3</th>
<th>Value 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>GRID</td>
<td>11</td>
<td>100.</td>
<td>0.</td>
<td>0.</td>
<td>345</td>
</tr>
<tr>
<td>LOAD</td>
<td>2</td>
<td>1.</td>
<td>1.</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>SPC1</td>
<td>1</td>
<td>1234</td>
<td>1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SPC1</td>
<td>1</td>
<td>234</td>
<td>11</td>
<td></td>
<td></td>
</tr>
<tr>
<td>FORCE</td>
<td>1</td>
<td>11</td>
<td>0</td>
<td>500.</td>
<td>1.</td>
</tr>
<tr>
<td>ENDDATA</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
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 results on the following page:
<table>
<thead>
<tr>
<th>MODE NO.</th>
<th>EXTRACTION ORDER</th>
<th>EIGENVALUE</th>
<th>RADIANS</th>
<th>CYCLES</th>
<th>GENERALIZED MASS</th>
<th>GENERALIZED STIFFNESS</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>2.735837E+04</td>
<td>1.654037E+02</td>
<td>2.632481E+01</td>
<td>1.000000E+00</td>
<td>2.735837E+04</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>3.748482E+05</td>
<td>6.122484E+02</td>
<td>9.744236E+01</td>
<td>1.000000E+00</td>
<td>3.748482E+05</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>1.816509E+06</td>
<td>1.347779E+03</td>
<td>2.145057E+02</td>
<td>1.000000E+00</td>
<td>1.816509E+06</td>
</tr>
</tbody>
</table>
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 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

Form Type: Basic

Select Results Cases
1.1-Default, Mode 1:FREQ=23.36

Select Deformation Result
1.1 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.