Objectives

- Analyze a stiffened beam for normal modes.
- Produce an MSC/NASTRAN input file that represents the 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>Property</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

\[ f_n = \frac{K_n}{2\pi} \left[ \frac{EIg}{Wl^4} \left( 1 + \frac{1}{Kr \frac{P}{l^2}} \right) \right]^{1/2} \]

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.
<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>
</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
   OK

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
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               +  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.+7            .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               39.9999 0.      0.              345
GRID     6               49.9999 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
```
Normal Modes with Differential Stiffness

GRID     11              100.    0.      0.              345
LOAD     2       1.      1.      1
SPC1     1       1234    1
SPC1     1       234     11
FORCE    1       11      0       500.    1.      0.      0.
ENDDATA  

MSC.Nastran 102 Exercise Workbook  14b-13
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 following results:

<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>GEN STI</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>2.735837</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>3.748482</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>1.816509</td>
</tr>
</tbody>
</table>
11. MSC.Nastran Users should 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

Action: Create
Object: Deformation
Select Results Cases: Default, Mode 1:Freq=26.325
Select Deformation Result: 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.