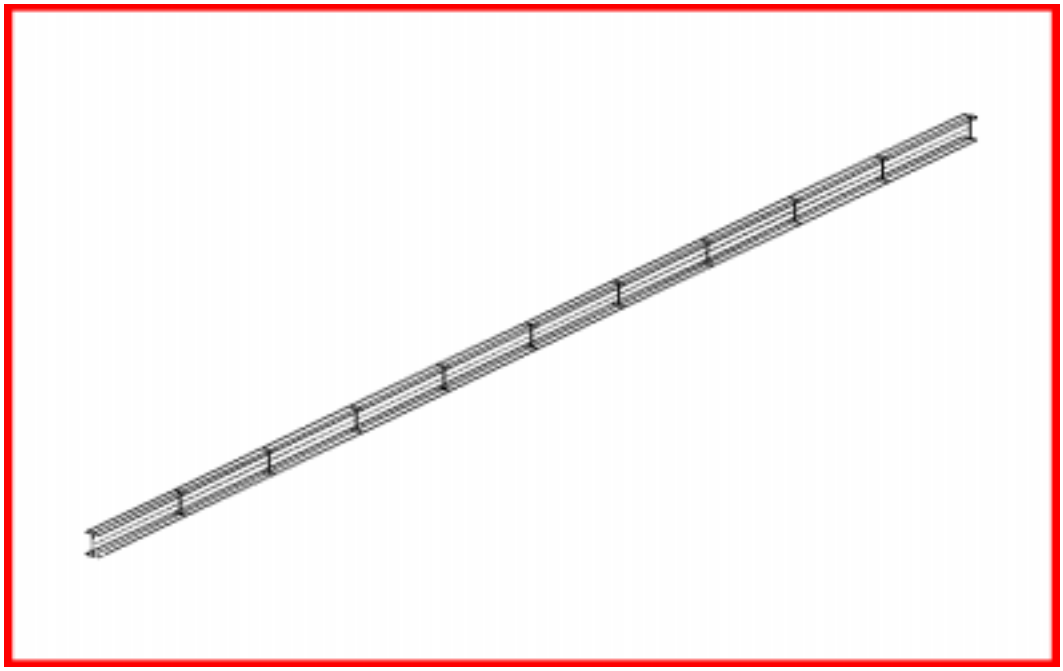
APPENDIX 1a

Modal Analysis of a Beam (SI Units)



Objectives

- Perform normal modes analysis of a cantilever beam.
- Submit the file for analysis in MSC/NASTRAN.
- Find the first three natural frequencies and mode shapes of the beam.

Model Description:

The goal of this example is to find the first 3 modes of a beam pinned at both ends.

Figure A-1a.1 below is a finite element representation of the beam. One end is contrained in all translations and the other is free to move in the X. Both ends are held in the X-rotation.

Figure A-1a.1-Grid Coordinates and Element Connectivities

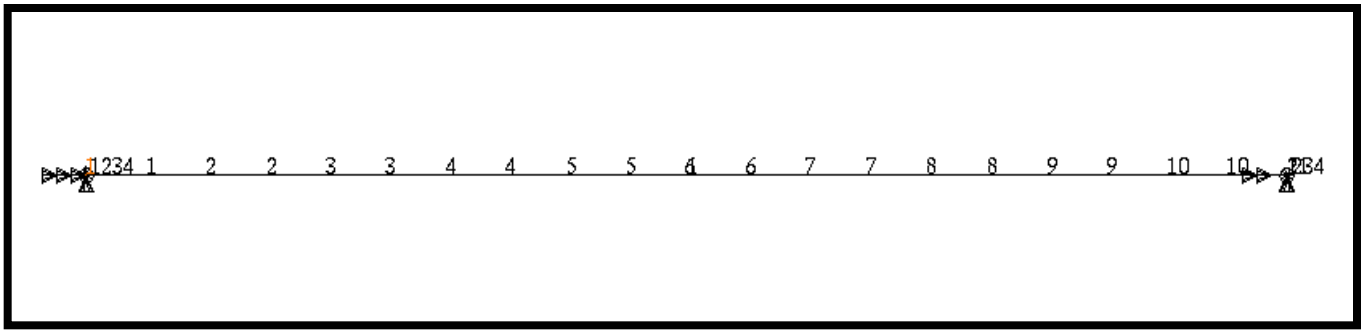


Table A-1a.1

Length	**1.0×10^{3} mm**
Elastic Modulus	**2.0684×10^{5} MPa**
Density	**7.8334×10^{-9}N-sec^2/mm^4**
Poisson's Ratio	**0.32**
Area	**5×10^{3} mm^2**
I_1	**1.0417×10^{6} mm^4**
Force	**1×10^{7} N**

Hand Calculations

$$f_n = \frac{K_n}{2\pi}\left[\frac{EIg}{Wl^4}\right]^{1/2}$$

$$f_n = K_n\left(\frac{1}{2\pi}\left[\frac{2.0684\times10^5(1.0417\times10^6)}{7.8334\times10^{-9}(5\times10^3)(1.0\times10^3)^4}\right]^{1/2}\right)$$

$$f_n = K_n(11.805)$$

From Theory

Mode	**K_n**	**f_n**
1	**9.87**	**116.51 Hz**
2	**39.5**	**466.28 Hz**
3	**88.8**	**1048.28 Hz**

Suggested Exercise Steps

- Explicitly generate a finite element representation of the beam structure. (i.e., the grids (GRID) and element connectivities (CBAR) should be defined manually.)
- Define material (MAT1) and element (PBARL) properties.
- Apply the fixed boundary constraints (SPC1).
- Prepare the model for a normal modes analysis (SOL 103 and PARAMS).
  - PARAM, COUPMASS, 1
  - EIGRL (To select Lanczos)
- Generate an input file and submit it to the MSC/NASTRAN solver for normal modes analysis.
- Review the results, specifically the eigenvalues.

```
ID SEMINAR,PROB1
```

```
CEND
```

```
BEGIN BULK
```

1	2	3	4	5	6	7	8	9	10

1	2	3	4	5	6	7	8	9	10

ENDDATA

Exercise Procedure:

1. Users who are not utilizing MSC/PATRAN for generating an input file should go to Step 10, otherwise, proceed to step 2.

2. Create a new database named **probap1.db**.

   File/New Database

   New Database Name **probap1**

   OK

   In the *New Model Preference* form set the following:

   Tolerance ◆ **Default**

   Analysis Code: **MSC/NASTRAN**

   OK

3. Activate the entity labels by selecting the Show Labels icon on the tool-bar.

    **Show Labels**

4. Create a curve.

   ◆ **Geometry**

   Action: **Create**

   Object: **Curve**

   Method: **XYZ**

   Vector Coordinates List **<1000, 0, 0>**

   Origin Coordinates List **[0, 0, 0]**

   Apply

5. Create the finite element model and mesh the surface.

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Curve**
Global Edge Length	**100**
Curve List	**Curve 1**

Apply

6. Create nodal displacements.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Displacement**
Type:	**Nodal**
New Set Name	**disp1**

Input Data...

Translations <T1 T2 T3>	**<0 0 0>**
Rotations <R1 R2 R3>	**<0 , , >**

OK

Select Application Region...

■ **Geometry**

Select Geometry Entities	**Point 1**

Add

OK

Apply

New Set Name	**disp2**

Input Data...

Translations <T1 T2 T3>	**< , 0 0>**

Rotations <R1 R2 R3> **<0 , , >**

OK

Select Application Region...

◆ **Geometry**

Select Geometry Entities **Point 2**

Add

OK

Apply

New Set Name **permanent_constraint**

Input Data...

Translations <T1 T2 T3> **< , , 0>**

Rotations <R1 R2 R3> **<0 , 0 , >**

OK

Select Application Region...

Select Geometry Entities **Curve 1**

Add

OK

Apply

7. Create a set of material properties for the bar.

◆ **Materials**

Action: **Create**

Object: **Isotropic**

Method: **Manual Input**

Material Name **mat_1**

Input Properties...

Elastic Modulus = **2.0684E5**

Poisson Ratio = **0.32**

Density = **7.8334E-9**

Apply

Cancel

8. Define the bar properties.

◆ **Properties**

Action: **Create**

Dimension: **1D**

Type: **Beam**

Property Set Name **bar**

Input Properties...

Material Name
*(Select from **Material Property Sets** box)* **m:mat_1**

Area **5E3**

Bar Orientation **Coord 0.2**

[Inertia 1,1] **1.0417E6**

OK

Select Members **Curve 1**

Add

Apply

9. Now, you will generate the input file for analysis.

◆ **Analysis**

Action: **Analyze**

Object: **Entire Model**

Method **Analysis Deck**

Job Name **probap1**

Solution Type...

Solution Type: **◆ NORMAL MODES**

Solution Parameters ...

<deselect Automatic Constraints> ❒ **Automatic Constraints**

Mass Calculation: **Coupled**

Data Deck Echo: **None**

OK

OK

Subcase Create...

Available Subcases **Default**

Subcase Parameters...

Number of Desired Roots = **3**

OK

Apply

Cancel *<to close form>*

Apply

An MSC/NASTRAN input file called **probap1.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 11.

Generating an input file for MSC/NASTRAN Users:

MSC/NASTRAN users can generate an input file using the data from Table A-1a.1. The result should be similar to the output below.

10. MSC/NASTRAN Input File: **probap1.dat**

```
SOL 103
TIME 600
CEND
TITLE = Normal Modes Example (SI UNITS)
SUBCASE 1
   METHOD = 1
   SPC = 1
   VECTOR(SORT1,REAL)=ALL
BEGIN BULK
PARAM    COUPMASS1
EIGRL    1                       3       0
PBAR     1       1       5000.   1.04+6
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       206840.         .32     7.83-9
GRID     1               0.      0.      0.              345
GRID     2               100.000 0.      0.              345
GRID     3               200.000 0.      0.              345
GRID     4               300.000 0.      0.              345
GRID     5               400.000 0.      0.              345
GRID     6               500.    0.      0.              345
GRID     7               600.000 0.      0.              345
GRID     8               700.000 0.      0.              345
GRID     9               800.000 0.      0.              345
GRID     10              900.000 0.      0.              345
GRID     11              1000.   0.      0.              345
SPC1     1       1234    1
SPC1     1       234     11
ENDDATA
```

Submit the input file for analysis

11. Submit the input file to MSC/NASTRAN for analysis.

    11a. To submit the MSC/PATRAN **.bdf** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran probap1.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

    11b. To submit the MSC/NASTRAN **.dat** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran probap1 scr=yes**. Monitor the run using the UNIX **ps** command.

12. When the run is completed, edit the **probap1.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.

13. While still editing **probap1.f06**, search for the word:

E I G E N (spaces are necessary)

What are the first three modes?

1st = _____Hz

2nd = _____Hz

3rd = _____Hz

Comparison of Results

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

R E A L E I G E N V A L U E S

MODE NO.	EXTRACTION ORDER	EIGENVALUE	RADIANS	CYCLES	GENERALIZED MASS	GENERALIZED STIFFNESS
1	1	5.352166E+05	7.315850E+02	1.164354E+02	1.000000E+00	5.352166E+05
2	2	8.561551E+06	2.926013E+03	4.656894E+0	1.000000E+00	8.561551E+06
3	3	4.329484E+07	6.579882E+03	1.047221E+03	1.000000E+00	4.329484E+07

15. MSC/NASTRAN Users have finished this exercise. MSC/PATRAN Users should proceed to the next step.

16. Proceed with the Reverse Translation process, that is importing the **probap1.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	**probap1.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=116.437**
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.