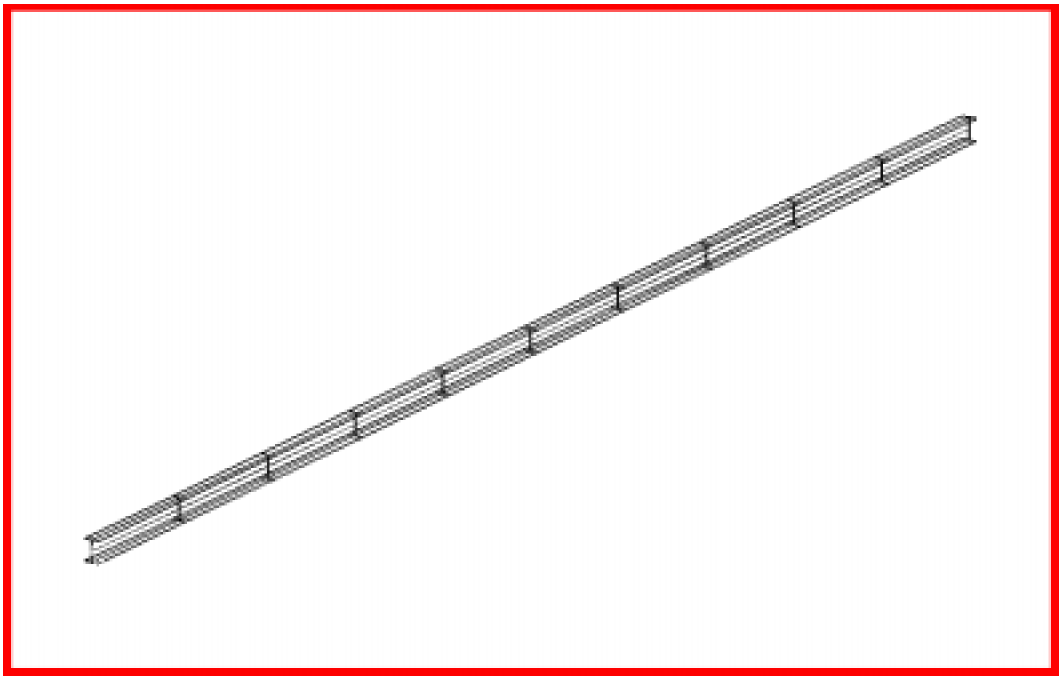
WORKSHOP PROBLEM 14a

Modal Analysis of a Beam



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 14a.1 below is a finite element representation of the beam. One end is contrained in all translation and the other is free to move in the X. Both ends are held in the X-rotation.

Figure 14a.1-Grid Coordinates and Element Connectivities

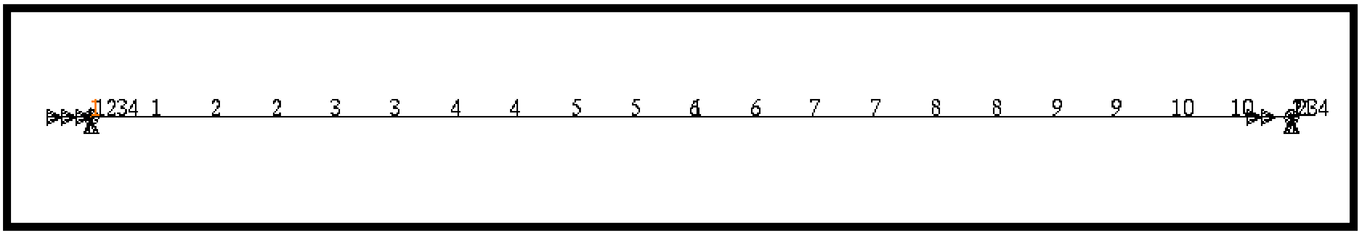


Figure 14a.2-Beam Cross Section

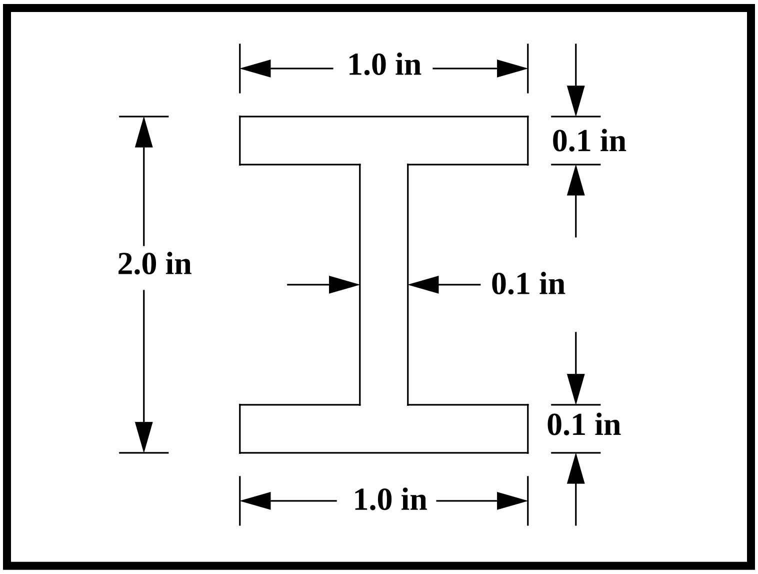


Table 14a.1

Length	**100 in**
Height	**2 in**
Width	**1 in**
Thickness	**0.100 in**
Area	**0.38** in^2
I_1	**0.229** in^4
I_2	**0.017** in^4

Hand Calculations

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

$$f_n = K_n\left(\frac{1}{2\pi}\left[\frac{10\times10^6(0.229)(386.4)}{(0.38)(0.101)(100)^4}\right]^{1/2}\right)$$

$$f_n = K_n(2.417)$$

* I of the strong axis is used since translational Z DOF has been constrained by the permanent constraint.

From Theory

Mode	K_n	f_n
1	**9.87**	**23.85 Hz**
2	**39.5**	**95.46 Hz**
3	**88.8**	**214.59 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, WTMASS, 0.00259
    - 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 **prob14a.db**.

   File/New Database

   New Database Name **prob14a**

   OK

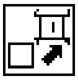
   In the *New Model Preference* form set the following:

   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 **<100, 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	**10**
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...

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 **alum**

Input Properties...

Elastic Modulus = **10.0E6**

Poisson Ratio = **.3**

Density = **.101**

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:alum**

■ **Use Beam Section** *<Click on Beam Library>*

New Section Name	**ibeam**
H	**2**
W1	**1**
W2	**1**
t	**0.1**
t1	**0.1**
t2	**0.1**

OK

Bar Orientation	**Coord 0.2**

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	**prob14a**
Solution Type...	
Solution Type:	◆ **NORMAL MODES**
Solution Parameters ...	
<deselect Automatic Constraints>	❐ **Automatic Constraints**
Mass Calculation:	**Coupled**
Data Deck Echo:	**None**
Wt. -Mass Conversion =	**.00259**
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 **prob14a.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 14a.1. The result should be similar to the output below.

10. MSC/NASTRAN Input File: **prob14a.dat**

```
SOL 103
TIME 600
CEND
TITLE = Normal Modes Example
SUBCASE 1
   METHOD = 1
   SPC = 1
   VECTOR=ALL
BEGIN BULK
PARAM,WTMASS,.00259
PARAM,COUPMASS,1
EIGRL    1                       3       0
PBARL   1       1              I                                +      A
+      A 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
GRID     11              100.    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 prob14a.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 prob14a scr=yes**. Monitor the run using the UNIX **ps** command.

12. When the run is completed, edit the **prob14a.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 **prob14a.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	2.239398E+04	1.496462E+02	2.381693E+01	1.000000E+00	2.239398E+04
2	2	3.549898E+05	5.958102E+02	9.482614E+01	1.000000E+00	3.549898E+05
3	3	1.771818E+06	1.331096E+03	2.118506E+02	1.000000E+00	1.771818E+06

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

15. Proceed with the Reverse Translation process, that is importing the **prob14a.op2** results file into MSC/PATRAN. To do this, return to the Analysis form and proceed as follows:

◆ **Analysis**

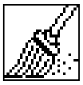
Action:	**Read Output2**
Object:	**Result Entities**
Method	**Translate**
Select Results File...	
Select Results File	**prob14a.op2**
OK	
Apply	

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

◆ **Results**

Form Type:	**Basic**
Select Results Cases	**1.1-Default, Mode 1:Freq=23.816**
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.