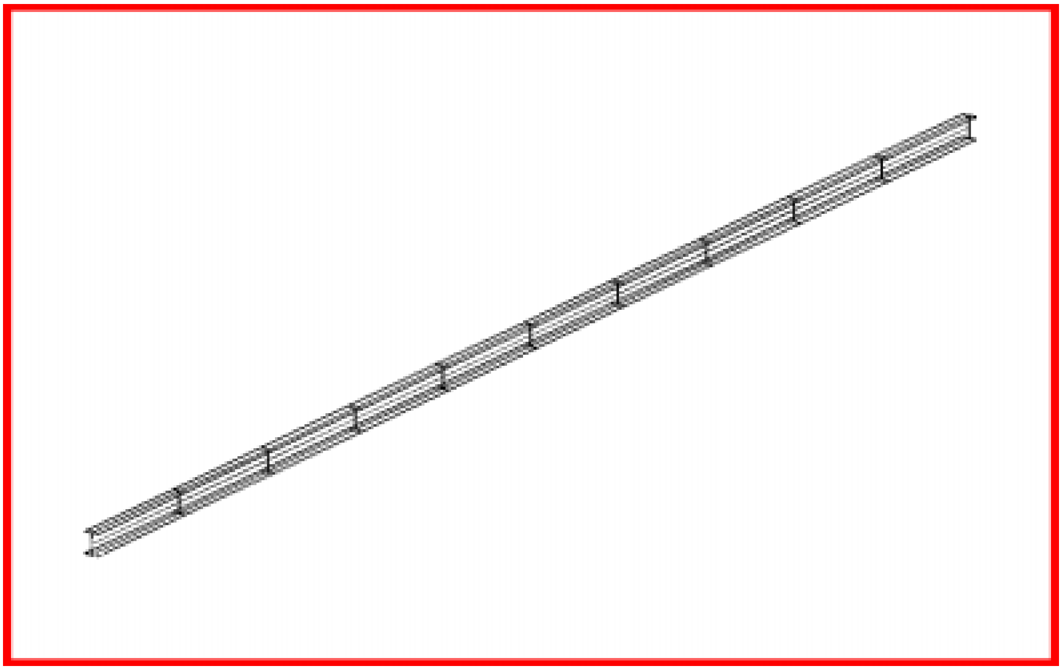
APPENDIX A

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-a.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-a.1

<u>Grid Coordinates and Element Connectivities</u>

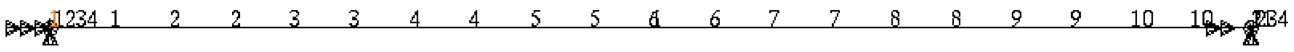


Table A-a.1

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**

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

Exercise Procedure:

1. Start up MSC/NASTRAN for Windows 3.0 and begin to create a new model.

Double click on the icon labeled MSC/NASTRAN for Windows V3.0 Model.

On the *Open Model File* form, select **New Model.**

Open Model File:	**New Model**

2. Create a material called **mat**.

From the pulldown menu, select **Model/Material.**

Model/Material...

Title:	**mat**
Youngs Modulus:	**2.0684e5**
Poisson's Ratio:	**.32**
Mass Density:	**7.8334e-9**

OK

Cancel

3. Create a property called **bar** to apply to the members of the beam.

From the pulldown menu, select **Model/Property**.

Model/Property...

Title:	**bar**

To select the material, click on the list icon next to the databox and select **mat**.

Material:	**1..mat**

Elem/Property Type...

Change the property type from plate elements (default) to bar elements. The reason why we model it as a bar instead of a beam is because this problem focus on a simple beam.

Line Element:	● **Bar**

OK

Area, A: **5e3**

Moment of Inertia, I1: **1.0417e6**

OK

Cancel

4. Create the necessary NASTRAN geometry.

Mesh/Between...

Property: **1..bar**

Mesh Size/ #Nodes: **11**

OK

	X:	*Y:*	*Z:*
Corner 1:	**0**	**0**	**0**

OK

	X:	*Y:*	*Z:*
Corner 2:	**1000**	**0**	**0**

OK

Now, specify the orientation vector for the bar elements.

	X:	*Y:*	*Z:*
Base:	**0**	**0**	**0**
Tip:	**0**	**1**	**0**

OK

To fit the display onto the screen, use the Autoscale feature.

View/Autoscale

5. Create the model constraints.

Before creating the appropriate constraints, a constraint set needs to be created. Do so by performing the following:

Model/Constraint/Set...

Title: **constraint**

OK

Now define the left end of the model.

Model/Constraint/Nodal...

Select **Node 1**.

OK

On the *DOF* box, select these boxes.

☒ **TX** ☒ **TY** ☒ **TZ**

☒ **RX**

OK

Next, define the right end of the model.

Select **Node 11**.

OK

On the *DOF* box, select these boxes.

☒ **TY** ☒ **TZ**

☒ **RX**

OK

Finally, define the permanent constraints of the model.

Select All

OK

On the *DOF* box, select these boxes.

☒ **TZ**

☒ **RX** ☒ **RY**

OK

A warning messaging will appear: “Selected Constraints Already Exist. OK to Overwrite (No = Combine)?” Select **No** to combine.

No

Cancel

6. Now create and submit the analysis file.

File/Export/Analysis Model...

Type: **2..Normal Modes/Eigenvalue**

OK

Change the directory to **C:\temp**.

File name: **appenA**

Write

☒ **Run Analysis**

Advanced...

Eigenvalues and Eigenvectors/ Number Desired: **3**

Mass: ● **Coupled**

OK

Problem ID: **Modal Analysis of a Beam**

OK

Under *Output Requests*, unselect all except:

☒ **Displacement**

OK

OK

When asked if you wish to save the model, respond **Yes**.

Yes

File name: **appenA**

Save

When the MSC/NASTRAN manager is through running, MSC/NASTRAN will be restored on your screen, and the *Message Review* form will appear. To read the messages, you could select **Show Details**. Since the analysis ran smoothly, we will not bother with the details this time.

Continue

7. List the results of the analysis.

To list the results, select the following:

List/Output/Query...

Under *the Output Set* pull down menu, what are the first three modes?

1st = ___________Hz

2nd = ___________Hz

3rd = ___________Hz

The answer is listed at the end of the exercise. Hit Cancel when you are done.

Cancel

8. Display the deformed plot on the screen.

Finally, you may now display the deformed plot. First, however, you may want to remove the labels and load and boundary constraint markers.

View/Options...

Quick Options...

Labels Off

☐ **Constraint**

Done

OK

Plot the deformation of the beam.

View/Select...

Deformed Style: ● **Deform**

Deformed and Contour Data...

From the *Output Set* pull down menu, select a mode case.

Output Vectors/Deformation: **1..Total Translation**

OK

OK

When finished, reset the display then exit MSC/NASTRAN for Windows.

View/Select...

Deformed Style: ● **Non-Model Only**

OK

File/Exit

This concludes this exercise.

Mode 1	**116.51 Hz**
Mode 2	**466.09 Hz**
Mode 3	**1049.14 Hz**