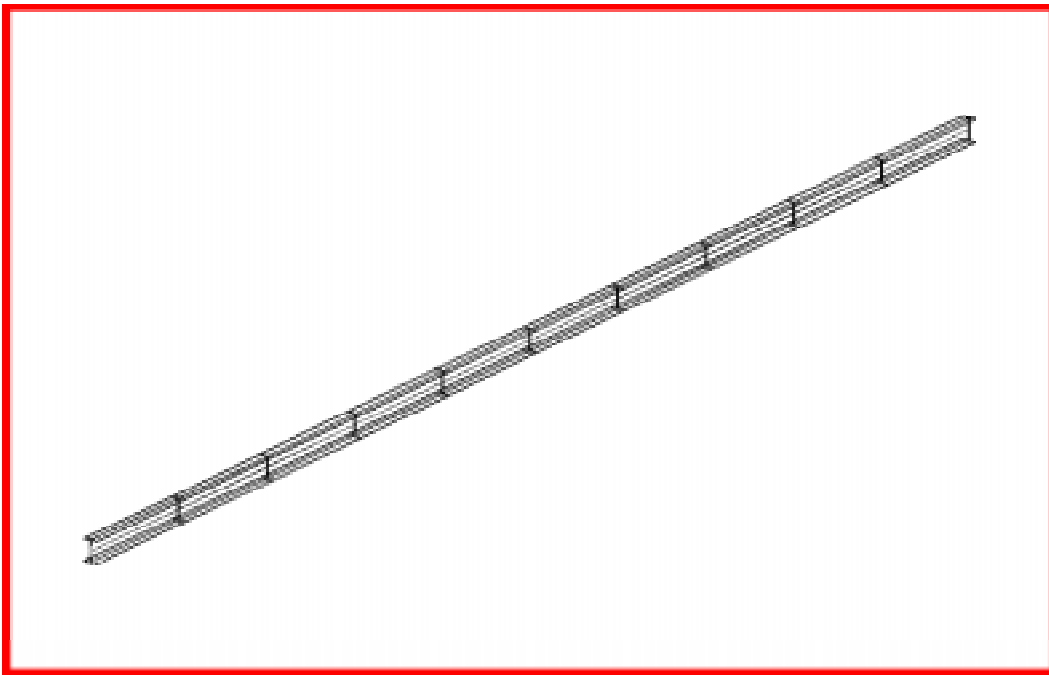


---

**WORKSHOP PROBLEM 9b**

*Normal Modes with  
Differential Stiffness*



**Objectives**

- Analyze a stiffened beam for normal modes
- Produce MSC.Nastran for Windows input file that represent beam and load
- Submit for analysis
- Find normal modes (natural frequencies)

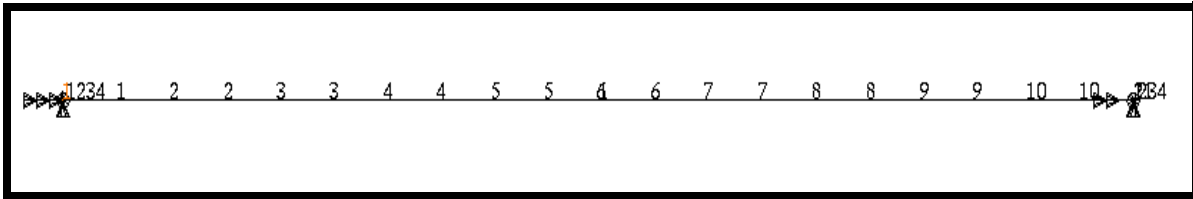


**Model Description:**

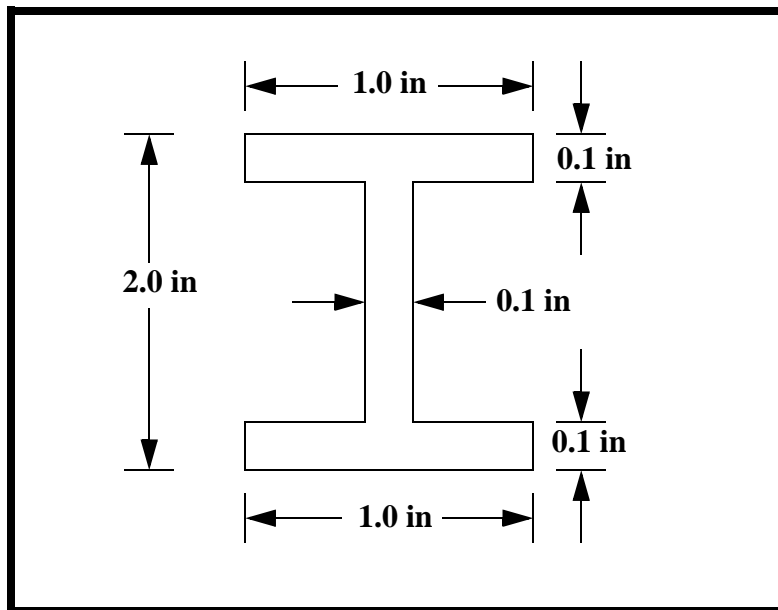
The goal of this example is to analyze a stiffened model. In this case, the beam from Problem 9a. is loaded with a 500 lb axially force.

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 in the x direction. The applied load creates differential stiffness terms which change the natural frequencies.

**Figure 9b.1 - Grid Coordinates and Element Connectivities**



**Figure 9b.2 - Beam Cross Section**



**Table 9b.1 - Beam Dimensions**

<b>Length</b>	<b>100 in</b>
<b>Height</b>	<b>2 in</b>
<b>Width</b>	<b>1 in</b>
<b>Thickness</b>	<b>0.100 in</b>
<b>Area</b>	<b>0.38 in<sup>2</sup></b>
<b>I<sub>1</sub></b>	<b>0.229 in<sup>4</sup></b>
<b>I<sub>2</sub></b>	<b>0.017 in<sup>4</sup></b>

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,  $K_n=9.87$

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

$$f_n = 26.36 \text{ Hz}$$

For Static Load

$$\Delta = \frac{PL}{AE}$$

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

$$\Delta = 0.0132$$

---

## Exercise Procedure:

1. Start up MSC.Nastran for Windows 4.0 and begin to create a new model.

Double click on the icon labeled MSC.Nastran for Windows V4.0.

On the *Open Model File* form, select the data base from **prob9.mod**.

*Open Model File:*

**View/Select** **<F5>**

Under the *Deformed Style* window, make the following selection:

*Deformed Style:*  **None - Model Only**

**View/Autoscale** **(Ctrl-A)**

2. Create the load set.

**Model/Load/Set...**

*Title:*

3. Create the point loads.

**Model/Load/Nodal...**

Select **Node 11**.

*(highlight)*

*FX*

**Cancel**

- In order to include the effects of the differential stiffness due to the applied load we must define the load to use in an analysis subcase and then in a second subcase define the eigenvalue extraction method with a method command and the static subcase with a statsub command. We set the loads analysis type to static but then manually enter the modal analysis commands.

**File/Export/Analysis Model...**

*Analysis Type:*

**1.. Static**

**OK**

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

*File name:*

**prob9b**

**Write**

**Run Analysis**

**Advanced...**

**Solution Number**

**Semodes**

**OK**

We first write a master case to define our output request and constraint.

Under the Output requests, unselect everything except:

**Displacement**

Under analysis case request unselect the load case

*Analysis Case Request*

**Loads =**

**Write Case...**

**OK**

Select the load case

**Loads =**

**1.. pull**

**Write Case...**

---

**OK**

Now, make a subcase to calculate the Modes.

Unselect the load case.

**Loads**

**Write Case...**

**OK**

Manually select the static load case and eigenvalue method

**Type Input...**

*Current Line:*

**STATSUB = 1**

**More**

*Current Line:*

**METHOD = 1**

**OK**

**Done**

**WTMASS**

**0.00259**

We must manually enter the eigenvalue extraction data and the optional coupmass parameter in bulk data

**Type Input...**

*Current Line:*

**EIGRL, 1,,3**

**More**

*Current Line:*

**PARAM, COUPMASS, 1**

**OK**

**OK**

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

**Yes**

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**

5. Review the results of the analysis.

**View/Select**

**<F5>**

Under the *Deformed Style* window, make the following selection:

*Deformed Style:*

**Deform**

**Deformed and Contour Data...**

**5.. Mode1, 26.32597**

*Deformation:*

**1.. Total Translation**

**OK**

**OK**

What are the three natural frequencies?

1st = \_\_\_\_\_ Hz

2nd = \_\_\_\_\_ Hz

3rd = \_\_\_\_\_ Hz

Are the answers consistent with the theoretical solutions?

---

6. List the displacement results of the analysis.

To list the displacement results, select the following:

**List/Output/Query...**

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

<i>Output Set:</i>	<input type="text" value="4.. MSC/NASTRAN CASE 1"/>
<i>Category:</i>	<input type="text" value="1..Displacement"/>
<i>Entity:</i>	<input type="radio"/> Node
<i>ID:</i>	<input type="text" value="11"/>
<input type="text" value="OK"/>	

What is the total displacement?

Displacement = \_\_\_\_\_

The answer is listed at the end of the exercise.

When finished, exit MSC.Nastran for Windows.

**File/Exit**

This concludes this exercise.

<i>Displacement</i>	0.013158
<i>Mode 3</i>	214.590 Hz
<i>Mode 2</i>	97.460 Hz
<i>Mode 1</i>	26.326 Hz

