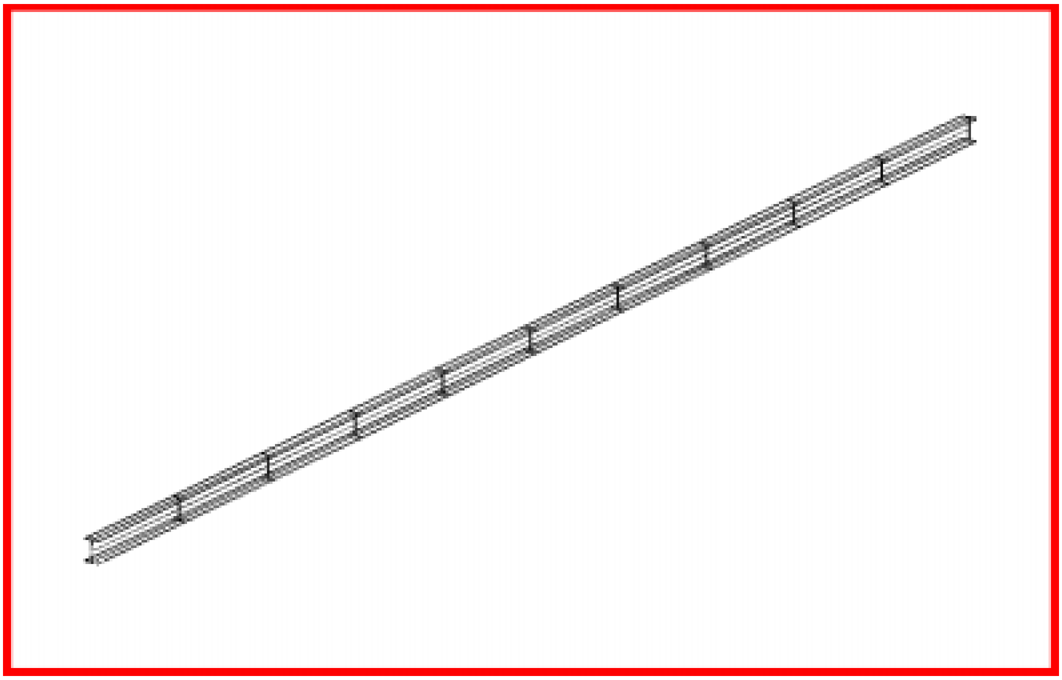
WORKSHOP PROBLEM 9b

Normal Modes with Differential Stiffness



Objectives

- Analyze a stiffened beam for normal modes.
- Produce NASTRAN 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. with a 500 lb force applied.

Figure 9b.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.

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 9b.1 - Grid Coordinates and Element Connectivities

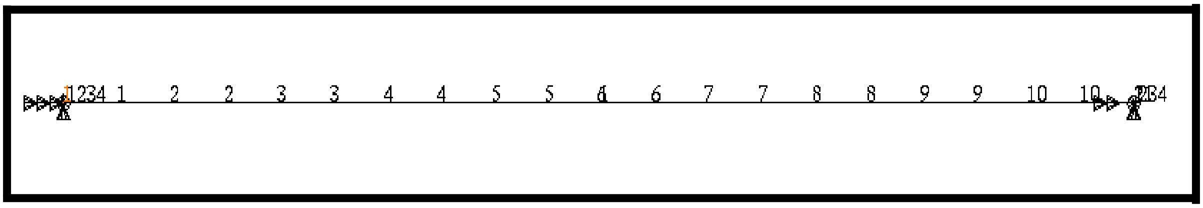


Figure 9b.2 - Beam Cross Section

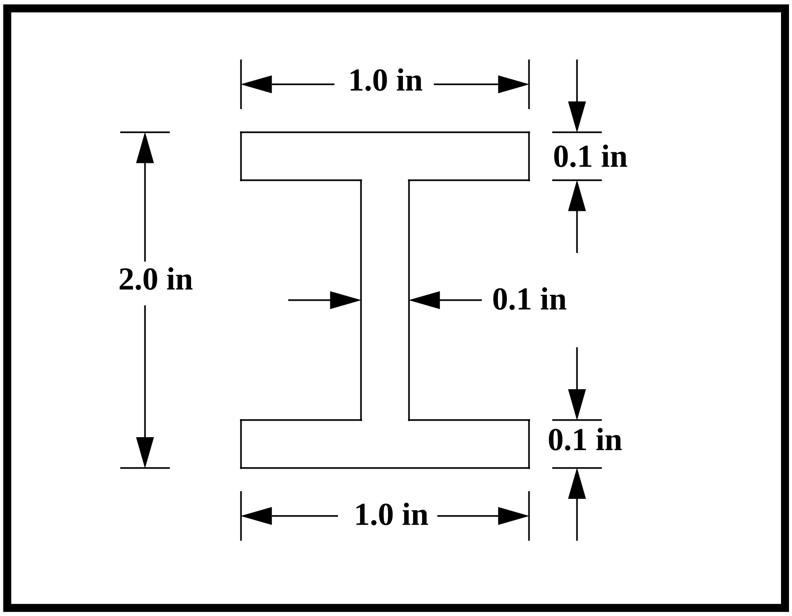


Table 9b.1 - Beam Dimensions

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**

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

$$f_n = \frac{9.87}{2\pi}\left[\frac{10\times10^6(0.229)(386.4)}{(0.38)(0.101)(100)^4}\mathrm{x}\left(1 + \frac{1}{9.87}\frac{(500)(100)^2}{(10\times10^6)(0.229)}\right)\right]^{1/2}$$

$$f_n = 26.36Hz$$

For Static Load

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

$$\Delta = \frac{500(100)}{0.38(10\times10^6)}$$

$$\Delta = 0.0132$$

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.

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

Open Model File: **New Model**

2. Import **prob1.DAT.**

File/Import/Analysis Model...

● **Nastran** **MSC/Nastran**

OK

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

File name: **prob9a.DAT**

Open

When ask, "Ok, to Adjust all massess by PARAM, WTMASS factor of 0.00259?", answer **No**. This information will be entered during analysis.

No

To reset the display of the model do the following.

View/Redraw

View/Autoscale

OK

3. Create the load set.

Create/Load/Set...

Title: **pull**

OK

4. Define the options for a nonlinear analysis.

Model/Load/Nonlinear Analysis...

Solution Type: ● **Static**

Defaults...

Basic/Number of Increments: **5**

OK

5. Create the point loads.

Model/Load/Nodal...

Select **Node 11**.

OK

(highlight) **Force**

FX ☒ **500**

OK

Cancel

6. Submit the job for analysis.

File/Export/Anaylsis Model...

Analysis Type: **10..Nonlinear Static**

OK

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

File name: **prob9b**

Write

☒ **Run Analysis**

Advanced...

Problem ID: **Normal Modes w/ Differential Stiffness**

OK

Under *Output Requests*, unselect everything except:

☒ **Displacement**

Also, change output to:

2..Print and PostProcess

Type Input...

Current Line: **METHOD = 10**

OK

OK

Under *PARAM*, enter the following:

☒ *WTMASS* **.00259**

Type Input...

Current Line: **PARAM, NMLOOP, 5**

More

Current Line: **EIGRL, 10, , , 3**

More

Current Line: **PARAM, COUPMASS, 1**

OK

OK

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

Yes

File name: **prob9b**

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

When asked if it is "OK to Begin Reading File C:\TEMP\prob9b.xdb", respond **Yes**.

Yes

7. Determine the results of the analysis.

In order to find the 3 eigenvalues go to **C:/temp** and view the **prob9b.f06** file. Search for the section of the file that resembles the one shown on the next page.

Under *CYCLES*, what are the three natural frequencies?

1st = ___________Hz

2nd = ___________Hz

3rd = ___________Hz

Are the answers consistent with the theoretical solutions?

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	2	##	##	##	##	##
3	3	##	##	##	##	##

8. 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?

Output Set:	**#..Case 1 Step 1.000000**
Category:	**1..Displacement**
Entity:	● **Node**
ID:	**11**

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.

Mode 1	**26.360 Hz**
Mode 2	**98.033 Hz**
Mode 3	**217.434 Hz**
Displacement	**0.013158**