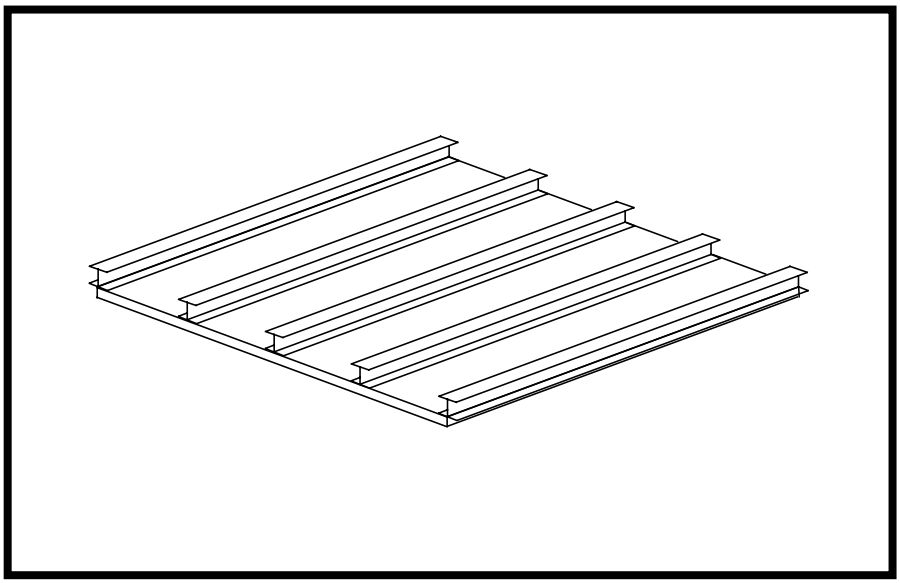
LESSON 13

Normal Modes Analysis of a Simply-Supported Stiffened Plate

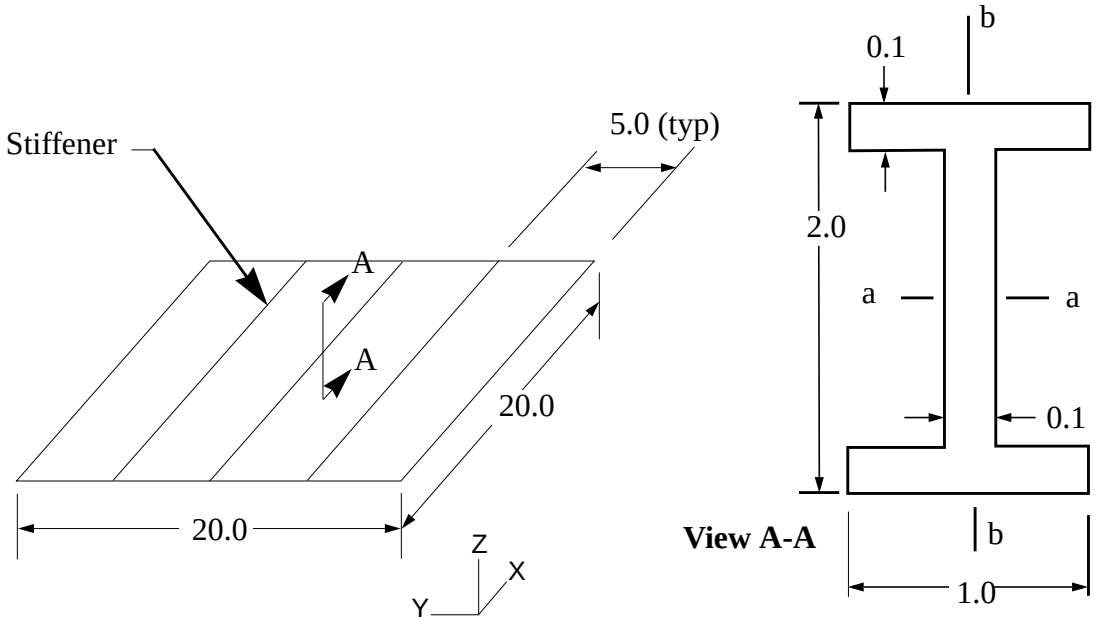


Objectives:

- Manually convert a Linear Static analysis (Sol 101) input file to a Normal Modes analysis (Sol 103) input file.
- Learn how to generate weight information for your model.
- Submit a Normal Modes analysis to MSC/NASTRAN.
- Import both model AND results into MSC/PATRAN via the MSC/NASTRAN binary results file (.op2).
- Review the results of a Normal Modes analysis.
- Visualize modal shapes.

Model Description:

The model used for this exercise is identical to the model used for Lesson 8.



Elastic Modulus:	**10.3E6 psi**
Poisson Ratio:	**0.3**
Density:	**0.101 lbs/in^3**
Plate Thickness:	**0.1 in**
Bar cross sectional area:	**0.38 in^2**
I_{aa}:	**0.2293 in^4**
I_{bb}:	**0.0168 in^4**
J:	**0.0013 in^4**

Exercise Procedure:

1. The input file you will be working with is called lesson13_work.bdf. This input file is identical to solution input file for Exercise 8. Your task is to edit this file so that the MSC/NASTRAN solver will extract the **first five** normal modes.

2. Input file items that you will need to consider:

Entry	Comments
FMS	
SOL	What solution sequence should we be using for a Normal Modes analysis?
Case Control	
LOAD	What is the significance of an externally applied load with respect to an eigenvalue problem?
METHOD	How does the solver know what eigenvalue extraction parameters to use?
SPCFORCES, STRESS	What is the significance of the force & stress results with respect to an eigenvalue problem?
Bulk Data (PARAMs)	
COUPMASS	Which mass matrix formulation should be used? Lumped or consistent?
GRDPNT	What useful information can result from this entry?
EIGRL	How does this entry relate to a Normal Modes analysis?

3. After you complete your revisions, submit the input file to the MSC/NASTRAN solver for analysis. To do this, find an available xterm window and at the prompt enter:

nastran lesson13_work.bdf scr=yes

Monitor the run using the UNIX **ps** command.

3a. When the run is completed, edit the **lesson13_work.f06** file and search for the word **FATAL**. If none exists, search for the word **WARNING**. Determine whether or not existing WARNING messages indicate modeling errors.

3b. While still editing **lesson13_work.f06**, search for the word:

W E I G H T (spaces are necessary)

What is the weight of our structure?

weight = _____

Where is the Center of the Gravity (C.G.) located?

X-C.G.= _____

Y-C.G.= _____

Z-C.G.= _____

Search for the word:

R E A L (spaces are necessary).

What are the first 5 modal frequencies for our structure?

mode 1	_____	**Hz**
mode 2	_____	**Hz**
mode 3	_____	**Hz**
mode 4	_____	**Hz**
mode 5	_____	**Hz**

4. Finally, visualize the results in MSC/PATRAN. Import **both** model & results into a **new** MSC/PATRAN database via the **lesson13_work.op2** results file.

◆ **Analysis**

Action: **Read Output2**

Object: **Both**

Method: **Translate**

Select Results File...

Filter

Selected Results File: *select the desired **.op2** file*

OK

Apply

When translation is completed and the Heartbeat turns green, bring up the **Results** form.

◆ **Results**

Action: **Create**

Object: **Quick Plot**

Choose the desired result case in the **Select Result Cases** list and select the result in the **Select Deformation Result** list. And hit **Apply** to view the result(s) in the viewport. Do this for each of your mode shapes.

If you wish to reset your display graphics to the state it was in before you began post-processing your model, remember to select the broom icon.

 **Reset Graphics**

Quit MSC/PATRAN when you have completed this exercise.