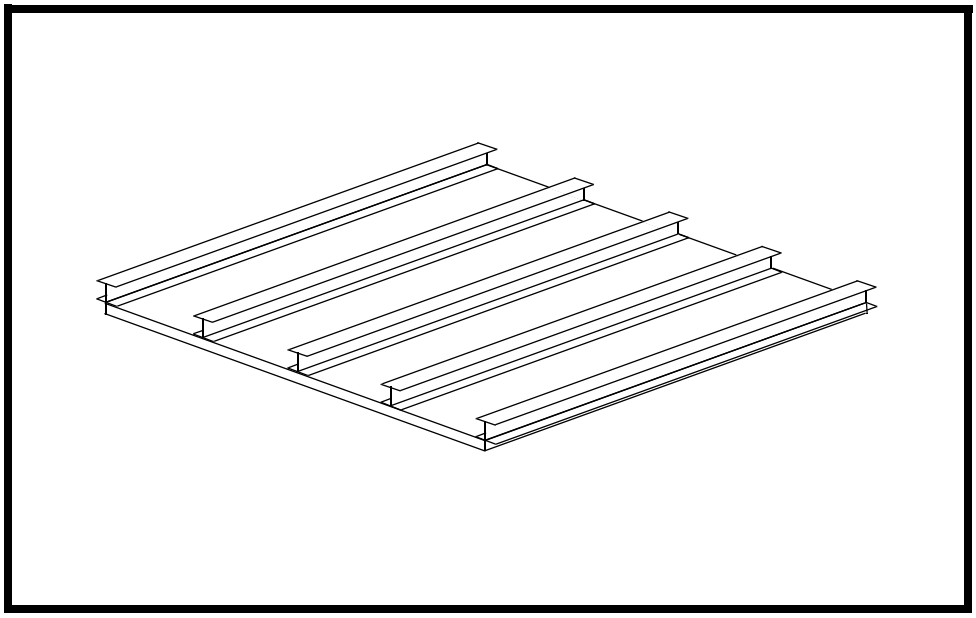
WORKSHOP 23

Modal Analysis of a Simply-Supported Stiffened Plate



Objectives:

- Manually convert a linear static analysis input file to a normal modes analysis input file.
- Learn how to generate weight information for your model.
- Submit a modal analysis to MSC.Nastran.
- Review the results and mode shapes from the analysis.

Model Description:

The model used for this exercise is identical to the model used for the previous exercise.

Figure 23.1 - Plate Geometry

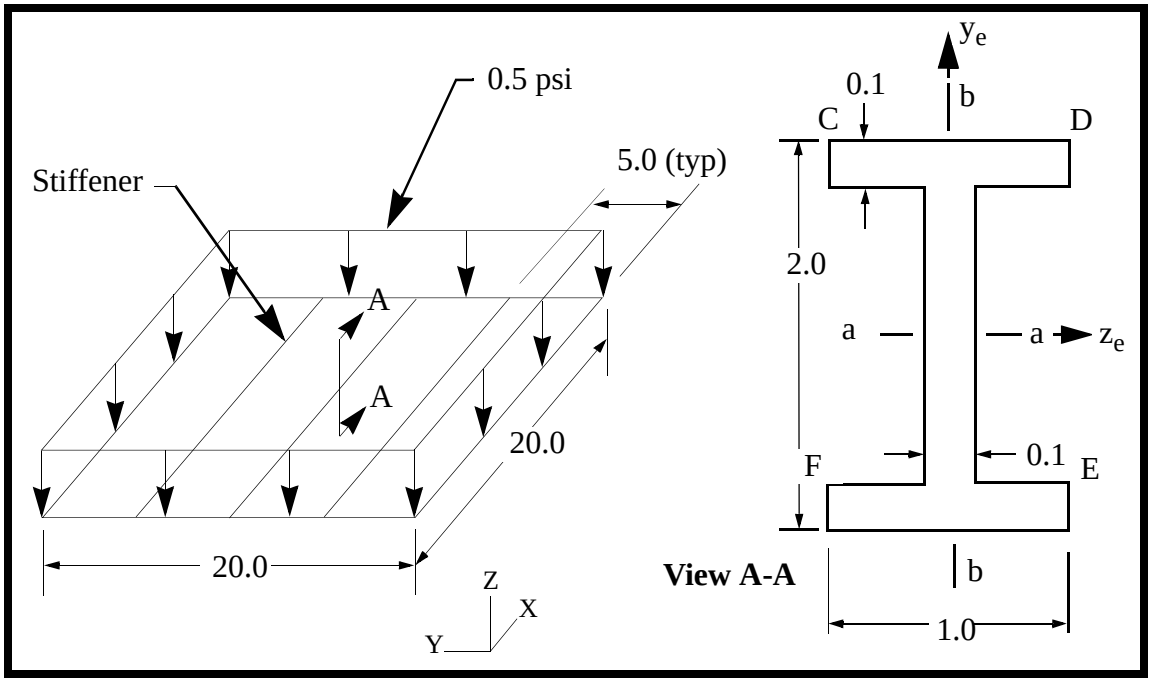


Table 23.1 - Material Properties

Youngs Modulus:	**10.3E+06 psi**
Poisson's 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. Start up MSC.Nastran for Windows V4.0 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, change to **C:\temp** directory and open the model from the previous exercise.

   Open Model File: **plate**

   Open

2. Create an input file for a modal analysis.

   File/Export/Analysis Model...

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

   OK

   File Name: **modal**

   Write

   Additional Info: ☒ **Run Analysis**

   Advanced...

   Eigenvalues & Eigenvectors/ Number Desired: **5**

   OK

   Problem ID: **Modal Sample Problem**

   OK

   Unselect all *Output Requests* except for **Displacement**.

   ☒ **Displacement**

   OK

   ☒ **WTMASS** **0.00259**

   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 successfully, we will not bother with the details this time.

Continue

3. Turn off all labels and load and boundary constraint markers.

View/Options...

Quick Options...

Labels Off

☐ **Load - Pressure**

☐ **Constraint**

Done

OK

4. Plot the desired mode shapes.

View/Select...

Deformed Style: ● **Deform**

Deformed and Contour Data...

Click on the *Output Set* listbox to get all modal frequencies, and answer the following questions:

What is the frequency for:
Mode 1 = _____
Mode 2 = _____
Mode 3 = _____
Mode 4 = _____
Mode 5 = _____

To view a mode shape, select one of the modes.

OK

OK

5. Using coupled mass matrix for modal analysis.

The previous results were calculated based on a lumped mass matrix. Repeat **Step 2** with the following modification:

.

(refer to Step 2 and answer **Yes** when asking for overwrite)

.

Advanced...

Eigenvalues & Eigenvectors/ Number Desired: **5**

Mass: ● **Coupled**

OK

OK

OK

☒ **WTMASS** **0.00259**

OK

Continue on just as before, and answer the following questions:

What is the frequency (with Coupled Mass) for:
Mode 1 = _____
Mode 2 = _____
Mode 3 = _____
Mode 4 = _____
Mode 5 = _____

This concludes the exercise.

Modal Analysis of a Stiffened Plate

	w/o COUPMASS	**w/ COUPMASS**
Mode 1	**17.56**	**17.61**
Mode 2	**36.25**	**36.72**
Mode 3	**68.36**	**69.18**
Mode 4	**91.38**	**93.37**
Mode 5	**147.18**	**150.74**