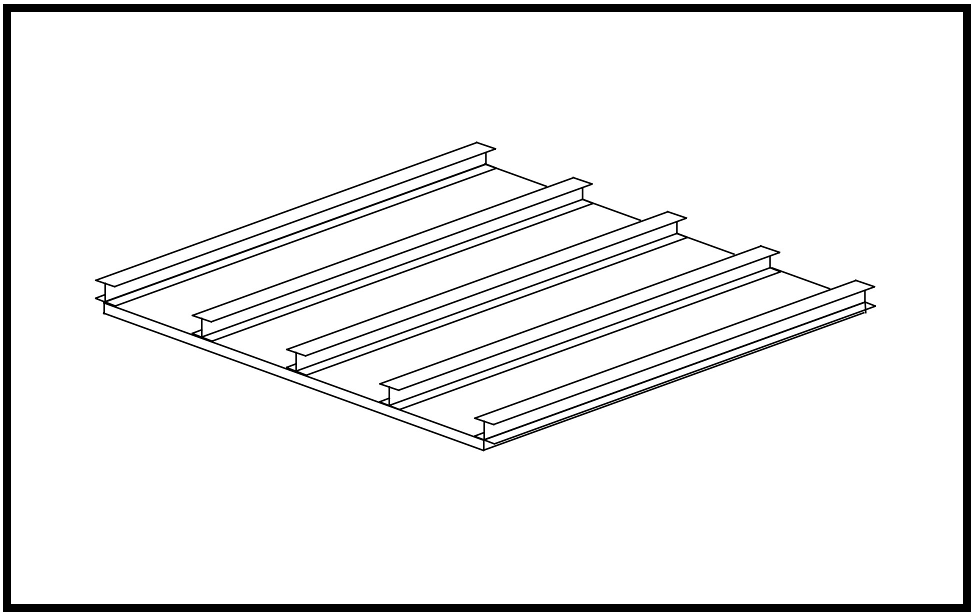
WORKSHOP 7

Linear Static Analysis of a Simply-Supported Stiffened Plate



Objectives:

- Create a geometric representation of a stiffened plate.
- Use the geometry model to define an analysis model comprised of plate & bar elements.
- Run an MSC/NASTRAN linear static analysis.
- View analysis results.

Model Description:

Below is a finite element representation of the stiffened plate shown on the title page. The plate is 0.1 inches thick, therefore thin-shell theory applies. I-beam stiffeners are mounted as shown. The structure is simply supported on its four corners and a uniform pressure of 0.5 psi is applied to the surface of the plate.

Hint: Because the centroidal axes of the stiffeners do not coincide with the mid-plane of the plate, you will need to account for this when you define the element properties for the stiffeners.

Figure 7.1 - Plate Geometry

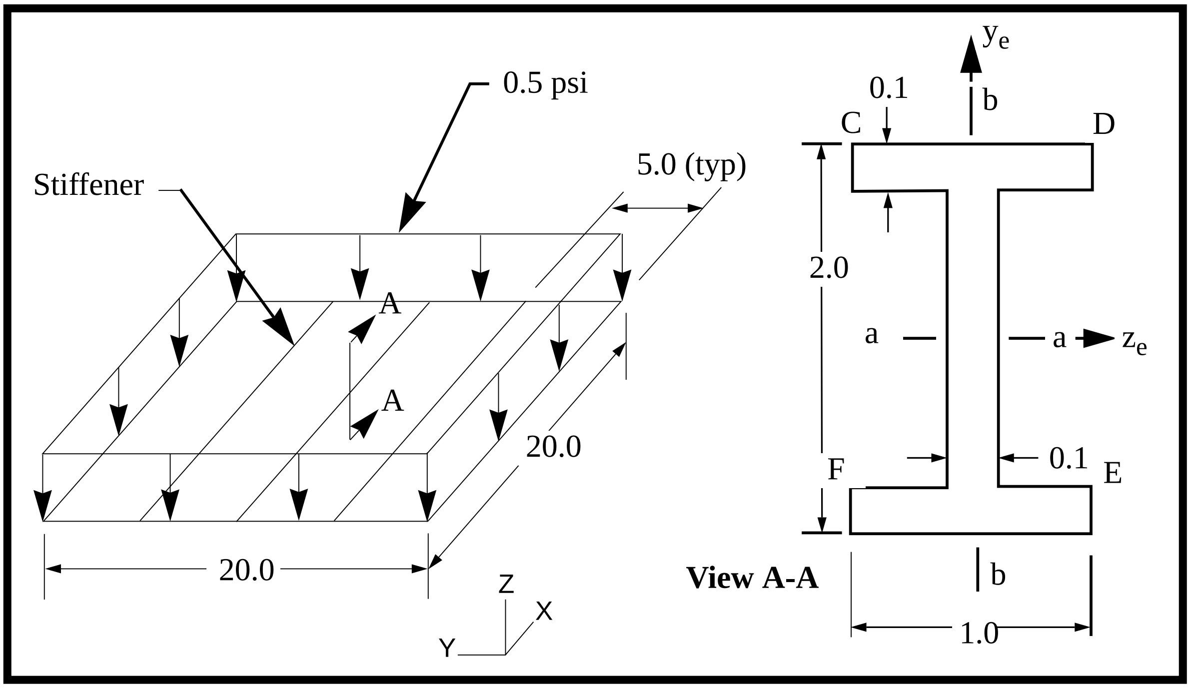


Table 7.1 -Material Properties

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. Start up MSC/NASTRAN for Windows V3.0 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. Create a material called **mat_1**.

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

Model/Material...

Title:	**mat_1**
Youngs Modulus:	**10.3e6**
Poisson's Ratio:	**.3**
Mass Density:	**.101**

OK

Cancel

3. Create a property called **plate** to apply to the members of the plate itself.

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

Model/Property...

Title:	**plate**

To select the material, click on the **list** icon next to the databox and select **mat_1**.

Material:	**1..mat_1**
Thickness, Tavg or T1:	**0.1**

OK

4. Create a property called **stiffener** for the bar elements of the model.

Title: **stiffener**

Material: **1..mat_1**

Elem/Property Type...

Change the property type from plate elements (default) to bar elements.

Line Elements: ● **Bar**

OK

A: **0.38**

I1: **0.2293**

I2: **0.0168**

J: **0.0013**

Note: Be certain that you understand the assumed bar orientation.

Stress Recovery:		
	Y	**Z**
1	**1**	**.5**
2	**1**	**-.5**
3	**-1**	**-.5**
4	**-1**	**.5**

OK

Cancel

5. Create the NASTRAN geometry for the plate.

Mesh/Between...

Property: **1..plate**

Mesh Size/ #Nodes/ Dir 1:	**13**
Mesh Size/ #Nodes/ Dir 2:	**11**

OK

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

OK

Repeat this process for the other 3 corners.

X:	*Y:*	*Z:*	
20	**0**	**0**	**OK**
20	**20**	**0**	**OK**
0	**20**	**0**	**OK**

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

View/Autoscale

6. Create the NASTRAN geometry for the stiffeners.

Mesh/Between...

Property:	**2..stiffener**
Mesh Size/ #Nodes/ Dir 1:	**11**
Corner Nodes:	**1**
	131

OK

Now, specify the orientation vector for the bar elements.

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

OK

Repeat the **Mesh/Between** procedure for the following corner node pairs:

NOTE: You will have to re-enter both the number of nodes and orientation vector for each pair of corner nodes, since FEMAP resets the values!

Corner Nodes:	**4**
	134
Corner Nodes:	**7**
	137
Corner Nodes:	**10**
	140
Corner Nodes:	**13**
	143

7. Remove coincident grids from the model.

As you may have noticed on the screen, additional grids were created while generating bar elements. To eliminate these coincident grids, do the following:

Tools/Check/Coincident Nodes...

Select All

OK

When asked if you wish to specify an additional range of nodes to merge, respond **No**.

No

Options: ☒ **Merge Coincident Entities**

OK

As the message window states, 55 nodes have now been merged.

8. Offset the stiffeners to correctly represent the model.

To properly model the bar stiffness, we will need to incorporate the bar offset property.

Modify/Update Elements/Offsets...

Method ^

✓ **Type**

Type: **2..L Bar**

OK

☒ **Update End A**

☒ **Set EndB = EndA**

OK

	X:	*Y:*	*Z:*
Base:	**0**	**0**	**0**
Tip:	**0**	**0**	**-1.05**

OK

In order to examine the offsets, you may want to rotate the model.

View/Rotate...

Isometric

When done viewing the offset, set the display back the way it was.

XY Top

OK

9. 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 relevant constraint for the model.

Model/Constraint/Nodal...

Select the four extreme corners, **Nodes 144, 13, 154, & 198**.

OK

On the *DOF* box, select all translations.

☒ **TX** ☒ **TY** ☒ **TZ**

OK

Cancel

10. Create the model loading.

Like the constraints, a load set must first be created before creating the appropriate model loading.

Model/Load/Set...

Title: **pressure_loading**

OK

Now, define the 0.5 psi surface pressure.

Model/Load/Elemental...

Select All

OK

Highlight **Pressure**.

Pressure

Pressure Value: **0.5**

OK

Face: **2**

OK

Cancel

11. Create the input file for analysis.

File/Export/Analysis Model...

Analysis Format/Type: **1..Static**

OK

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

File Name: **plate**

Write

☒ **Run Analysis**

Advanced...

Problem ID: **Stiffened Plate Sample Problem**

OK

Under *Output Requests*, unselect the following:

☐ **Applied Load**

☐ **Element Force**

OK

OK

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

Yes

File Name: **plate**

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

12. List the results of the analysis.

To list the results, select the following:

List/Output/Unformatted...

Select All

OK

Unselect **All Vectors** and instead select **T3 Translation**.

☐ **All Vectors, or**

4..T3 Translation

OK

NOTE: You may want to expand the message box in order to view the results.

Answer the following questions using the results. The answers are listed at the end of the exercise.

What is the minimum displacement in the Z direction?

Min Disp Z = _______

Also, repeat the List/Output/Unformatted procedure to find the answer to the second question.

What is the maximum Von Mises stress on the top of the plate?

Max Von Mises = _______

13. 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

☐ **Load - Pressure**

☐ **Constraint**

Done

OK

Plot the deformation of the beam.

View/Select...

Deformed Style: ● **Deform**

Deformed and Contour Data...

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

OK

OK

To get a better view of the deformation, you will want to rotate the model as follows:

View/Rotate...

Dimetric

OK

14. Add the contour plot of Von Mises stress to the deformed plot.

View/Select...

Contour Style: ● **Contour**

Deformed and Contour Data...

Output Vectors/ Contour: **7033..Plate Top VonMises Stress**

OK

OK

Return the model to the original display.

View/Select...

Deformed Style: ● **None-Model Only**

Contour Style: ● **None-Model Only**

OK

This concludes the exercise.

Min Z Disp:	***-1.01626***
Max Von Mises:	***13522.5***