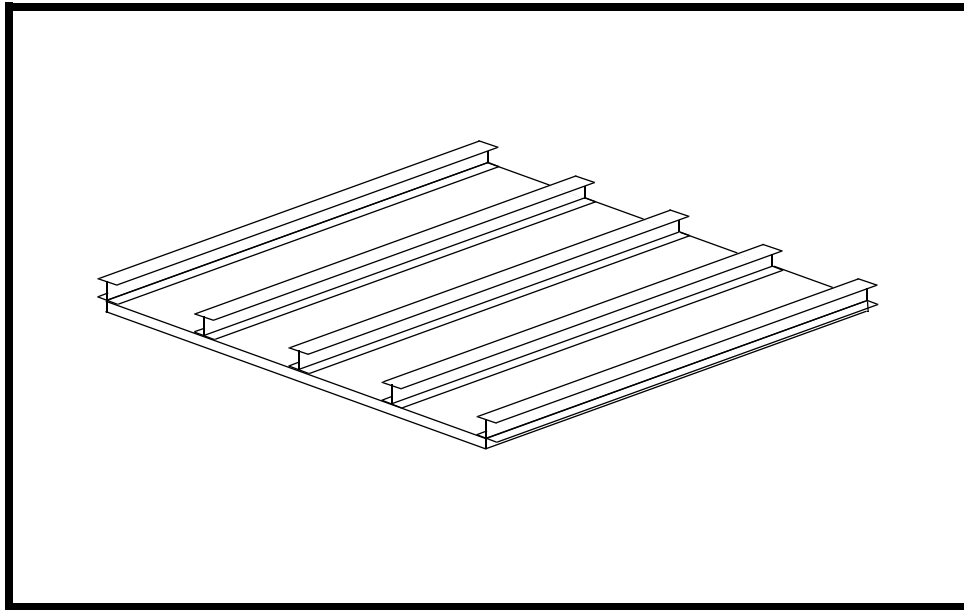

WORKSHOP 19

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 19.1 - Plate Geometry

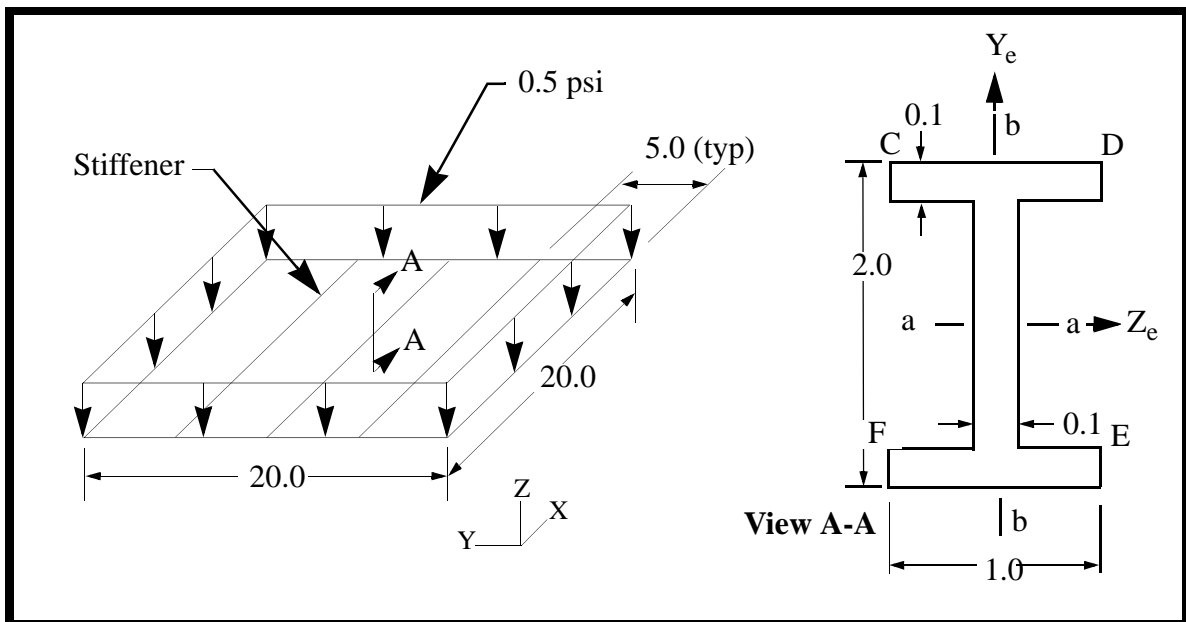


Table 19.1 -Material Properties

Youngs Modulus:	10.3E+06 psi
Poisson's Ratio:	0.3
Density:	0.101 lbs/in³
Plate Thickness:	0.1 in
Bar Cross-Sectional Area:	0.38 in²
I_{aa}:	0.2293 in⁴
I_{bb}:	0.0168 in⁴
J:	0.0013 in⁴

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, select **New Model**.

Open Model File:

2. Create a material called **mat_1**.

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

Model/Material...

Title:

Youngs Modulus:

Poisson's Ratio:

Mass Density:

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:

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

Material:

Thickness, Tavg or T1:

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

Title:

Material:

Elem/Property Type...

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

Line Elements: Bar

OK

A:

I1:

I2:

J:

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

<i>Stress Recovery:</i>		
	Y	Z
1	1	.5
2	1	-.5
3	-1	-.5
4	-1	.5

OK

Cancel

5. Create the MSC.Nastran geometry for the plate.

Mesh/Between...

Property:

Mesh Size/ #Nodes/ Dir 1:

Mesh Size/ #Nodes/ Dir 2:

Corner 1: X: Y: Z:

Repeat this process for the other 3 corners.

X:	Y:	Z:	
<input type="text" value="20"/>	<input type="text" value="0"/>	<input type="text" value="0"/>	<input type="text" value="OK"/>
<input type="text" value="20"/>	<input type="text" value="20"/>	<input type="text" value="0"/>	<input type="text" value="OK"/>
<input type="text" value="0"/>	<input type="text" value="20"/>	<input type="text" value="0"/>	<input type="text" value="OK"/>

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

View/Autoscale (Ctrl-A)

Turn off the workplane.

Tools/Workplane... (F2)

Draw Workplane

View/Regenerate... (Ctrl-G)

To better visualize the siffener connectivity, we will turn on the node label first.

View/Option...

Option:

Label Mode:

6. Create the MSC.Nastran geometry for the stiffeners.

Mesh/Between...

<i>Property:</i>	<input type="text" value="2..stiffener"/>
<i>Mesh Size/ #Nodes/ Dir 1:</i>	<input type="text" value="11"/>
<i>Corner Nodes:</i>	<input type="text" value="1"/>
	<input type="text" value="131"/>

Now, specify the orientation vector for the bar elements.

	X:	Y:	Z:
<i>Base:</i>	<input type="text" value="0"/>	<input type="text" value="0"/>	<input type="text" value="0"/>
<i>Tip:</i>	<input type="text" value="0"/>	<input type="text" value="0"/>	<input type="text" value="1"/>

Use a mesh/copy command to create the rest of the bar elements.

Mesh/Copy/Element...

<input type="text" value="Method ^"/>	<input type="text" value="Type"/>
<Select one of the two L Bar from the pulldown menu>	
<input type="text" value="OK"/>	
<i>Repetitions:</i>	<input type="text" value="4"/>

	X:	Y:	Z:
<i>Base:</i>	<input type="text" value="0"/>	<input type="text" value="0"/>	<input type="text" value="0"/>
<i>Tip:</i>	<input type="text" value="5"/>	<input type="text" value="0"/>	<input type="text" value="0"/>

-
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.

View/Regenerate... (Ctrl-G)

8. Offset the stiffeners to correctly represent the model.

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

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

View/Rotate... (F8)

Dimetric

OK

Modify/Update Elements/Offsets...

Method ^

Type

Type:

2..L Bar

OK

- Update End A
- Set EndB = EndA

OK

	X:	Y:	Z:
Base:	<input type="text" value="0"/>	<input type="text" value="0"/>	<input type="text" value="0"/>
Tip:	<input type="text" value="0"/>	<input type="text" value="0"/>	<input type="text" value="-1.05"/>

OK

NOTE: The offset value is 1/2 the height of the cross section and 1/2 the thickness of the plate.

- 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:

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

Continue

- 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

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... (F6)

Node

Load - Pressure

Constraint

Plot the deformation of the beam.

View/Select... (F5)

Deformed Style:

Deform

Deformed and Contour Data...

Output Vectors/Deformation:

1..Total Translation

OK

OK

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

View/Select... (F5)

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... (F5)

Deformed Style:

None-Model Only

Contour Style:

None-Model Only

OK

This concludes the exercise.

<i>Min Z Disp:</i>	<i>-1.10626</i>
<i>Max Von Mises:</i>	<i>13518.45</i>

