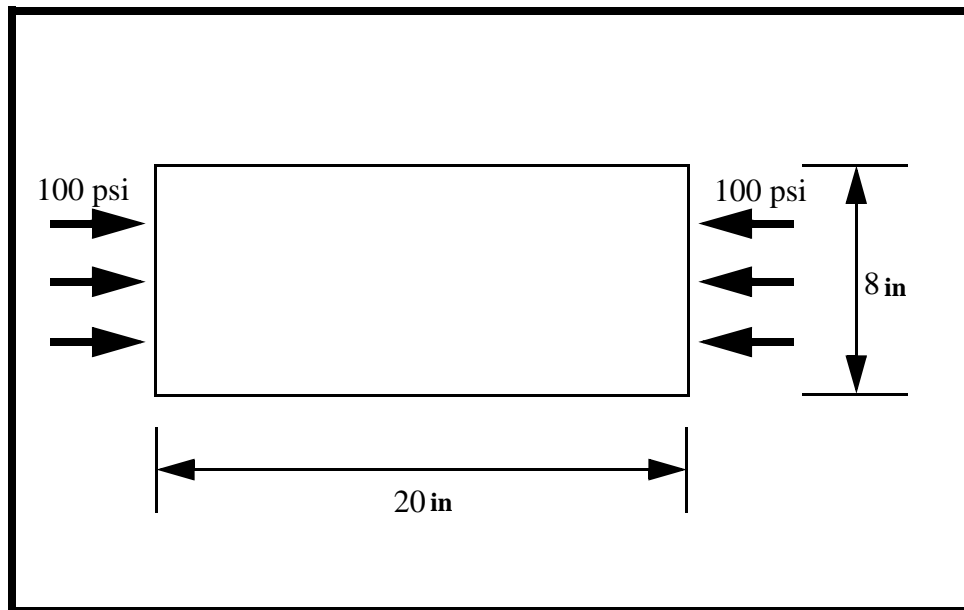

WORKSHOP 5

Elastic Stability of Plates (Plate Buckling Analysis)



Objectives:

- Create a geometric representation of a plate.
- Apply a compression load to two opposite sides of the plate.
- Run a buckling analysis of the plate.

Model Description:

Below is a finite element representation of a rectangular plate under equal, uniform compression on two opposite edges. Assume that all edges are simply supported.

Figure 5.1 - Load Conditions

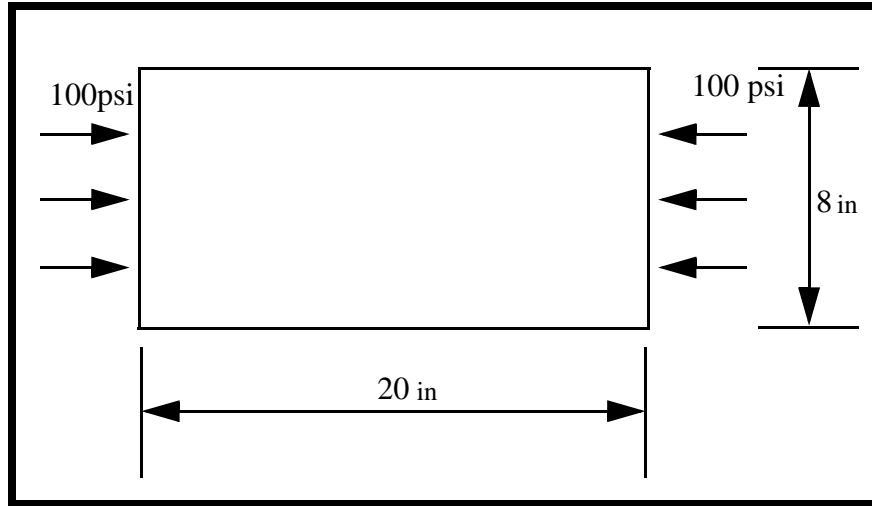


Table 5.1 - Material Properties

Youngs Modulus:	29E+06 psi
Poisson's Ratio	0.3
Plate Thickness:	0.01 in

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:

New Model

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

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

Model/Material...

Title:

mat_1

Youngs Modulus:

29E6

Poisson's Ratio:

0.3

OK

Cancel

3. Create a property called **prop_1** to apply to the members of the plate itself.

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

Model/Property...

Title:

prop_1

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:

0.01

OK

Cancel

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

Make the geometry in standard form.

Tools/Advanced Geometry...

Geometry Engine:

● **Standard**

OK

Geometry/Surface/Corners...

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

OK

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="8"/>	<input type="text" value="0"/>	<input type="text" value="OK"/>
<input type="text" value="0"/>	<input type="text" value="8"/>	<input type="text" value="0"/>	<input type="text" value="OK"/>

Cancel

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

View/Autoscale (Ctrl-A)

- Place mesh seeds on the newly created surface.

Mesh/Mesh Control/Mapped Divisions on Surface...

Select All

OK

	s	t
<i>Number of Elements:</i>	<input type="text" value="10"/>	<input type="text" value="4"/>

Bias:

1.

1.

OK

Cancel

6. Create the appropriate elements on the surface of the plate.

Mesh/Geometry/Surface...

Select All

OK

Property:

1..prop_1

OK

Turn off the workplane.

Tools/Workplane... (F2)

Draw Workplane

Done

View/Regenerate... (Ctrl-G)

7. Create the constraints for the model.

Before creating the appropriate constraints, a constraint set needs to be created. Do so by performing the following:

Model/Constraint/Set...

Title:

constraint_1

OK

Now define the relevant constraint for the model.

Model/Constraint/Nodal...

Select all 5 nodes on the left edge.

HINT: Use

Method ^

On Curve

to easily select the nodes on the left edge.

OK

On the *DOF* box, select all translations.

TX **TY** **TZ**

OK

Now select all 5 nodes on the right edge.

OK

On the *DOF* box, select the following translations.

TY **TZ**

OK

Finally, select the nodes on the top and bottom edges without selecting the corner nodes.

OK

On the *DOF* box, select the following translation.

TZ

OK

Cancel

8. Create the appropriate model loading.

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

Model/Load/Set...

Title:

load_1

OK

Next, convert the edge pressure of 100 psi to appropriate nodal force. Total edge force will be (100 psi) x (0.01 in) x (8 in) = 8 lb. Thus, 2 lb each will be used for the 3 middle nodes and 1 lb each will be used for the 2 corner nodes.

Model/Load/Nodal...

Select the middle 3 nodes of right edge

OK

Highlight **Force**.

FX **Force**
OK **-2**

Now select the top and bottom nodes of right edge

OK

Highlight **Force**.

FX **Force**
OK **-1**
Cancel

This will put a total of 8 lb along the right edge.

9. Create the input file and run the analysis.

File/Export/Analysis Model...

Analysis Format/Type: **7..Buckling**
OK

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

File Name: **plbuck**
Write

Additional Info:

Run Analysis

Advanced...

Modal Solution Method:

Lanczos

*Eigenvalues & Eigenvectors/
Number Desired:*

1

OK

Problem ID:

**Plate Buckling
Sample Problem**

OK

OK

OK

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

Yes

File Name:

plbuck

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

- Look at the results to find the first eigenvalue.

Answer the following question:

What is the first eigenvalue?

Eigenvalue 1 = _____

Since the applied pressure = $8/(8)(.01) = 100$ psi,

$$s_{cr} = 1.722(100) \\ = \mathbf{172.2 \text{ psi}}$$

11. Theory.

From: Formulas for Stress & Strain, Roark & Young, McGraw-Hill

$$\sigma_{cr} = K \frac{E}{1 - \nu} \left(\frac{t}{b} \right)^2$$

Here K depends on ratio a/b.

When $a/b = 20/8 = 2.5$, $K = 3.373$

Thus,

$$\sigma_{cr} = 3.373 \left(\frac{29e6}{1 - (.3)^2} \right) \left(\frac{.01}{8} \right)^2 \\ = \mathbf{167.96 \text{ psi}}$$

This concludes the exercise.

1.722	Eigenvalue 1
-------	--------------