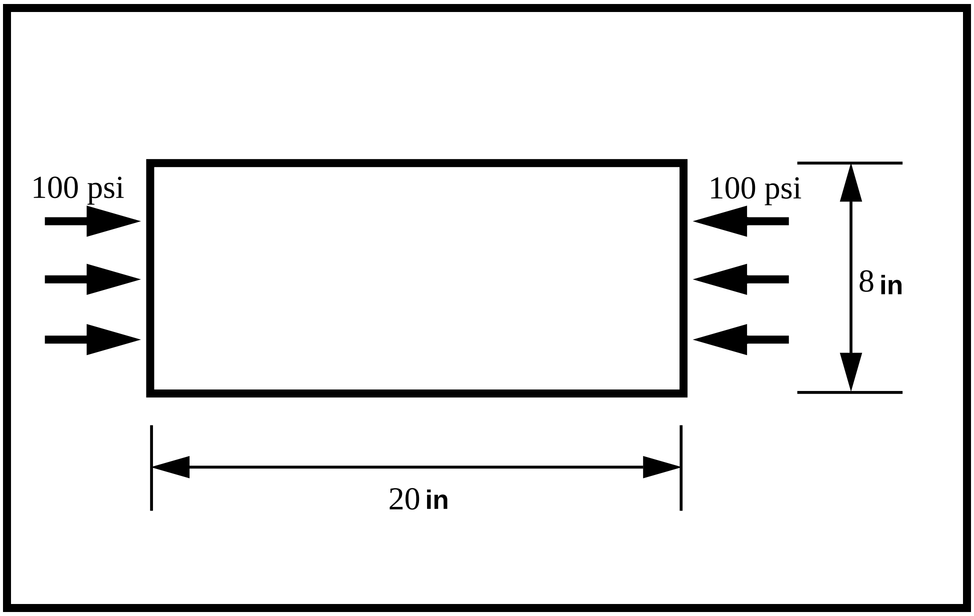
WORKSHOP 13

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 13.1 - Load Conditions

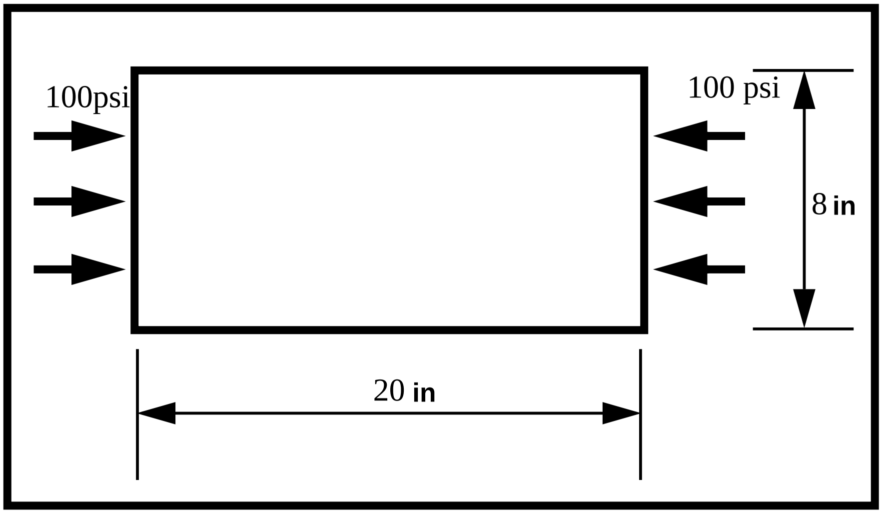


Table 13.1 - Material Properties

Elastic Modulus:	**29E6 psi**
Poisson Ratio	**0.3**
Plate Thickness:	**0.01 in**

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:	**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 NASTRAN geometry for the plate.

Geometry/Surface/Corners...

	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	**8**	**0**	**OK**
0	**8**	**0**	**OK**

Cancel

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

View/Autoscale

5. Place mesh seeds on the newly created surface.

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

Select All

OK

	s	**t**	
Number of Elements:	**10**	**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

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, **Nodes 1, 25, 26, 27 & 28**. (Hint - you may want to use the rectangular pick to box in the desired nodes - shift & left mouse button)

OK

On the *DOF* box, select all translations.

☒ **TX** ☒ **TY** ☒ **TZ**

OK

Now select all 5 nodes on the right edge, **Nodes 11, 12, 13, 14 & 15**.

OK

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

☒ **TY** ☒ **TZ**

OK

Finally, select **Nodes 2** through **10 & 16** through **24** on the top and bottom edges. These nodes do not include the four corners.

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 **Nodes 12, 13, & 14** (the middle 3 nodes of right edge):

OK

Highlight **Force**.

Force

FX ☒ **-2**

OK

Now select **Nodes 11 & 15** (the top and bottom nodes of right edge):

OK

Highlight **Force**.

	Force
FX ☒	**-1**
OK	
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	
	☒ **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 smoothly, we will not bother with the details this time.

Continue

10. 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,

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

11. Theory.

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

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

Here K depends on ratio a/b.

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

Thus,

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

= 167.96 psi

This concludes the exercise.

Eigenvalue 1	**1.722**