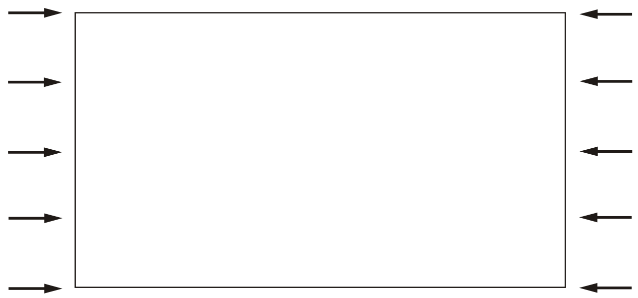
Workshop 9

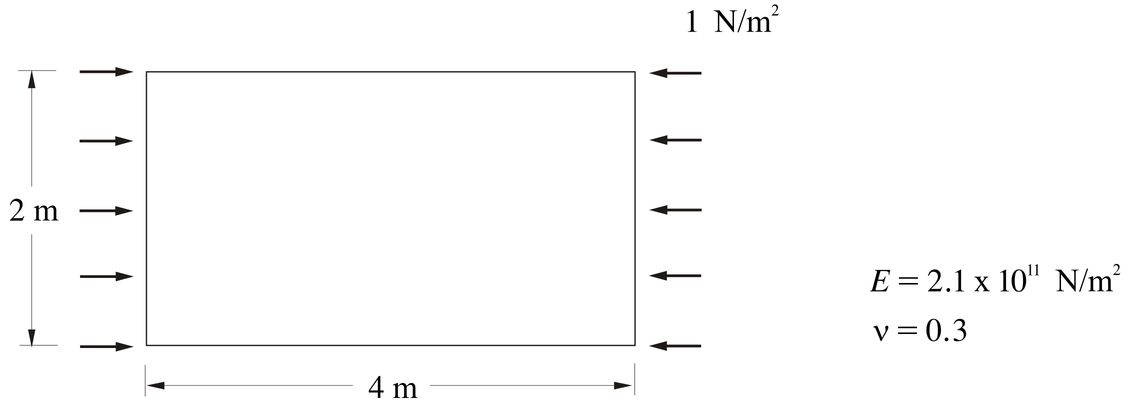
Linear Buckling Analysis of a Plate



Objectives

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

Model Description



Above is a simply supported rectangular plate, of thickness 0.01 m, subjected to a uniform compressive load of magnitude 1 N/m^2 on two opposite edges. In linear buckling analysis we solve for the eigenvalues which are scale factors that multiply the applied load (unit in this case) in order to produce the buckling load.

Exercise Procedure

1. Start up **MSC/NASTRAN for Windows 4.5** and begin to create a new model.

   Double click on the icon for the **MSC/NASTRAN for Windows V4.5**.

   On the *Open Model File* form, select **New Model**.

   Turn off the workplane:

   Tools / Workplane (or **F2**) / ☐ **Draw Workplane / Done**

   View / Regenerate (or **Ctrl G**).

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

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

Title	**mat_1**
Young's Modulus	**2.1e11**
Poisson's Ratio	**0.3**

   Select **OK / Cancel**.

   NOTE: In the *Messages Window* at the bottom of the screen, you should see a verification that the material was created. You can check here throughout the exercise to both verify the completion of operations and to find an explanation for errors which might occur.

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

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

Title	**prop_1**
Material	**mat_1**

   Note that the default element type is **Plate** element, **not parabolic**.

Thickness, Tavg or T_1	**0.01**

   Select **OK / Cancel**.

4. Create the geometry for the plate surface.

   Make the geometry in standard form:

   Tools / Advanced Geometry

Geometry Engine	✓ **Standard**

   Select **OK**.

   Geometry / Curve-Line / Project Points

CSys	**Basic Rectangular**						
First Location	*X*	*0*	*Y*	*0*	*Z*	*0*	**OK**
Second Location	*X*	*0*	*Y*	*2*	*Z*	*0*	**OK**

   Create a second line curve:

First Location	*X*	*4*	*Y*	*0*	*Z*	*0*	**OK**
Second Location	*X*	*4*	*Y*	*2*	*Z*	*0*	**OK**

   Select **Cancel**.

   To fit the display onto the screen, select **View / Autoscale / Visible** (or **Ctrl A**).

   Turn on the curve labels:

   View / Options (or **F6**).

Options	**Curve**
Label Mode	**ID**

   Select **OK**.

   Geometry / Surface / Ruled

From Curve	*1*	To Curve	*2*	**OK**

   Select **Cancel**.

5. Define the mesh size.

   Mesh / Mesh Control / Size Along Curve

   Select curves **1**, **2** and, then, **OK**.

Number of Elements	**8**
Node Spacing	✓ **Equal**

   Select **OK**.

   Select curves **3**, **4** and, then, **OK**.

Number of Elements	**16**
Node Spacing	✓ **Equal**

   Select **OK** / **Cancel**.

6. Generate the finite elements.

   Mesh / Geometry / Surface / Select All / OK

Property	**prop_1**

   Select **OK**.

7. 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_1**

   Select **OK**.

   Model / Constraint / Nodal / Method^/ on Curve

   Select curves **1, 2** and, then, **OK**.

   On the *DOF* box, select

☐	**TX**	☐	**TY**	☑	**TZ**
☑	**RX**	☐	**RY**	☐	**RZ**

Select **OK**.

Method^/ on Curve

Select curves **3, 4** and, then, **OK**.

On the *DOF* box, select

☐	**TX**	☐	**TY**	☑	**TZ**
☐	**RX**	☑	**RY**	☐	**RZ**

Select **OK**.

A warning message will appear: *Selected Constraints Already Exist. OK to Overwrite (No = Combine)?* Select **No** to combine.

Displacement in the X and Y directions and rotation about Z axis must be restraint (just to remove rigid body motion):

Select the node on the bottom left corner / **OK**.

On the *DOF* box, select

☑	**TX**	☑	**TY**

Select **OK**.

A warning message will appear: *Selected Constraints Already Exist. OK to Overwrite (No = Combine)?* Select **No** to combine.

Select the node on the bottom right corner / **OK**.

On the *DOF* box, select

☑ **TY**

Select **OK**.

A warning message will appear: *Selected Constraints Already Exist. OK to Overwrite (No = Combine)?* Select **No** to combine and, then, **Cancel**.

8. Create the loading conditions.

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

Model / Load / Set (or **Ctrl F2**)

Title	**load_1**

Select **OK**.

Since the type of the given load (pressure) is not an available option for the edge of the plate, it must first be converted into nodal forces or distributed along the edge length and, then, applied to the model.

In this model, a 1 N/m^2 pressure force acting over the 0.02 m^2 (2 m $\times$ 0.01 m) can be converted to a total equivalent nodal force of 0.02 N. Since we are going to distribute this force over 2 m of edge length, the force per length will be 0.01 N/m.

Model / Load / On Curve

Select the left edge / **OK**.

Highlight **Force Per Length**

Load	**FX**	✓	**0.01**

Select **OK**.

Select the right edge / **OK**.

Highlight **Force Per Length**

Load	**FX**	✓	**-0.01**

Select **OK / Cancel**.

To visualize nodal forces:

Model / Load / Expand / OK

View / Options (or **F6**)

Category	✓ **Labels, Entities and Color**
Options	**Load Vectors**
Vector Length	**Scale by Magnitude**
Options	**Load-Force**
Label Mode	**Load Value**

Select **OK**.

View / Regenerate (or **Ctrl G**).

Note that the nodes at the corners are loaded half as much as the inner nodes because they are surrounded by half as much area.

9. Run the analysis.

File / Analyze

Analysis Type	**Buckling**
Loads	✓ **load_1**
Constraints	✓ **constraint_1**
Number of Eigenvalues	**1**
	✓ **Run Analysis**

Select **OK**.

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

Be sure to set the desirable working directory.

File Name	**work_9**

Select **Save**.

When the MSC/NASTRAN manager is through running, MSC/NASTRAN for Windows 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. Then select **Continue**.

10. What is the first eigenvalue?

In general, only the lowest buckling load is of interest, since the structure will fail before reaching any of the higher-order buckling loads. Therefore, usually only the lowest eigenvalue needs to be computed. Select

View / Select (or **F5**) **/ Deformed and Contour Data / Output Set**

or

List / Output / Query / Output Set.

11. Display the deformed plot on the screen.

Finally, you may now display the first eigenvector (first buckling mode). You may want to remove the curve, load and boundary constraint markers.

View / Options / Quick Options (or **Ctrl Q**)

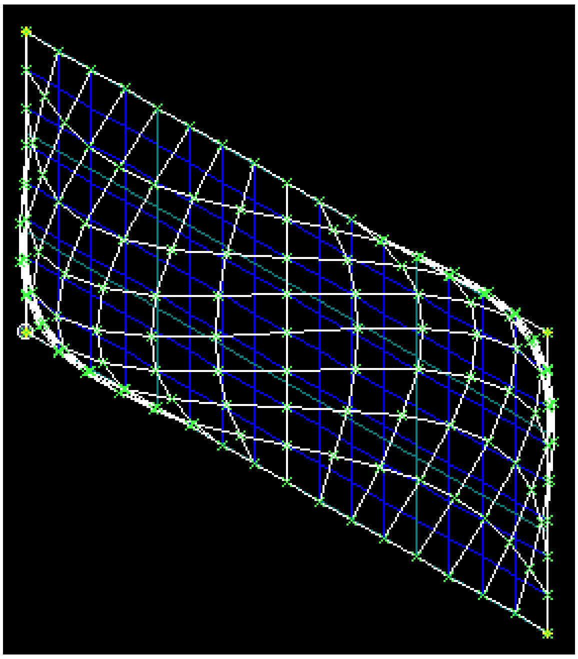
☐ **Curve** / ☐ **Force** / ☐ **Constraint** / **Done** / **OK**

View / Select (or **F5**)

Deformed Style	**Deform**

Select **Deformed and Contour Data / Output Set / OK / OK**.

View / Rotate / Isometric / OK



This concludes the exercise.

File / Save

File / Exit.

Answer

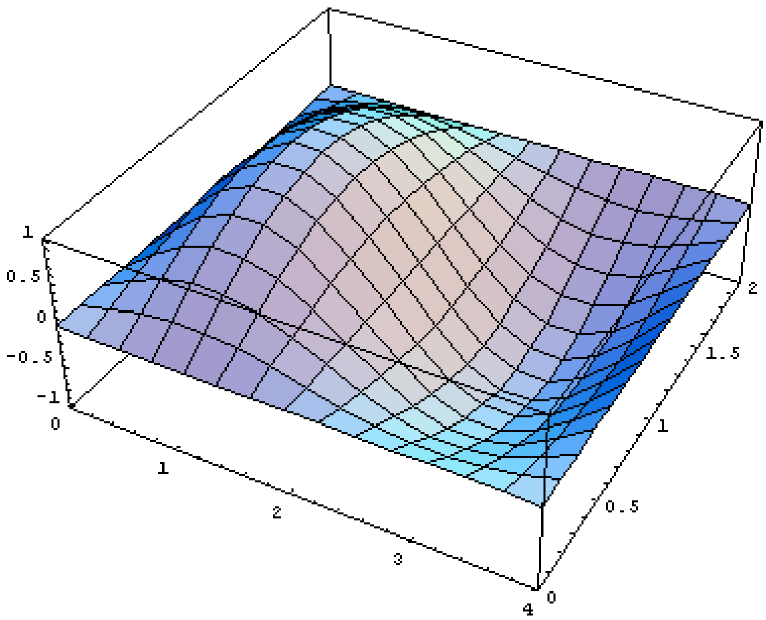
Eigenvalue 1	1.8998×10^7

Based on Kirchhoff plate theory (Reddy, J. N., 1999, *Theory and Analysis of Elastic Plates*, Taylor and Francis, Philadelphia, page 362), the first buckling load is given by

$$\begin{aligned}\sigma_{cr} &= \frac{4\pi^2 D}{b^2 h} = \frac{4\pi^2 E h^2}{12 b^2 (1-\nu^2)} = \frac{4\pi^2\,(2.1\times 10^{11})\,0.01^2}{12(2^2)\,(1-0.3^2)}\\ &= 1.8980 \times 10^7\ \mathrm{N/m^2}\end{aligned}$$

which corresponds to the buckling mode

$$w(x,y) = C \operatorname{sen}\frac{2\pi x}{4} \operatorname{sen}\frac{\pi y}{2}$$



Would you like to improve the result by refining the mesh?