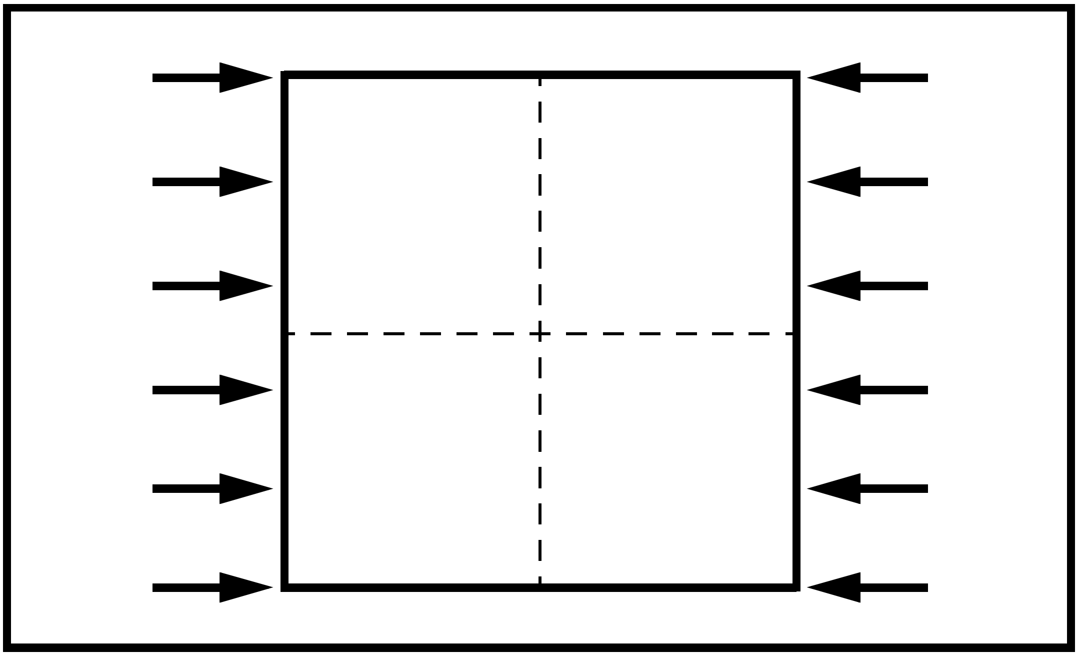
LESSON 18

Linear Bifurcation Buckling Analysis of a Thin Plate



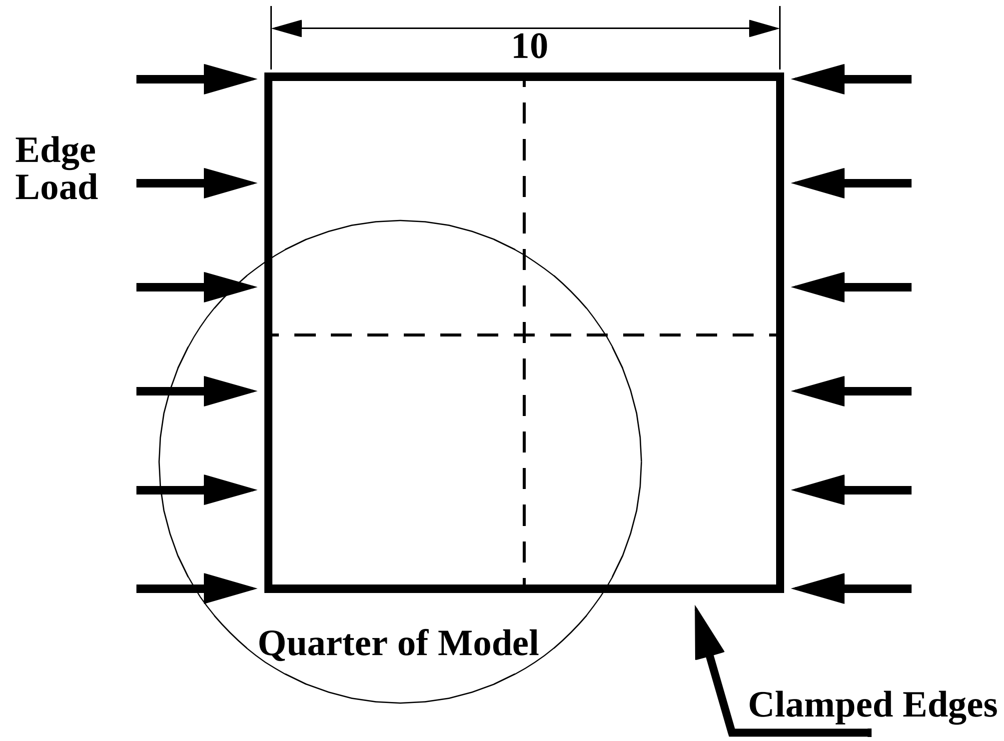
Objectives:

- Construct a quarter model of a simply supported plate.
- Place an edge load on the plate.
- Run a bifurcation buckling analysis.

Model Description:

In this exercise, a thin plate is subjected to a static load on opposing edges. This load exceeds the critical load required to induce buckling. In addition, the axial direction of the beam is slightly offset from the direction of the load. In this exercise you are to run a MSC.Marc nonlinear static analysis on this plate in order to see the effects of buckling.

The model is created using a surface meshed 6x6 with 2D shell elements. The elements are uniformly spaced along the edges of the plate. Due to symmetry, the problem will be analyzed using a quarter model, imposing symmetry boundary conditions.



Thickness = 0.1
Young's Modulus = 10E6
Poisson's Ratio = 0.33

Exercise Procedure:

1. Open a new database. Name it **buckling.db**.

File/New ...

New Database Name:	**buckling.db**

OK

The viewport (PATRAN’s graphics window) will appear along with a *New Model Preference* form. The *New Model Preference* sets all the code specific forms and options inside MSC/PATRAN.

In the *New Model Preference* form set the *Analysis Code* to **MSC.Marc.**

Tolerance:	● **Default**
Analysis Code:	**MSC.Marc**
Analysis Type:	**Structural**

OK

2. Create the model geometry.

◼ **Geometry**

Action:	**Create**
Object:	**Surface**
Method:	**XYZ**
Vector Coordinate List:	**<5, 5, 0>**
Origin Coordinate List:	**[0, 0, 0]**

Apply

3. Create the finite element mesh.

◼ **Elements**

Action:	**Create**
Object:	**Mesh Seed**
Type:	**Uniform**

Number:	**6**

Click in the *Curve List* databox and screen select the left and lower edges of the surface.

Curve List:	select the left & bottom edges

4. Create the model's finite element mesh. On the *Element* form change:

Action:	**Create**
Object:	**Mesh**
Type:	**Surface**
Element Topology:	**Quad4**
Surface List:	select the surface

Apply

5. Now set the material and element properties of the plate. The plate is made of a material with Young's modulus of 10.0E6 lb/in2, with Poisson's ratio of 0.33.

■ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name:	**aluminum**

Input Properties...

Constitutive Model:	**Elastic**
Elastic Modulus:	**10.0E6**
Poisson's Ratio:	**0.33**

OK

Apply

6. Input the properties of the thin plate under **Properties.**

■ **Properties**

Action:	**Create**
Dimension:	**2D**
Type:	**Thin Shell**
Property Set Name:	**plate**
Input Properties...	
Material Name:	**aluminum**
Thickness:	**0.1**
OK	
Select Members:	select the surface
Add	
Apply	

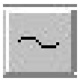
7. Now apply the boundary conditions to the plate.

First, clamp the outer edges of the plate in the z-direction.

■ **Load/BCs**

Action:	**Create**
Object:	**Displacement**
Type:	**Nodal**
New Set Name:	**outer_edges**
Input Data...	
Translations:	**< , , 0 >**
OK	
Select Application Region...	
Geometry Filter:	● **Geometry**
Select Geometric Entities:	select left & bottom edges

In order to do so, select the Curve or Edge icon.



Curve or Edge

Add

OK

Apply

Next, set up the x-symmetry boundary condition of the model

New Set Name:	**x_symmetry**
Input Data...	
Translations:	**< 0, , >**
Rotations:	**< , 0, 0 >**
OK	
Select Application Region...	
Select Geometric Entities:	select the right edge
Add	
OK	
Apply	

Finally, set up the y-symmetry boundary condition

New Set Name:	**y_symmetry**
Input Data...	
Translations:	**< , 0, >**
Rotations:	**<0, , 0 >**
OK	
Select Application Region...	
Select Geometric Entities:	select the top edge
Add	

OK

Apply

8. Next, you will create the edge load on the model.

Action:	**Create**
Object:	**Force**
Type:	**Nodal**
New Set Name:	**fx_1**

Input Data...

Force:	**<208.33, 0 ,0 >**

OK

Select Application Region...

Geometry Filter:	● **FEM**
Select Nodes:	top left and lower left corner nodes

Add

OK

Apply

New Set Name:	**fx_2**

Input Data...

Force:	**<416.67, 0 ,0 >**

OK

Select Application Region...

Geometry Filter:	● **FEM**
Select Nodes:	drag and select all nodes along left edge except corner nodes

Add

OK

Apply

9. Your model is now ready for analysis.

■ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Full Run**
Job Name:	**buckling**
Load Step Creation...	
Job Step Name:	**buckling**
Solution Type:	**Buckling**
Solution Parameters...	
Extraction Method:	**Inverse Power Sweep**
OK	
Apply	
Cancel	
Load Step Selection...	
Selected Job Steps:	**buckling**
OK	
Apply	

The MSC.Marc analysis job **buckling** will then be submitted for analysis to the workstation designated in the Submit Script (usually your local workstation).

When the job is done there will be a results file titled **buckling.t16** in the same directory you started MSC/PATRAN in. You can monitor the progression of the job by looking at ***buckling.log***, ***buckling.sts***, and ***buckling.out*** files. You can also monitor the analysis using the **Analysis** option **Monitor**.

Action:	**Monitor**
Object:	**Job**
View Status File...	

A successful run with end with job exit number **3004**.

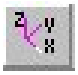
10. When the analysis job is finished, you may read the results back into PATRAN.

■ **Analysis**

Action:	**Read Results**
Object:	**Result Entities**
Method:	**Attach**
Available Jobs:	**buckling**
Select Results File...	
Selected Results File:	**buckling.t16**
OK	
Apply	

11. We will now use MSC/PATRAN to post process the results of the nonlinear static analysis.

First, change the viewing angle of the model using the following toolbar icon:



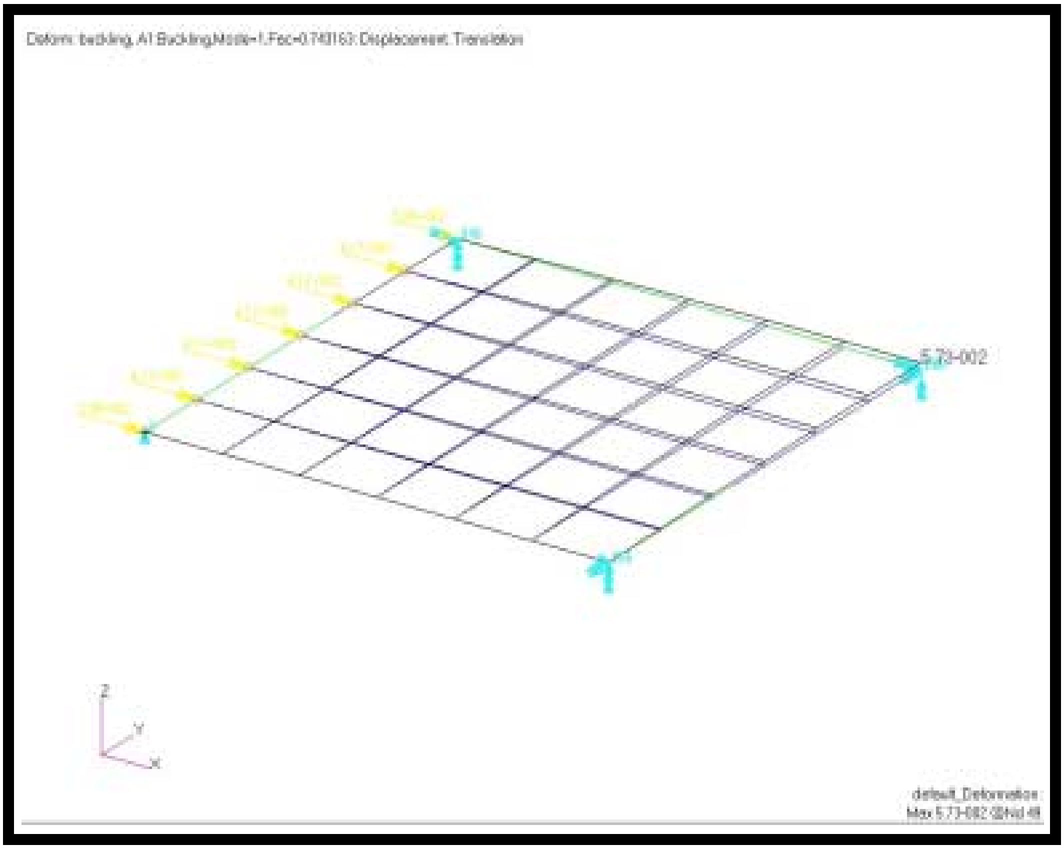
Iso 3 View

■ **Results**

Action:	**Create**
Object:	**Quick Plot**
Select Result Cases:	**buckling, A1..., Mode=1,...**
Select Deformation Result:	**Displacement, Translation**
Apply	

If you look at the result case, you can see that the title contains the first eigenvalue of the model. As a result of buckling, the free corner of the plate “snaps” outward from the original plane, as Figure 18.1 suggests.

Figure 18.1 - This depicts the First Eigenvalue Buckling Mode deformation.



Close the database and quit PATRAN.

This concludes the exercise