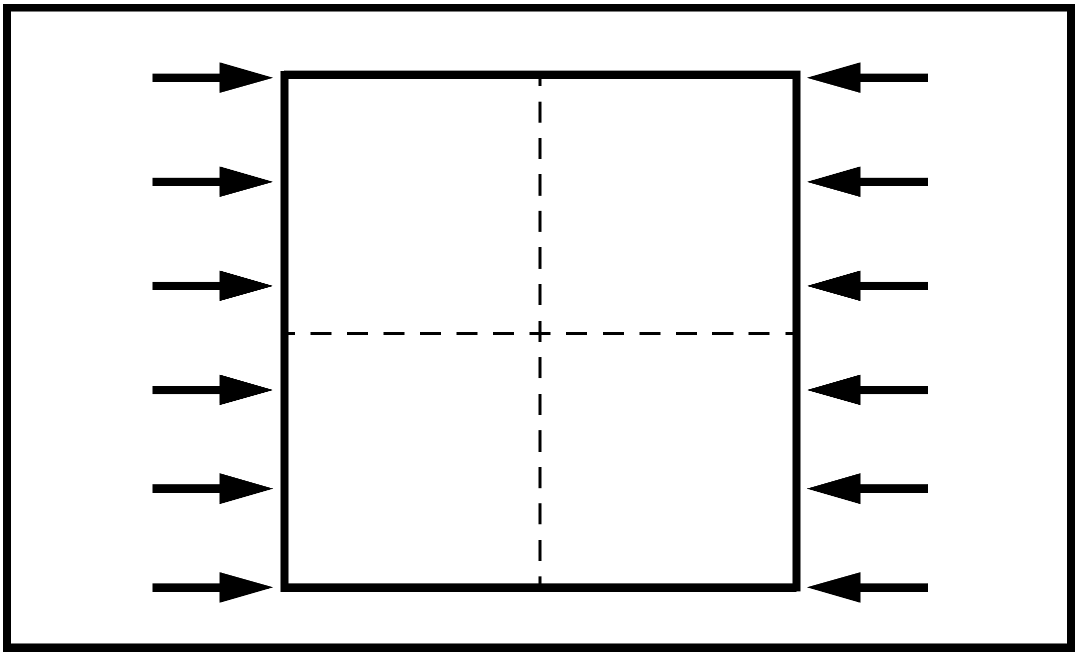
LESSON 19

Post-Buckling Analysis of a Thin Plate



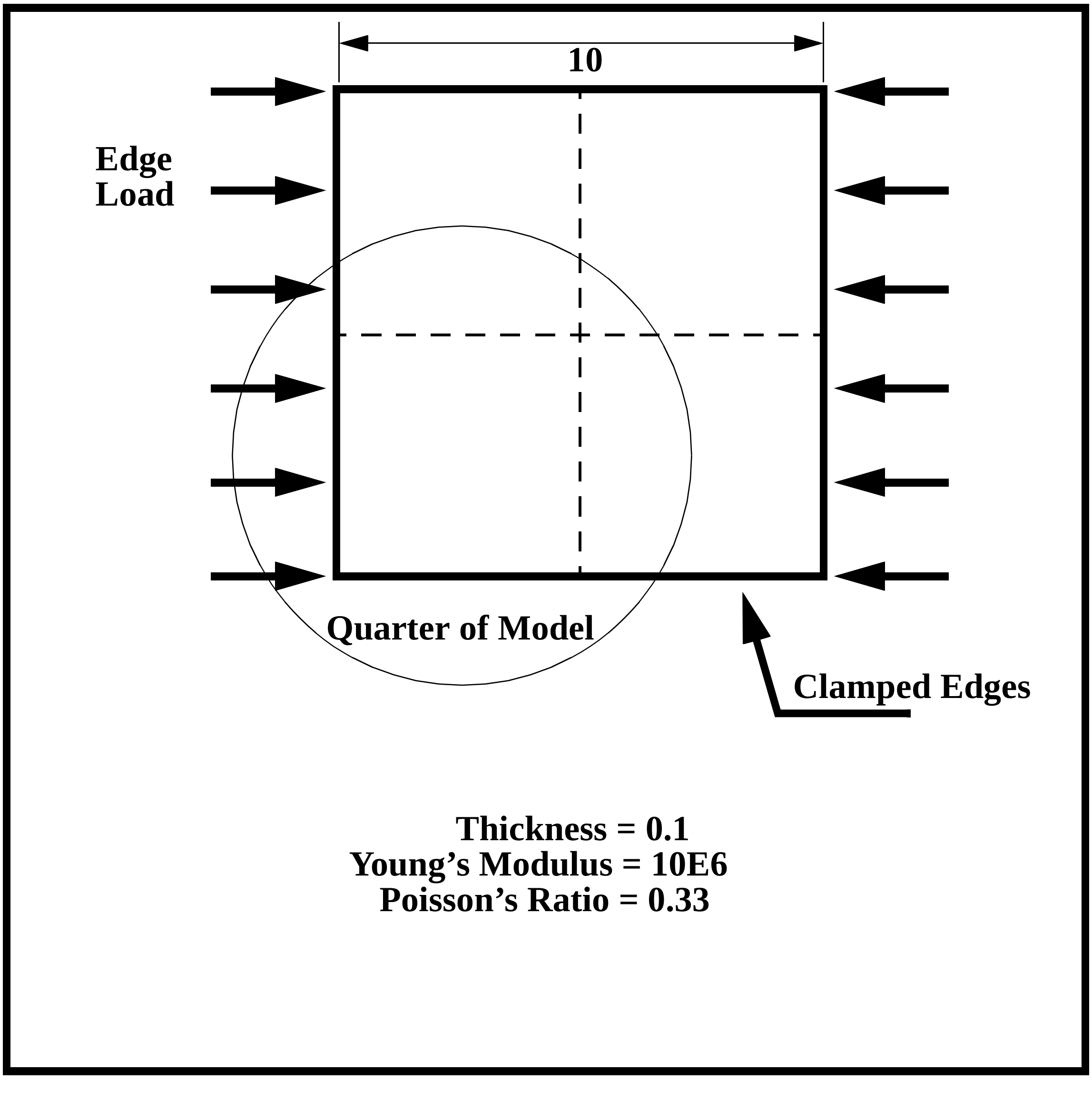
Objectives:

- Construct a thin plate (with slight imperfection).
- Place an axial load on the plate.
- Run a nonlinear static analysis in order to see the behavior of the plate prior to post-buckling.

Model Description:

In this exercise, a thin plate is subjected to a static load. This load exceeds the critical load required to induce buckling. The plate is given a slight imperfection (the top right corner is offset by .001 inches in the z-direction). In this exercise you are to run a MSC.Marc nonlinear static analysis on this thin plate in order to track down the behavior up to post-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.



Exercise Procedure:

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

File/New...

New Database Name:	**post_buckle.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, 0.001>**
Origin Coordinate List:	**[0, 0, 0]**
Apply	

Notice that the z-value in the Vector Coordinate is not zero. This is done to simulate a small imperfection in the geometry, and is usually necessary to do post-buckling analysis.

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 bottom and left curve.

Curve List:	**Surface 1.1 1.4**

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

Action:	**Create**
Object:	**Mesh**
Type:	**Surface**
Element Topology:	**Quad4**
Surface List:	**Surface 1**

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:	**Surface 1**

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...

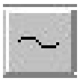
Translations:	**< , , 0 >**

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:	**post_buckle**
Load Step Creation...	
Job Step Name:	**nl_static**
Solution Type:	**Static**
Solution Parameters...	
Load Increment Parameters...	
Maximum Time Step:	**0.1**
Max # of Steps:	**20**
OK	
OK	
Apply	
Cancel	
Load Step Selection...	
Selected Job Steps:	**nl_static**
OK	
Apply	

The MSC.Marc analysis job **post_buckle** 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 **post_buckle.t16** in the same directory you started MSC/PATRAN in.

You can monitor the progression of the job by looking at ***post_buckle.log***, ***post_buckle.sts***, and ***post_buckle.out*** files. You can also monitor the analysis using the **Analysis** option, **Monitor**.

Action:	**Monitor**
Object:	**Job**

View Status File...

A successful job will have completed with: Job ends with 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:	**post_buckle**
Select Results File...	**post_buckle.t16**

Ok

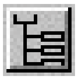
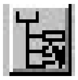
Apply

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

■ **Results**

Action:	**Create**
Object:	**Graph**
Method:	**Y vs X**

Click on the **View Subcases** icon then the **Select Subcases** to bring up the *Select Result Case* form

Select Result Case:	**nl_static**
Filter Method	**All**

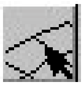
Filter

Apply

Close

Y:	**Result**
Select Y Result:	**Displacement, Translation**
Quantity:	**Z Component**
X:	**Global Variable**
Variable:	**Time**

Select the **Target Entity** icon

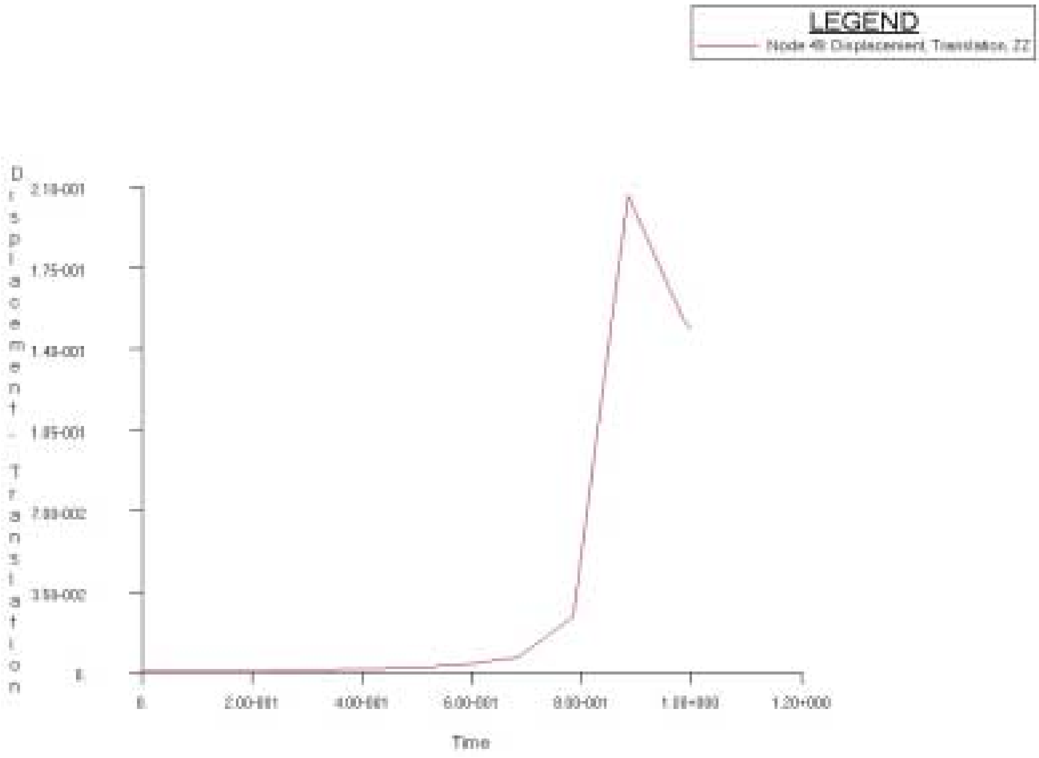


Target Entity:	**Nodes**
Select Nodes	**Node 49** (top right node)

Apply

Notice in Figue 19.1 that as you near the end of the step (when the load has been almost entirely applied), the normal deflection of the plate changes drastically.

Figure 19.1: Graph of the Post-Buckling deformation.



Close the database and quit PATRAN.

This concludes the exercise