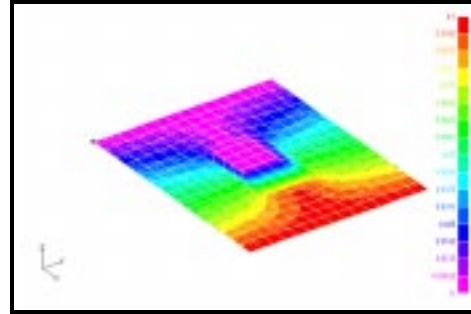


## Shell vs. Bending Pannels



### Objectives

- Build a plate model.
- Create Properties and loads on the plate model.
- Analyze the model for each set of properties and compare the results

### Model Description:

The goal of this example is to compare the differences between the following 2D properties: *Shell* and *Bending Panel*. The model is a simple plate consisting of nine surfaces. We will model the plate using different properties and loading conditions for comparison.

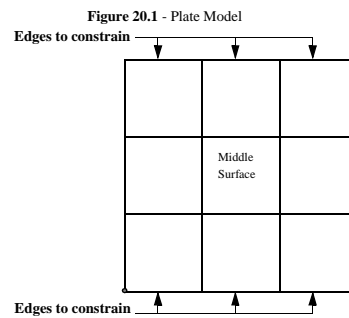


Table 20.1 - Model Properties

Width:	30 in
Length:	30 in
Thickness:	0.2 in
Elastic Modulus:	10E+06 lb/in <sup>2</sup>
Poisson Ratio:	0.3

## Suggested Exercise Steps:

- Build the plate geometry.
- Define materials and properties (shell).
- Mesh and equivalence.
- Apply constraints to top and bottom edges.
- Apply 1 psi pressure to the top of the plate.
- Generate an input file and submit it to the MSC.Nastran solver for analysis.
- Review the results.
- Modify properties (Bending Panel) on the plate model and re-analyze.
- Compare the results.
- Apply a new, inplane load.
- Re-analyze and review the results.

## 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:*

2. Create the 9 surfaces for the plate geometry.

First create one surface

**Geometry/Surface/corners...**

X:                      Y:                      Z:

0	0	0	OK
10	0	0	OK
10	10	0	OK
0	10	0	OK

Click on **View/AutoScale** to see the surface.

**View/AutoScale**                      <Ctrl+A>

Turn off the workplane.

**Tools/Workplane...** <F2>

Draw Workplane

**View/Regenerate...** <Ctrl+G>

Next, copy and paste the surface.

**Geometry/Copy/Surface...**

*Repetitions:*                     

*Base:*                                                                 

*Tip:*                                                                 

Click on **View/AutoScale** to see the 3 surfaces.

**View/AutoScale**                      <Ctrl+A>

Finally, copy and paste the 3 surfaces.

**Geometry/Copy/Surface...**

*Repetitions:*                     

*Base:*                                                                 

*Tip:*                                                                 

Click on **View/AutoScale** to center on the 9 surfaces.

**View/AutoScale**                      <Ctrl+A>

3. Create the material alum.

**Model/Material...**

*Title:*                     

*Youngs Modulus, E:*                     

*Poisson's Ratio, nu:*                     

4. Create the property shell.

**Model/Property...**

*Title:*                     

*Material:*                     

*Thicknesses, Tavg or T1:*                     

5. Define the mesh size, and mesh the surface.

**Mesh/Mesh Control/Default Size...**

*Size:*                     

**Mesh/Geometry/Surface...**

*Property:*                     

6. Constraint the model.

First create the constraint set.

Model/Constraint/Set...

Title:

Constraint

OK

Model/Constraint/On Curve...

Select the top and bottom 3 curves, refer to Figure 20.1

+5  
+18  
+36  
+42  
+48  
+54

OK

● Pinned - No Translation

OK

Cancel

7. Apply a pressure load to the top of the plate.

First create the load set.

Model/Load/Set...

Title:

Pressure

OK

Model/Load/Elemental...

Select All

OK

Pressure

Load/Pressure:

1

OK

Selection Info/Face:

2

OK

Cancel

8. Now, the model is ready for analysis.

File/Export/Analysis Model...

Type:

1..Static

OK

File name:

Shell1

Write

Under Additional Info section:

Run Analysis

OK

When prompt, "OK to Save Model Now" choose Yes

Yes

File name:

Prob20

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 went through without a fatal error, choose *Continue*.

Continue

9. Use **Quick Option** to set the model to display Nodes and Elements only.

Quick Options...

<Ctrl+Q>

All Entities Off

To turn on Elements:

Element

Done

10. Rotate the model for a better view of the results.

View/Rotate...

<F8>

Dimetric

OK

11. To see the translation of the model in Z direction.

View/Select...

<F5>

Deformed Style:

● Deform

Contour Style:

● Contour

Deformed and Contour Data...

Output Set:

1..MSC/NASTRAN Case 1

Deformation:

4..T3 Translation

Contour:

4..T3 Translation

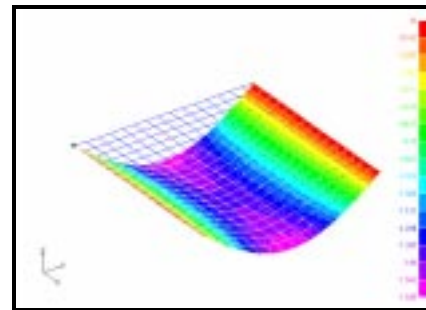
OK

OK

The model should be similar to Figure 20.2.

Notice the symmetry in these two contour plots.

Figure 20.2 - Deformation Plot of the First Model (shell only)



Then do a contour plot of the Xdir Top Stress.

View/Select...

<F5>

Deformed Style:

● Deformed

Contour Style:

● Contour

Deformed and Contour Data...

Output Set:

1..MSC/NASTRAN Case 1

Deformation:

4..T3 Translation

Contour:

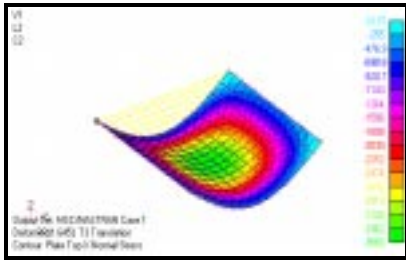
7020..Plate Top X Normal Stress

OK

OK

The model should be similar to Figure 20.3.

Figure 20.3 Contour plot of Xdir Plate Top Stress



12. Turn off result displays.

View/Select... <F5>  
 Deformed Style:  None - Model Only  
 Contour Style:  None - Model Only

13. Now, modify the property of the middle surface.

First create the new property, and this property will only have bending properties (MID2 on MSC/Nastran PSHELL card).

Model/Property...  
 Title:   
 Elem/Property Type...  
 Plane Elements:  Bending Only  
  
 Material:   
 Area, Thicknesses, Tavg or TI:

Now, modify the middle surface.

Modify/Update Elements/Type...

Method^  
 (highlight)  
 select the middle surface

Entry ID:  (highlight)

14. Now, the model is ready for analysis.

File/Export/Analysis Model...

Type:

File name:

Under Additional Info section:

Run Analysis

When prompt, "OK to Save Model Now" choose Yes

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 went through without a fatal error, choose **Continue**.

15. To see the translation of the model in Z direction.

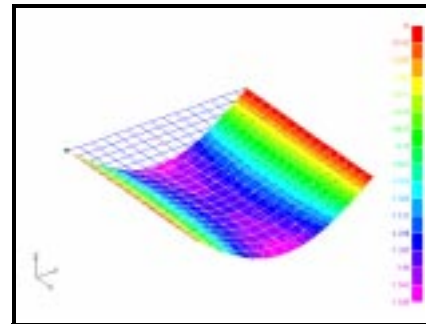
View/Select... <F5>  
 Deformed Style:  Deform  
 Contour Style:  Contour  
  
 Output Set:   
 Deformation:   
 Contour:

The model should be similar to Figure 20.4.

Again, notice the symmetry in these next two contour plots. We have changed the element properties to "bending only," but this has not change the results because the pressure load only creates bending loads.

In the next step, we will change the loading, and the "bending only" property will be obvious.

Figure 20.4 - Deformation Plot of the Second Model

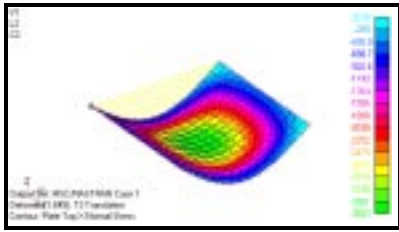


Again, do a contour plot of Xdir Top Plate Stresses.

View/Select... <F5>  
 Deformed Style:  Deformed  
 Contour Style:  Contour  
  
 Output Set:   
 Deformation:   
 Contour:

The model should be similar to Figure 20.5.

Figure 20.5 Xdir Top plate stresses contour plot



16. Turn off result displays.

View/Select... <F5>  
 Deformed Style:  None - Model Only  
 Contour Style:  None - Model Only

17. Now, create new constraint and load sets.

First the load set.

Model/Load/Set...  
 ID:   
 Title:

Model/Load/On Curve...  
 select the 3 right curves

Displacement (highlight)  
 TX:

Now, the constraint set.

Model/Constraint/Set...  
 ID:   
 Title:

Model/Constraint/On Curve...  
 select the 3 left curves

DOF:  Fixed

Model/Constraint/Nodal...  
 Method^  
 (highlight)  
 select the 3 right curves

DOF:  TX

The TX constraint is necessary because MSC.Nastran will displace these nodes (0.1 in x-dir) first, and then fix the nodes in this position. Both the "displacement" load and the constraint are necessary for enforced displacements.

18. Now, the model is ready for analysis.

File/Export/Analysis Model...  
 Type:   
  
 File name:   
  
 Under Additional Info section:  
 Run Analysis  
  
 When prompt, "OK to Save Model Now" choose Yes  
  
 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 went through without a fatal error, choose Continue.

19. To see the translation of the model in X direction.

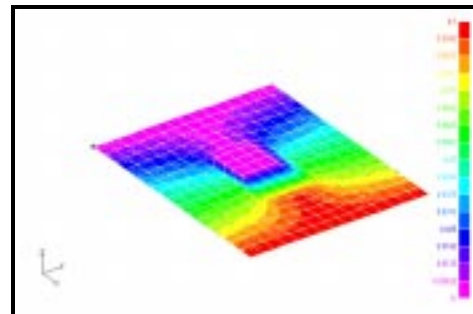
View/Select... <F5>

Deformed Style:  Deform  
 Contour Style:  Contour  
  
 Output Set:   
 Deformation:   
 Contour:

The model should be similar to Figure 20.6.

Notice that the center nodes do not displace. They do not displace because the elements on the center surface are "bending only" elements, and they carry no in-plane load.

Figure 20.6 - Contour plot of Xdir Top Plate stresses



Also, do the Xdir Top Plate stresses.

View/Select... <F5>

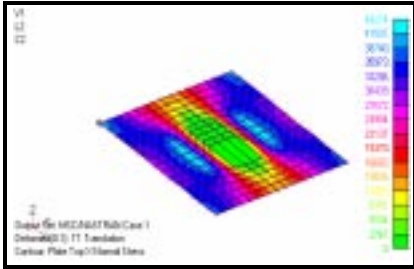
Deformed Style:  Deformed  
 Contour Style:  Contour

Deformed and Contour Data...  
 Output Set: 3..MSC.Nastran Case 1  
 Deformation: 2..T1 Translation  
 Contour: 7020..Plate Top X Normal Stress

OK  
 OK

The model should be similar to Figure 20.7.

Figure 20.7 Xdir Top plate stresses contour plot.



20. Modify the center surface back to the original property type.

Modify/Update Elements/Type...  
 Method^  
 (highlight) on Surface  
 (select the middle surface) +9  
 OK

Entity ID: 1..shell  
(highlight)  
 OK

21. Now, the model is ready for analysis.

File/Export/Analysis Model...  
 Type: 1..Static  
 OK  
 File name: Shell4  
 Write

Under Additional Info section:

Run Analysis

OK

When prompt, "OK to Save Model Now" choose Yes

Yes

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 went through without a fatal error, choose **Continue**.

Continue

22. To view deformation and plots:

View/Select... <F5>  
 Deformed Style:  Deform  
 Contour Style:  Contour  
 Deformed and Contour Data...

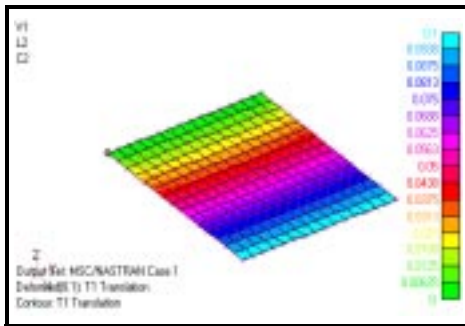
Output Set: 4..MSC/NASTRAN Case 1  
 Deformation: 2..T1 Translation  
 Contour: 2..T1 Translation

OK  
 OK

The model should be similar to Figure 20.8.

Notice the symmetry now for in-plane loading.

Figure 20.8 - Deformation and Stress Plot



This concludes this exercise.

File/Save

File/Exit

FINAL NOTE: When comparing these models, notice how removing the membrane properties affects only the load case with in-plane tension loading. Without the membrane properties, these elements only take bending loads, and when the load is in plane tension, the results are unsymmetrical. The bending (out of plane) load case is symmetric and identical whether or not the membrane property was removed.