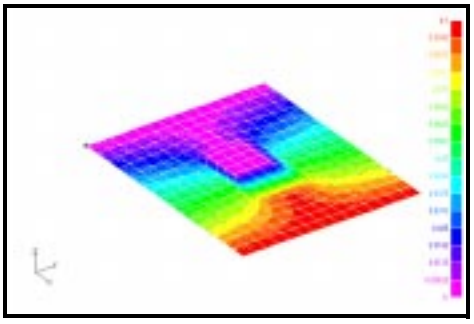
WORKSHOP 20

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.

Figure 20.1 - Plate Model

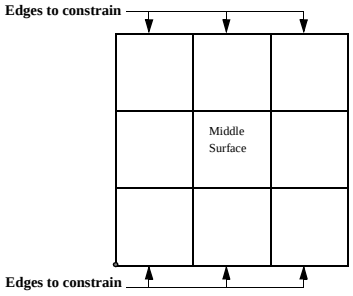


Table 20.1 - Model Properties

Width:	**30 in**
Length:	**30 in**
Thickness:	**0.2 in**
Elastic Modulus:	**10E+06 lb/in^2**
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: **New Model**

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

   Cancel

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

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

   Turn off the workplane.

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

   ☐ **Draw Workplane**

   Done

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

Next, copy and paste the surface.

Geometry/Copy/Surface...

More

OK

Repetitions: **2**

OK

Base:	0	0	0
Tip:	0	10	0

OK

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

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

Finally, copy and paste the 3 surfaces.

Geometry/Copy/Surface...

Select All

OK

Repetitions: **2**

OK

Base:	0	0	0
Tip:	10	0	0

OK

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

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

3. Create the material alum.

   Model/Material...

   Title: **alum**

   Youngs Modulus, E: **10E6**

   Poisson's Ratio, nu: **0.3**

   OK

   Cancel

4. Create the property shell.

   Model/Property...

   Title: **shell**

   Material: **1..alum**

   Thicknesses, Tavg or T1: **0.2**

   OK

   Cancel

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

   Mesh/Mesh Control/Default Size...

   Size: **2**

   OK

   Mesh/Geometry/Surface...

   Select All

   OK

   Property: **1..shell**

   OK

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 Nodes: ☒ **Node**

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**

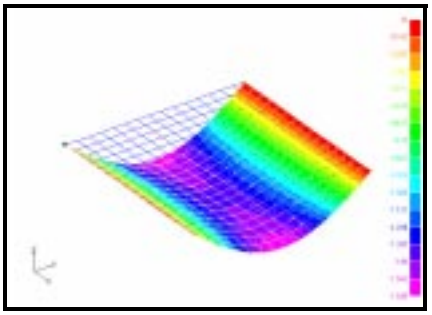
OK

OK

The model should be similar to Figure 20.2.

Notice the symmetry in these two countor 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**

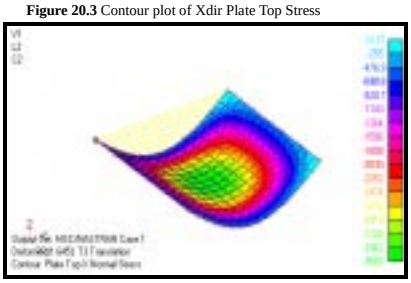
Contour: **7020..Plate Top X Normal Stress**

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**

OK

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: **bending**

Elem/Property Type...

Plane Elements: ● **Bending Only**

OK

Material: **1..alum**

Area, Thicknesses, Tavg or T1: **0.2**

OK

Cancel

Now, modify the middle surface.

Modify/Update Elements/Type...

Method^

on Surface *(highlight)*

select the middle surface

+9

OK

Entity ID: **2..bending** *(highlight)*

OK

14. Now, the model is ready for analysis.

File/Export/Analysis Model...

Type: **1..Static**

OK

File name: **Shell2**

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

15. 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: **2..MSC/NASTRAN Case 1**

Deformation: **4..T3 Translation**

Contour: **4..T3 Translation**

OK

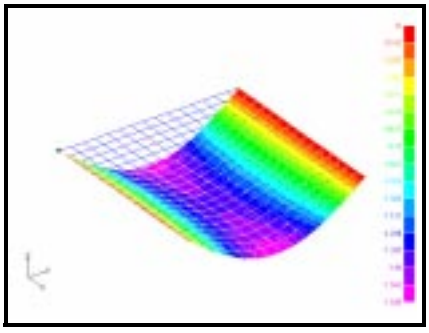
OK

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**

Deformed and Contour Data...

Output Set: **2..MSC/NASTRAN Case 1**

Deformation: **4..T3 Translation**

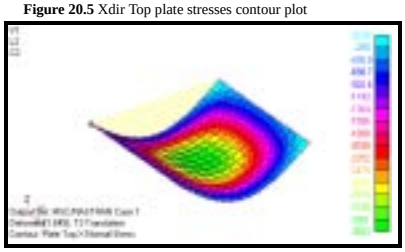
Contour: **7020..Plate Top X Normal Stress**

OK

OK

The model should be similar to Figure 20.5.

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**

OK

17. Now, create new constraint and load sets.

First the load set.

Model/Load Set...

ID:	**2**
Title:	**Displacement**

OK

Model/Load/On Curve...

select the 3 right curves

+47
+51
+55

OK

Displacement *(highlight)*

TX:	**0.1**

OK

Cancel

Now, the constraint set.

Model/Constraint/Set...

ID:	**2**
Title:	**Constraint2**

OK

Model/Constraint/On Curve...

select the 3 left curves

+8
+13
+17

OK

DOF: ● **Fixed**

OK

Cancel

Model/Constraint/Nodal...

Method^

on Curve *(highlight)*

select the 3 right curves

+47
+51
+55

OK

DOF: ☒ **TX**

OK

Cancel

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:	**1..Static**

OK

File name:	**Shell3**

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

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

View/Select...	**<F5>**

Deformed Style:	● **Deform**
Contour Style:	● **Contour**

Deformed and Contour Data...

Output Set:	**3..MSC/NASTRAN Case 1**
Deformation:	**2..T1 Translation**
Contour:	**2..T1 Translation**

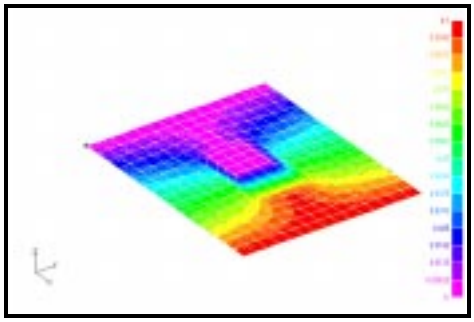
OK

OK

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: **4..T3 Translation**

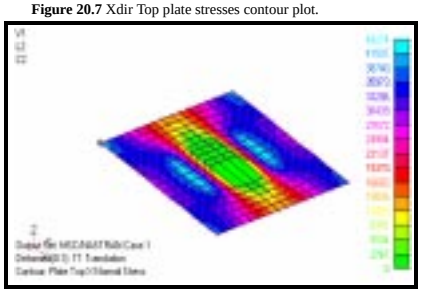
Contour: **7020..Plate Top X Normal Stress**

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**

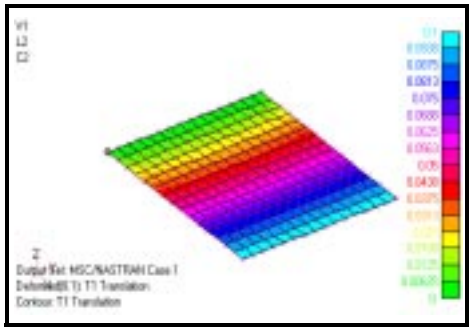
OK

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.