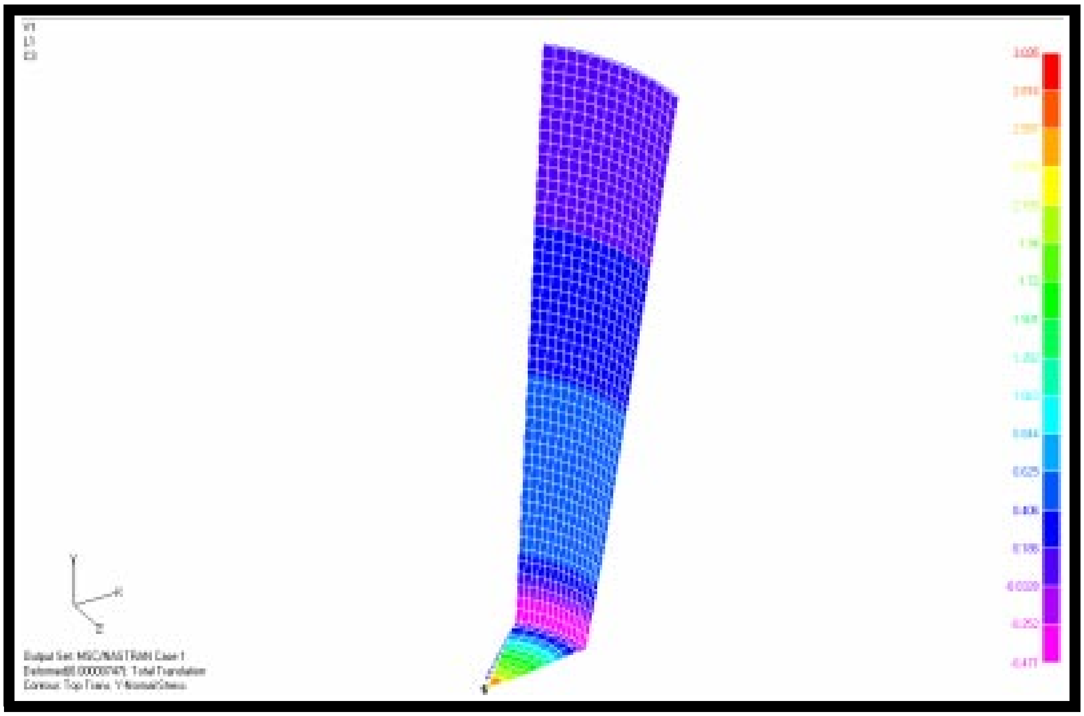
WORKSHOP 13a

Results Post-Processing -- Coffee Cup



Objectives:

- Manually create a geometry of the coffee cup using the given dimensions.
- Input the hydrostatic loading conditions by creating an equation.
- Submit the job for analysis.
- Review the results from the analysis.

Model Description:

The coffee cup is a "real-life" problem which requires using a function to define the fluid pressure load.

In this exercise, we will model a coffee cup. The load from the liquid inside the cup will be applied as hydrostatic pressure on the inside surfaces of the cup. We will only model 1/8th of the cup, since the geometry and the loading is symmetric. This will decrease the file and model size, which in turn will decrease the analysis run time. We will apply a symmetry boundary conditions along the outer edges of the cup to simulate the entire geometry. In addition, we will apply constraint z-direction on the bottom lip of the cup (where it touches the table). After meshing with 2D elements on the geometry, we will then apply the equation which defines the fluid pressure load on the cup. We will specify our own cylindrical coordinate system from which the geometry, meshing, load and constraints and results will be read. The model's dimensions are shown below.

Figure 13a.1 - Coffee Cup

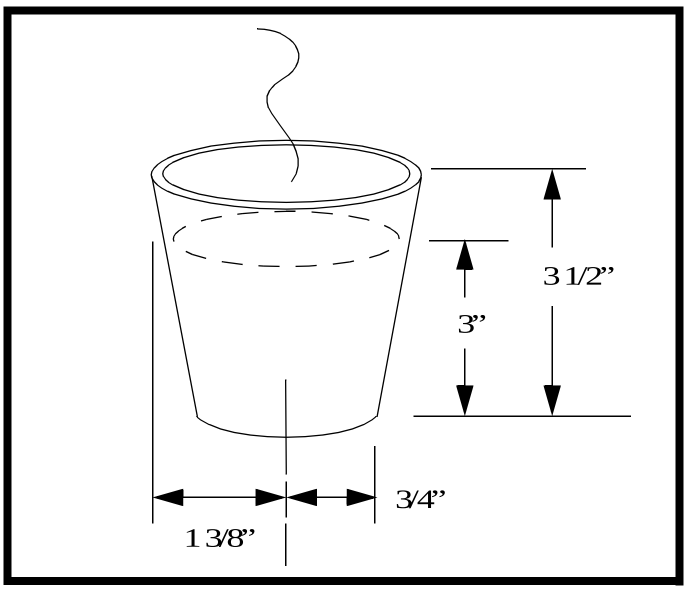


Table 13a.1 - Material Properties

Thickness:	**1/8 in**
Youngs Modulus:	**4.7E+05 psi**
Poisson's Ratio:	**0.333**

Suggested Exercise Steps:

- Create the material properties.
- Create the plate element properties.
- Create a cylindrical coordinate system.
- Create the curves that define the midplane of the cup.
- Revolve the curves 45 degrees to create the surfaces.
- Specify mesh size on the surfaces.
- Mesh the surfaces.
- Verify element orientation.
- Apply constraints and pressure equation.
- Update the Output Coordinate System to the user defined cylindrical coordinate system.
- Submit for analysis.
- Update the material angle for the elements to align with the user define cylindrical coordinate system.
- Transform the results to the user defined coordinate system.
- Plot the results.
- Look at .DAT file. Notice the PLOAD4 cards. The pressure is evaluated at the element centroids, hence one value per element.
- Return to Nastran for Windows and edit the pressure load to be evaluated at the corners.
- Look at new .DAT file. Notice the PLOAD4 cards. The pressure is evaluated at the element corners, hence four values per element. The element centroid value is averaged over the four corner nodes.
- Plot the results.

Exercise Procedure:

1. Start up MSC.Nastran for Windows 4.0 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 a material called **mat_1**.

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

Model/Material...

Title:	**mat_1**
Youngs Modulus:	**4.7E5**
Poisson's Ratio:	**0.333**

OK

Cancel

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

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

Model/Property...

Title:	**prop_1**
Material:	**1..mat_1**
Thickness:	**0.125**

OK

Cancel

4. Create the cup's geometry.

Tools/Advanced Geometry...

Geometry Engine: ● **Standard**

OK

First, create a cylindrical coordinate system.

Model/Coord Sys...

ID:	**99**
Title:	**Coord 99**
Method:	● **XY Axes**
Type:	● **Cylindrical**

OK

Define Coordinate System Origin.

X:	*Y:*	*Z:*	
0	**0**	**0**	**OK**

Define Vector along CSys X-Axis.

	X:	*Y:*	*Z:*	
Base:	**0**	**0**	**0**	
Tip:	**1**	**0**	**0**	**OK**

Define Vector in CSys XY-Plane.

	X:	*Y:*	*Z:*	
Base:	**0**	**0**	**0**	
Tip:	**0**	**0**	**-1**	**OK**

Cancel

5. View labels of the curves by:

View/Options **<F6>**

Quick Options... **<Ctrl+Q>**

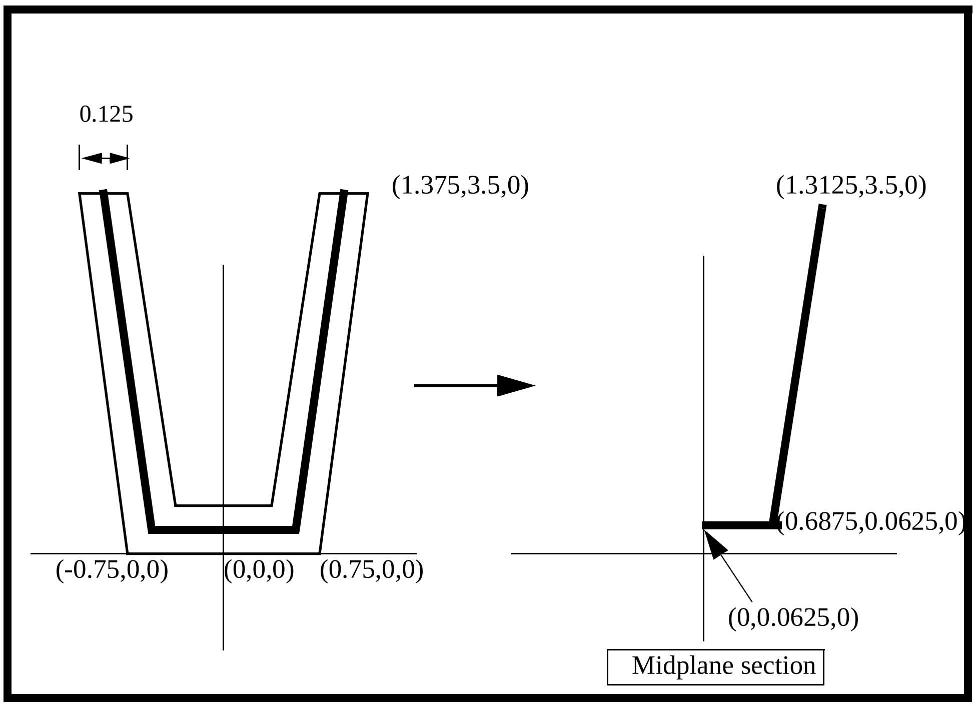
Labels On

Done

OK

6. We wish to create a midplane section to analyze the coffee cup. Figure 13a.2 illustrates the midplane section of the cup and its location in the default rectangular coordinate system.

Figure 13a.2



We begin by creating curves that define the midplane section of the cup.

Geometry/Curve-Line/Coordinates...

Enter First Location for Line of first curve. (**Curve 1**)

X:	*Y:*	*Z:*	
0	**0.0625**	**0**	**OK**

Enter Second Location for Line.

X:	*Y:*	*Z:*	
0.6875	**0.0625**	**0**	**OK**

Enter First Location for Line of second curve. (**Curve 2**)

X:	*Y:*	*Z:*	
0.6875	**0.0625**	**0**	**OK**

Enter Second Location for Line.

X:	*Y:*	*Z:*	
1.3125	**3.5**	**0**	**OK**

Enter First Location for Line of third curve. (**Curve 3**)

X:	*Y:*	*Z:*	
-3	**3**	**0**	**OK**

Enter Second Location for Line.

X:	*Y:*	*Z:*	
3	**3**	**0**	**OK**

Cancel

7. Turn off the workplane.

Tools/Workplane...

☐ **Draw Workplane**

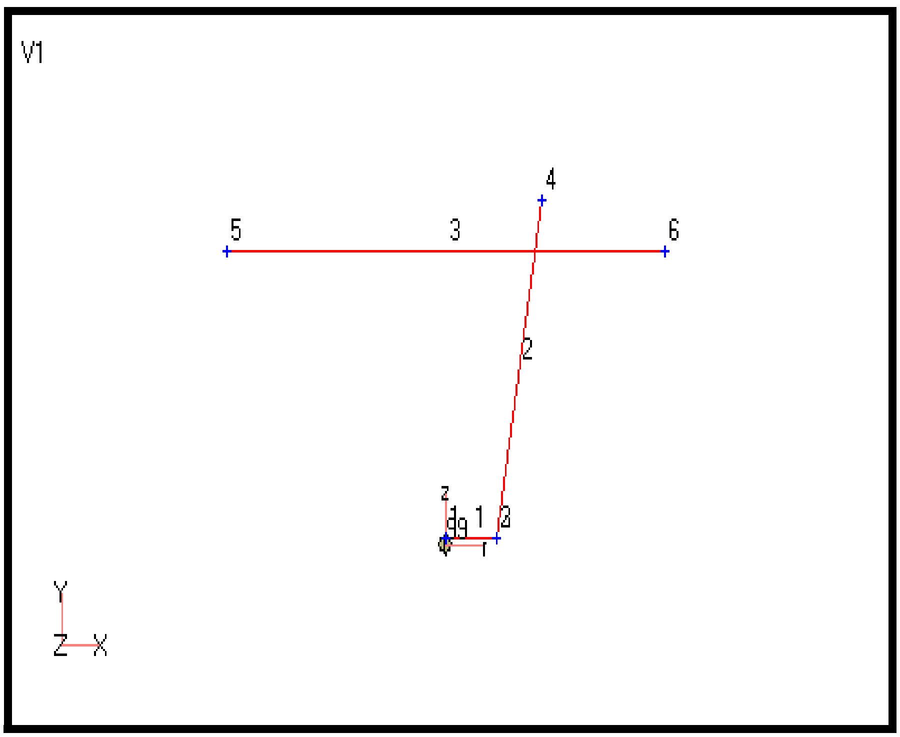
Done

View/Autoscale <Ctrl+A>

8. Break the vertical curve with the newly constructed line.

Refer to the following picture for locations of curves and node points.

Figure 13a.3



Modify/Break...

Select Curve(s) to Break: <Select **Curve 2**>

OK

Methods^

	Intersect - Curves
Curve ID 1:	**3**
Curve ID 2:	**2**

OK

Cancel

9. Delete the line used for the break.

Delete/Geometry/Curve...

<Select **Curve 3**>

OK

When asked if it is OK to Delete 1 Selected Curve(s), Click **Yes**.

Yes

10. Delete the points remaining from the deleted line.

Delete/Geometry/Point...

<Select **Points 5** and **6**>

OK

When asked if it is OK to Delete 2 Selected Point(s), Click **Yes**.

Yes

11. Create the surface of the cup.

NOTE: Only a section of the cup will be modeled. The solution will use symmetry for the overall analysis.

Geometry/Surface/Revolve...

Select All

OK

Select Axis of Rotation.

CSys:	**99..Coord 99**

	R:	*T:*	*Z:*
Base:	**0**	**0**	**0**
Tip:	**0**	**0**	**1**

OK

Rotation Angle: **45**

OK

Cancel

12. Fit the model to display.

    Use **Rotate** to better view model.

    View/Rotate... <F8>

    Trimetric

    OK

13. Turn off labels.

    View/Options... <F6>

    Quick Options...

    Labels Off

    Done

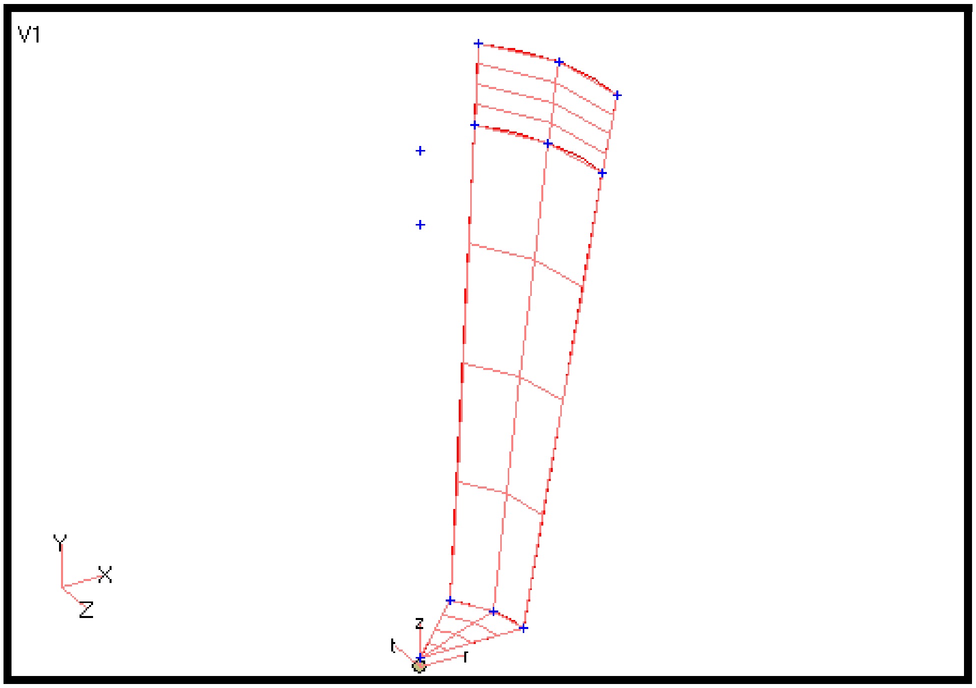
    OK

    Refit the model to the screen

    View/Autoscale... <Ctrl+A>

    The viewport should appear as follows:

Figure 13a.4



14. Now define the mesh size on the cup section.

Mesh/Mesh Control/Size on Surface...

Select All

OK

Element Size: **0.075**

OK

Cancel

To avoid boundary mesh on Surfaces 2 & 3, we will put additional mesh control on Surface 2.

Mesh/Mesh Control/Mapped Divisions on Surface...

<Select **Surface 2**>

OK

	s	t	
Number of Elements:	**14**	**41**	
Bias:	**1**	**1**	**OK**

Cancel

15. Finally, create the finite element entities.

Mesh/Geometry/Surface...

Select All

OK

When asked if it is OK to create a Boundary Mesh, click **Yes**.

Yes

CSys:	**99..Coord 99**
Property:	**1..prop_1**

Node Param...

Output Coordinate System:	**99..Coord 99**

OK

Quick Options... <Ctrl+Q>

☐ **Node**

Done

16. Verify the normal vectors to each of the loading surfaces.

View/Options... <F6>

Options:	**Element - Directions**
Normal Style:	**1..Normal Vectors**
Show Direction	☒

OK

17. Reverse the direction of the normal vectors. They should be pointing outwards. Check for coincident nodes.

First, you have to merge coincident nodes.

Tools/Check/Coincident Nodes...

Select All

OK

When asked if it is OK to Specify Additional Range of Nodes to Merge, click No.

No

Options: ☒ **Merge Coincident Entities**

OK

The status window should reveal 30 Node(s) Merged.

Modify/Update Elements/Reverse...

Select All

OK

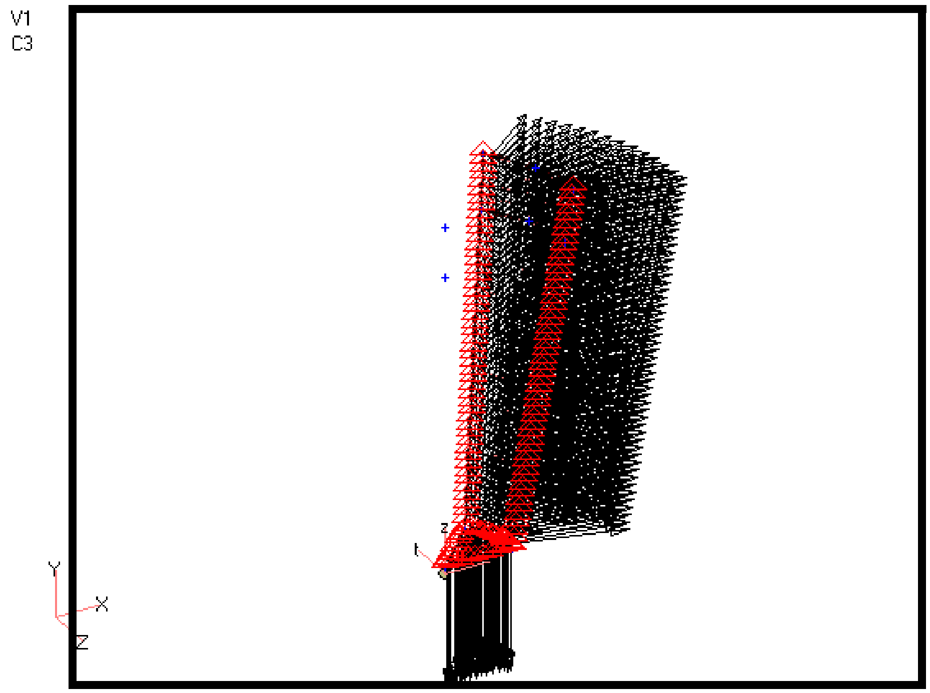
● **All Normals Outward**

OK

View/Autoscale <Ctrl+A>

The viewport should appear as follows:

Figure 13a.5



View/Options... <F6>

Options:	**Element - Directions**
Show Direction	☐

OK

18. Create the constraints.

Model/Constraint/Set...

ID:	**1**
Title:	**bottom_corner**

OK

Now define the relevant constraint for the bottom corner of the cup.

Model/Constraint/Nodal...

Methods^	
	On Curve
< Select **Curve 6** >	(see Figure 13a.6 for curve 6)
OK	
Coord Sys:	**99..Coord 99**
DOF (click to select):	☐ **TR** ☐ **TT** ☒ **TZ**
	☐ **RR** ☐ **RT** ☐ **RZ**
OK	
Cancel	

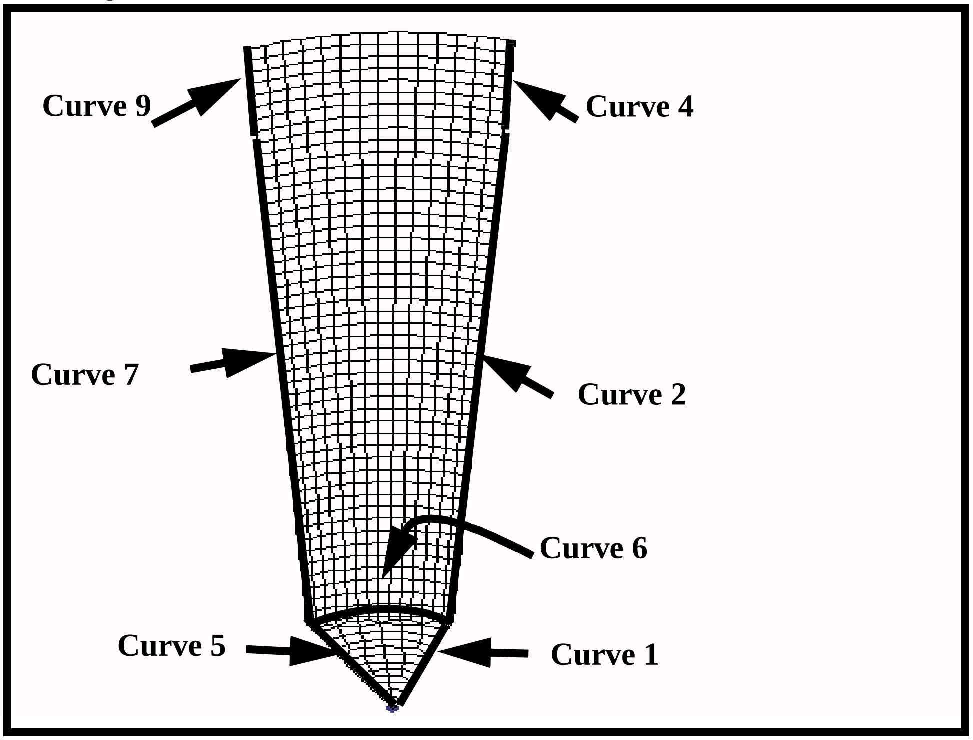
Define the relevant constraint for the sides of the cup.

Model/Constraint/Set...

ID:	**2**
Title:	**sides**
OK	

Refer to the following picture for the curve locations.

Figure 13a.6



Model/Constraint/Nodal...

Method^	
	on Curve
<Select **curves 1, 2, 4, 5, 7 and 9** as shown in **Fig. 13a.6** >	
OK	
Coord Sys:	**99..Coord 99**
Y Symmetry	
OK	
Cancel	

Combine all of the constraints.

NOTE: We only created two constraint sets in the exercise to demonstrate the next feature of combining constraint sets. Creating the second constraint set was not necessary since we are going to run the analysis with both constraint sets.

Model/Constraint/Combine...

From Set: **1..bottom_corner**

More

From Set: **2..sides**

Last One

NOTE: Now, there is a new constraint set (#3) that we will use in the analysis.

19. Create the loading condition.

Model/Load/Set...

Title: **pressure**

OK

Now define the pressure on the relevant surfaces. The pressure that will be created will be applied to the center of each element. Later in the exercise, you will create a pressure applied over the entire element.

Model/Load/On Surface...

<Select **Surfaces 1** and **2**>

OK

Create Loads on Surfaces:

(*highlight*) **Pressure**

Coord Sys: **99..Coord 99**

Method: ● **Variable**

Advanced...

Multiply By: ● **Equation**

Equation: **0.0362*(3-!z)**

OK

Pressure/Value: **-1**

OK

Cancel

20. Transfer the pressure load to finite element to see the pressure contour

Model/Load/Expand...

OK

View/Options... <F6>

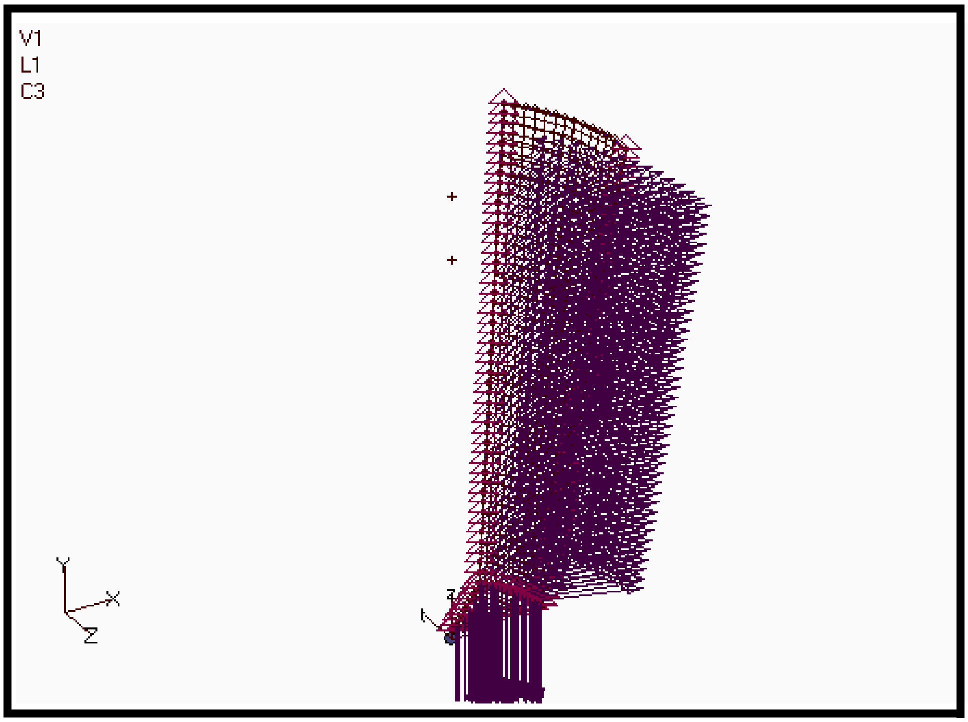
Options: **Load Vectors**

Vector Length: **1..Scale by Magnitude**

OK

The viewport should appear as follows:

Figure 13a.7



21. Now run the model.

File/Export/Analysis Model...

Type: **1..Static**

OK

Change the directory to **C:\temp**.

File name: **coffee_cup**

Write

☒ **Run Analysis**

OK

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

Yes

File name: **coffee_cup**

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 ran successfully, we will not bother with the details this time.

Continue

22. Now, we will analyze the results. The following steps involve updating the material angle and transforming the output to the cylindrical coordinate system.

Set deform style to actual value.

View/Options... <F6>

Category: ● **Post Processing**

Options: **Deformed Style**

Scale %: **1**

Scale Act: **1**

OK

Modify the material angle for each element to Coord 99.

Modify/Update Elements/Material Angle...

Select All

OK

● **Coordinate Direction**

CSys: **99..Coord 99**

● **R**

OK

Model/Output/Transform...

Transform Type: ● **Plate Forces, Stresses and Strains**

From Output Set: **1..MSC.Nastran Case 1**

OK

Select All

OK

NOTE: May not get all of the comments below.

Transform Tria3 Stress

Current Output Orientation: ● **Element First Edge**

OK

Transform Quad4 Stress

Current Output Orientation: ● **Element Diagonal Bisector**

OK

Transform Tria3 Force

Current Output Orientation: ● **Element First Edge**

OK

Transform Quad4 Force

Current Output Orientation: ● **Element Diagonal Bisector**

OK

Quick Options... **<Ctrl+Q>**

All Entities Off

☒ **Element**

Done

View/Select... <F5>

Deformed Style: ● **Deform**

Contour Style: ● **Contour**

Deformed and Contour Data...

Output Set: **1..MSC.Nastran Case 1**

Output Vectors/Deformation: **1..Total Translation**

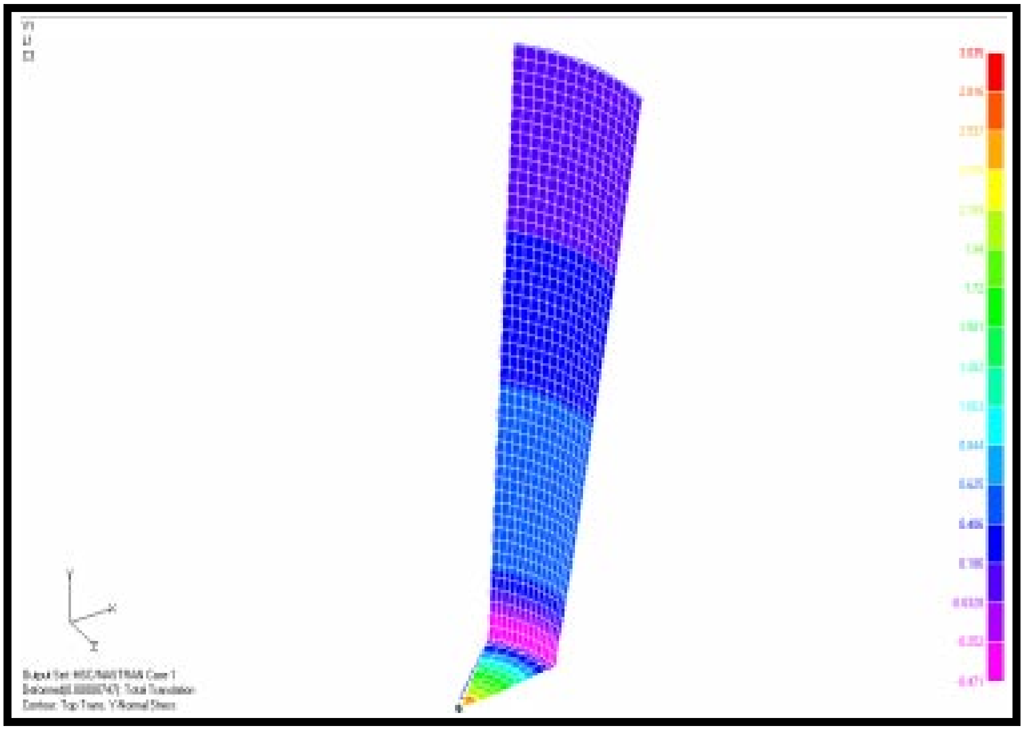
Output Vectors/Contour: **300002..Top Trans.Y-Normal StressB**

OK

OK

The plot should appear as follows:

Figure 13a.8



23. Open a text editor and view the .DAT file. Examine the pressure load that you created. Notice the PLOAD4 cards. The pressure is evaluated at the element centroids, hence one value per element

    The file should appear similar to the excerpts shown on the next page.

coffee_cup.dat

```
$ MSC/NASTRAN for Windows Load Set 1 : pressure
PLOAD4         1       1  0.1096
PLOAD4         1       2 0.10939
PLOAD4         1       3 0.11086
PLOAD4         1       4 0.10992
            .
            .
PLOAD4         1     643  0.1248
PLOAD4         1     644 0.12585
$ MSC/NASTRAN for Windows Constraint Set 1 : bottom_corner
SPC            1       2       3      0.
SPC            1       5       3      0.
SPC            1       6       3      0.
SPC            1       7       3      0.
            .
            .
$ MSC/NASTRAN for Windows Constraint Set 2 : sides
SPC            2      15     246      0.
SPC            2      16     246      0.
SPC            2      17     246      0.
SPC            2      18     246      0.
SPC            2      19     246      0.
            .
            .
$ MSC/NASTRAN for Windows Constraint Set 3 : Combined Set
SPC            3       2       3      0.
SPC            3       5       3      0.
SPC            3       6       3      0.
SPC            3       7       3      0.
SPC            3       8       3      0.
SPC            3       9       3      0.
SPC            3      14       3      0.
SPC            3      15    2346      0.
SPC            3      16     246      0.
            .
            .
$ MSC/NASTRAN for Windows Property 1 : prop_1
PSHELL         1       1   0.125       1               1              0.
$ MSC/NASTRAN for Windows Material 1 : mat_1
MAT1           1 470000.           0.333      0.      0.      0.
GRID           2      99  0.6875 3.21429  0.0625      99
GRID           5      99  0.6875 12.8571  0.0625      99
GRID           6      99  0.6875 16.0714  0.0625      99
GRID           7      99  0.6875 19.2857  0.0625      99
            .
CQUAD4       741       1     808     807      87     823
CQUAD4       742       1     809     808     823     824
ENDDATA ac660e1a
```

24. Return to NASTRAN for Windows and change the type of surface pressure. Instead of having varying step pressure, we will now assign continuous varying pressure onto the corners of each element

File/Save As...

File Name: **coffee_cup2**

Save

Model/Load/Expand

Operation: ● **Compress**

OK

Modify/Edit/Load...

☒ *Select All*

Defined On: ● **Surface**

Elemental Loads: ● **Pressures**

OK

Surface 1

☒ *At Corners*

OK

Surface 2

☒ *At Corners*

OK

Re-analyze the model.

File/Export/Analysis Model...

Type: **1..Static**

OK

Change the directory to **C:\temp**.

File name: **coffee_cup2**

Write

Additional Info: ☒ **Run Analysis**

OK

When asked if it is OK to save the model, respond **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 ran successfully, we will not bother with the details this time.

Continue

25. Analyze the results again.

Model/Output/Transform...

Transform Type: ● **Plate Forces, Stresses and Strains**

From Output Set: **2..MSC.Nastran Case 1**

OK

Select All

OK

NOTE: May not get all of the comments below.

Transform Tria3 Stress

Current Output Orientation: ● **Element First Edge**

OK

Quad3 Stress

Current Output Orientation: ● **Element Diagonal Bisector**

OK

Transform Tria4 Force

Current Output Orientation: ● **Element First Edge**

OK

Quad4 Force

Current Output Orientation: ● **Element Diagonal Bisector**

OK

View/Select... <F5>

Deformed Style: ● **Deform**

Contour Style: ● **Contour**

Deformed and Contour Data...

Output Set: **2..MSC.Nastran Case 1**

Output Vectors/Deformation: **1..Total Translation**

Output Vectors/Contour: **300002..Top Trans. Y-Normal StressB**

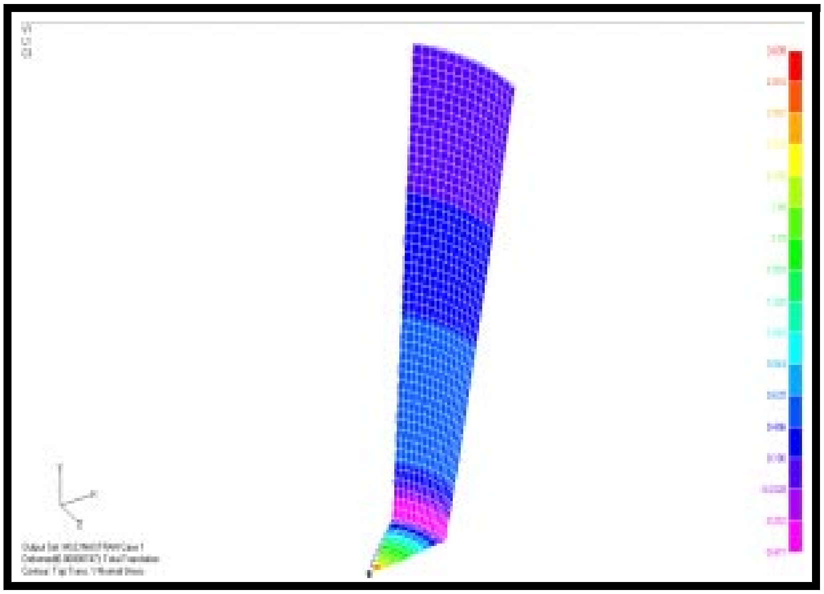
OK

OK

Notice that the new contour plot and the results are exactly the same as in the first case.

The plot should appear as follows:

Figure 13a.9



26. Open a text editor and view the .DAT file. Examine the pressure load that you created. Notice the differences in the PLOAD4 cards. This time, NASTRAN for Windows created a variable pressure across each element.

    The file should appear similar to the excerpts shown on the next page.

coffee_cup2.dat

```
$ MSC/NASTRAN for Windows Coordinate System 99 : coord 99
CORD2C        99       0      0.      0.      0.      0.      1.
0.+CS   2R
+CS   2R      1.      0.      0.
$ MSC/NASTRAN for Windows Load Set 1 : pressure
PLOAD4         1       1  0.1086 0.10966 0.11056      1.
PLOAD4         1       2  0.1086  0.1086 0.11072 0.10966
PLOAD4         1       3 0.11056 0.10966 0.11072 0.11251
            .
            .
$ MSC/NASTRAN for Windows Constraint Set 1 : bottom_corner
SPC            1       2       3      0.
SPC            1       5       3      0.
SPC            1       6       3      0.
SPC            1       7       3      0.
            .
            .
$ MSC/NASTRAN for Windows Constraint Set 2 : sides
SPC            2      15     246      0.
SPC            2      16     246      0.
SPC            2      17     246      0.
SPC            2      18     246      0.
            .
            .
$ MSC/NASTRAN for Windows Constraint Set 3 : Combined Set
SPC            3       2       3      0.
SPC            3       5       3      0.
SPC            3       6       3      0.
SPC            3       7       3      0.
SPC            3       8       3      0.
SPC            3       9       3      0.
SPC            3      14       3      0.
SPC            3      15    2346      0.
SPC            3      16     246      0.
            .
            .
$ MSC/NASTRAN for Windows Property 1 : prop_1
PSHELL         1       1   0.125       1               1               0.
$ MSC/NASTRAN for Windows Material 1 : mat_1
MAT1           1 470000.           0.333      0.      0.      0.
GRID           2      99  0.6875 3.21429  0.0625      99
GRID           5      99  0.6875 12.8571  0.0625      99
GRID           6      99  0.6875 16.0714  0.0625      99
GRID           7      99  0.6875 19.2857  0.0625      99
GRID          21      99 0.22917     45.  0.0625      99
GRID          22      99 0.15278     45.  0.0625      99
            .
            .
```

19. Further post-process the results.

First, animate the results for deformation and stress contours.

View/Select... <F5>

Deformed Style: ● **Animate**

OK

View/Options... <F6>

Category: ● **Post Processing**

Options: **Contour/Criteria Levels**

☒ **Animate**

Options: **Animated Style**

Frames: **12**

OK

After you are done viewing the animation,

View/Select... <F5>

Deformed Style: ● **Deform**

OK

27. Now create a group to isolate the part of the model we are primarily concerned about (the bottom of the cup). This will allow us to simply view the results on just that area only.

Group/Set... <Alt+F2>

Title: **Results**

OK

Group/Element/On Surface...

<Select **Surface 1**> (Surface 1 is the bottom surface)

OK

View/Select... <F5>

Model Data...

Group: ● **Active**

OK

OK

28. We can show a contour plot of the stress at a particular range that we specify ourselves.

View/Options... <F6>

Category: ● **Post Processing**

Options: **Contour/Criteria Levels**

Level Mode: **2..Max Min**

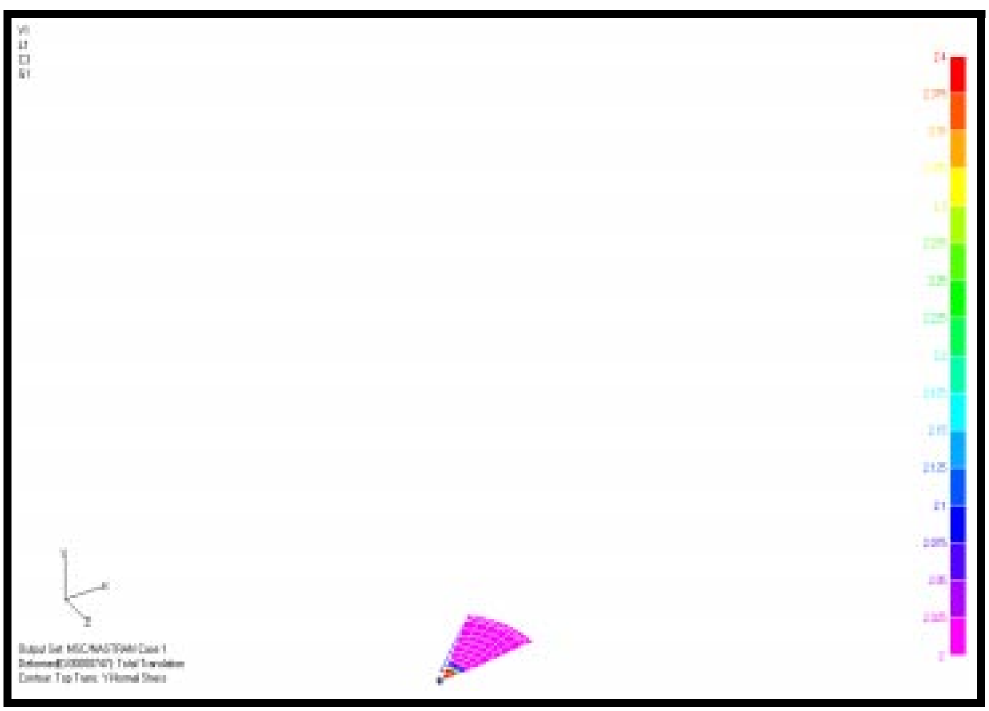
Minimum: **2.0**

Maximum: **2.4**

OK

The viewport should appear as follows:

Figure 13a.10



29. Now we want to see an XY plot of stress versus position (from the center of the cup to the edge).

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

Model Data...

Group: ● **None**

OK

To view the XY plot of stress versus position, change the coordinate system to Coord 99. X = 0 on the plot will be the center of the cup.

XY Style: ● **XY vs Position**

XY Data...

Data Selection/Category: **4..Stress**

Position: ● **X**

Coord Sys: **99..Coord 99**

Output Set: **1..MSC.Nastran Case 1**

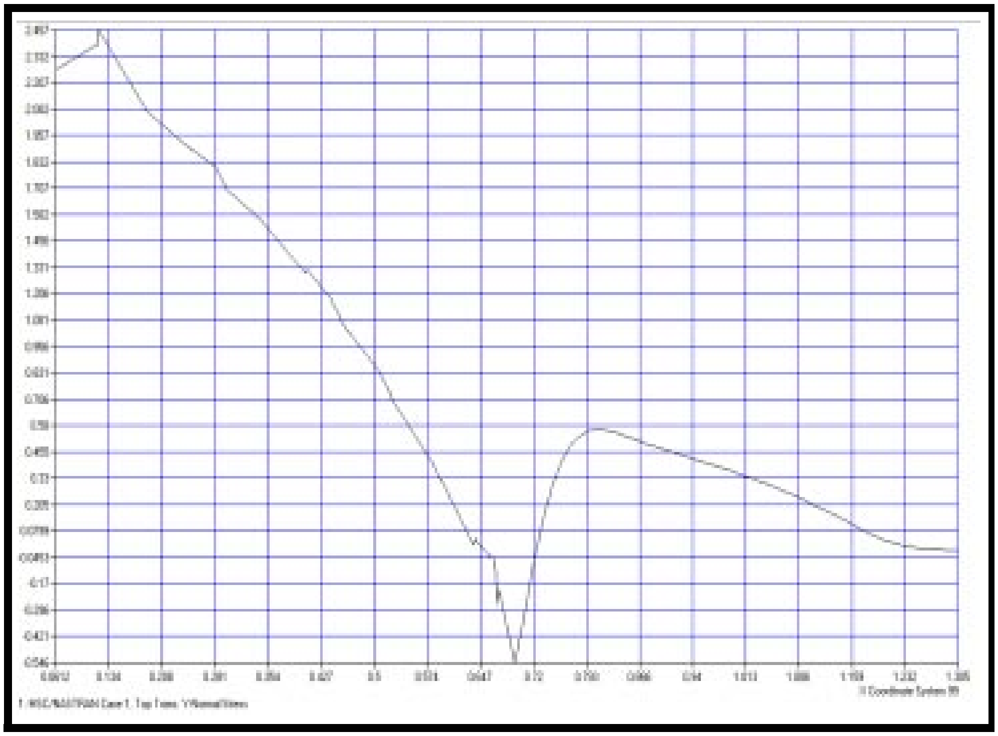
Output Vector: **30002..Top Trans. Y-Normal StressB**

OK

OK

The viewport should appear as follows:

Figure 13a.11



NOTE: To remove the XY Plot,

View/Select... <F5>

Model Style: ● **Draw Model**

OK

This concludes this portion of the exercise.

File/Save

File/Exit