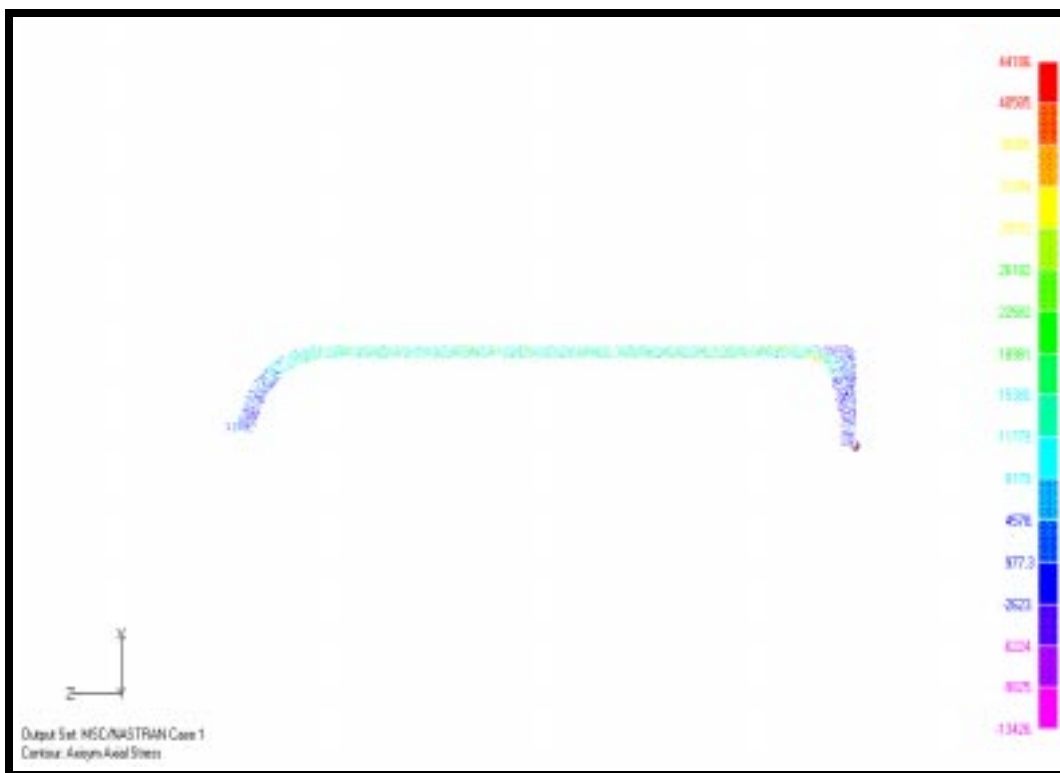


## WORKSHOP 21c

# *Scuba Tank: Axisymmetric*



### Objectives:

- Import cross-section of tank.
- Model with axisymmetry elements.
- Analysis model with 5000 psi internal pressure.





---

## Suggested Exercise Steps:

- Create a new file.
- Complete the geometry.
- Create the material and property.
- Mesh the model.
- Apply loads and constraints.
- Run analysis and evaluate 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. Import cross-section of tank.

**File/Import/FEMAP Neutral...**

Change directory to **c:\Mscn4w40\examples**.

*File name:*

**tank\_outline.NEU**

**Open**

**OK**

The workplane will not be necessary.

*Right Click* on screen to invoke the pop up menu.

**Workplane...**

Uncheck **Draw Workplane**.

**Draw Workplane**

**Done**

3. Rotate the model.

**View/Rotate... <F8>**

X:

**0**

Y:

**-90**

Z:

**90**

**Mag...**

<b>Fill View</b>
<b>OK</b>

**Figure 21c.2**



4. Create a boundary surface with imported curves.

**Geometry/Boundary Surface...**

<b>Select All</b>
<b>OK</b>
<b>Cancel</b>

5. Create material.

**Model/Material...**

<b>Load...</b>
----------------

*Library Entry:*

<b>17-4PH Stainless H1025</b>
-------------------------------

<b>OK</b>
<b>OK</b>
<b>Cancel</b>

6. Create element properties for axisymmetric element. MSC.Nastran requires CTRIAX elements with 6 nodes.

**Model/Property...**

<b>Elem/Property Type...</b>
------------------------------

**Parabolic Elements**

*Volume Elements:*

**Axisymmetric**

**OK**

*Title:*

**Axisymmetric**

*Material:*

**1..17-4PH Stainless H1025**

**OK**

**Cancel**

7. Mesh the geometry.

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

*Size:*

**0.125**

**OK**

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

**Select All**

**OK**

*Property:*

**1..Axisymmetric**

*Max Element Aspect Ratio:*

**1.**

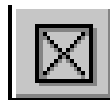
**: 1**

*Element Shape:*

**All Triangles**

**OK**

Click the *Quick Options* icon.



**All Entities Off**

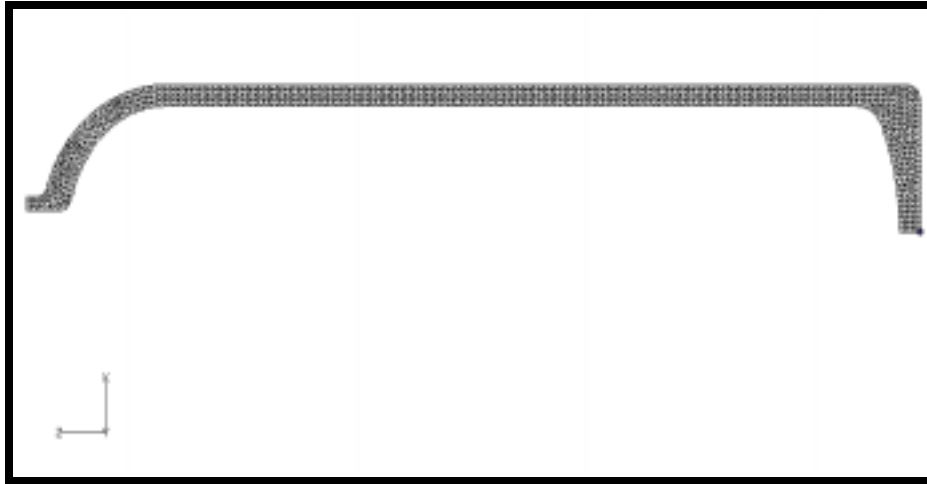
**Element**

**Load - Pressure**

**Constraint**

Done

Figure 21c.3



- 8. Apply constraints to model.

Model/Constraint/Set... <Shift+F2>

Title:

Symmetric

OK

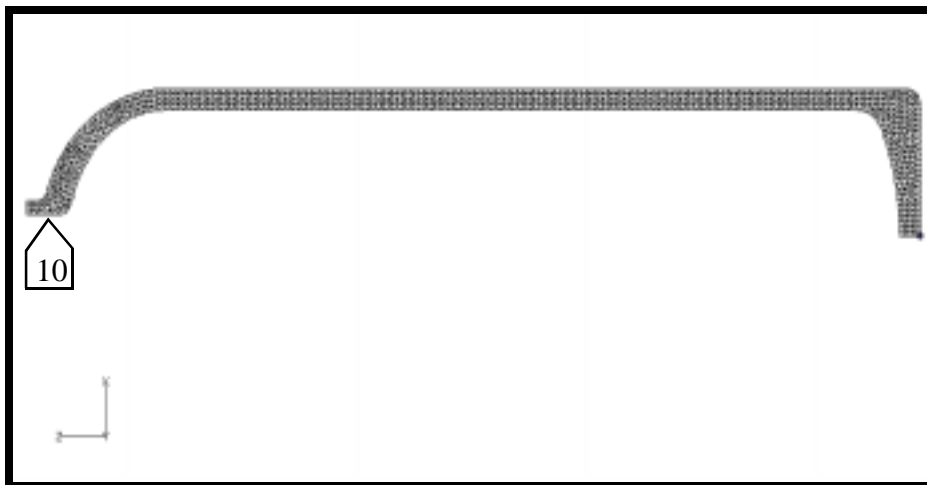
Model/Constraint/Nodal...

Method ^

on Curve

Select Curve 10.

Figure 21c.4



OK

DOF:

TZ

OK

Cancel

9. Create loads.

Model/Load/Set... <Ctrl+F2>

Title:

5\_Ksi

OK

Model/Load/Elemental...

Select All

OK

(highlight)

Pressure

Pressure:

5000

OK

Method:

Adjacent Faces

Click in the **Element** box and select an edge on the inside surface.

Face:

3

OK

Cancel

**Note:** Using this method of applying the pressure, we select the first element and the tolerance to determine the other element “adjacent” to the first.

10. Run analysis.

File/Export/Analysis Model... <Ctrl+T>

OK

---

*File Name:*

tank\_axi

**Write**

*Additional Info:*

**Run Analysis**

**OK**

**Yes**

*File Name:*

tank\_axi

**Save**

**Continue**

11. Review results.

Click the *Quick Options* icon.



**All Entities Off**

**Element**

**Done**

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

*Deformed Style:*

**Deform**

*Contour Style:*

**Contour**

**Deform and Contour Data...**

*Deformation (pull-down):*

**1..Total Translation**

*Contour (pull-down):*

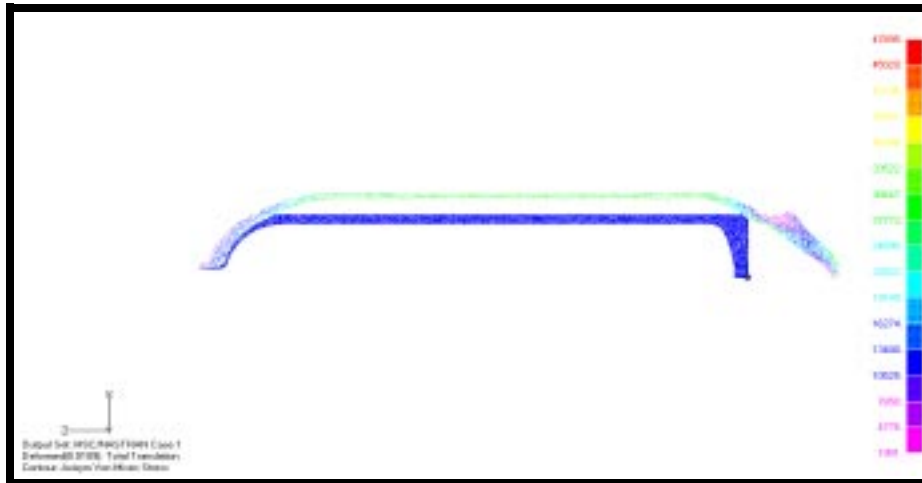
**6035..Axisym Von Mises Stress**

**OK**

**OK**

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

Figure 21c.5



View/Select... <F5>

Deformed Style:

None - Model Only

Contour Style:

Contour

Deform and Contour Data...

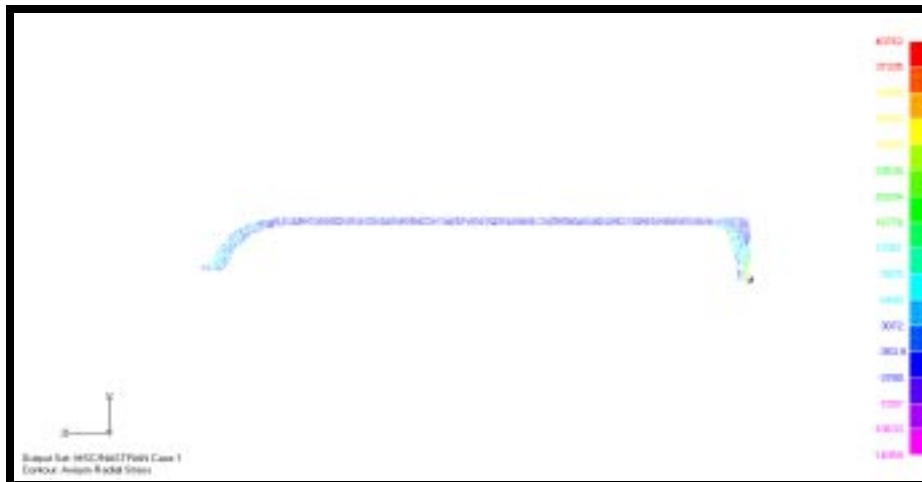
Contour (pull-down):

6028..Axisym Radial Stress

OK

OK

Figure 21c.6



---

View/Select... <F5>

Deform and Contour Data...

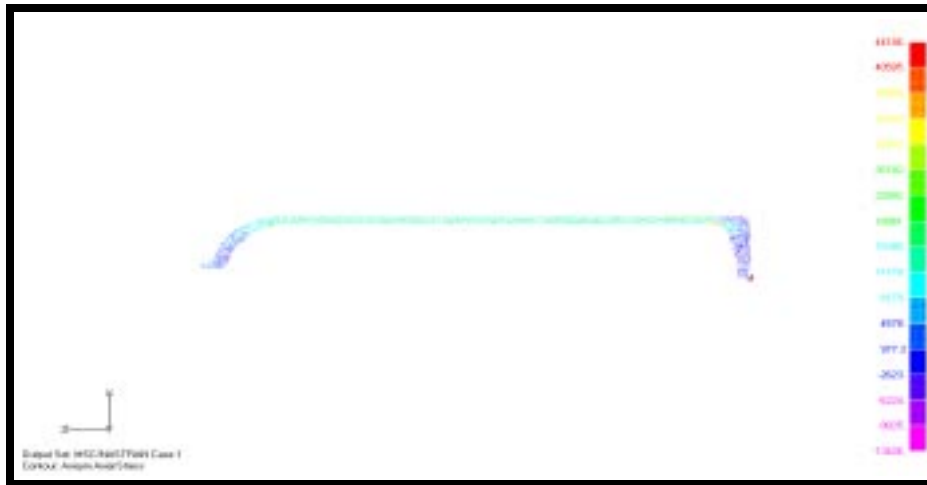
Contour (pulldown):

6030..Axisym Axial Stress

OK

OK

Figure 21c.7



This concludes this exercise.

File/Save

File/Exit