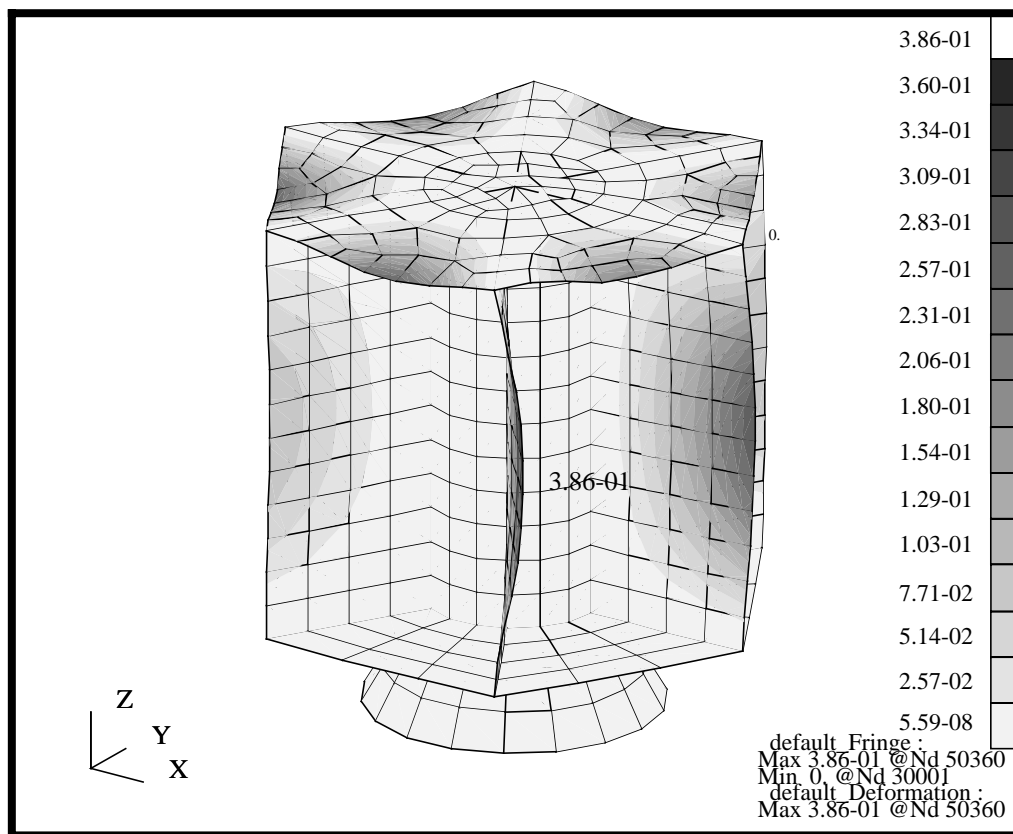


LESSON 7

Post-Processing Static Results of a Space Satellite



Objectives:

- Post Process model Results from an OP2
- Familiarize you with the results application and graphical interface.
- Create deformed, fringe, vector, tensor, XY plots and Freebody plots of various results quantities.



Model Description:

In this exercise, you will examine the static results of the Satellite model by using the results tools. You will be interrogating a variety of result types, namely deformation, reaction forces and stresses. You will create Deformed, Fringe, and Vector plots of the displacement. You will also set up an animation and create an XY plot of the results along an edge. You will create Fringe and Tensor plots of stresses for Bars and Quad type elements. You will learn how to control your results coordinate system, results averaging and extrapolation. Additionally, you will learn how to enhance you plots by changing titling, results labels, scale factor. Finally, you will create graph of stress versus path for von Mises across the Upper platform.

Suggested Exercise Steps:

- Start MSCPATRAN and create a new database `static_results.db`.
- Change the *Tolerance to Default* and the *Analysis Code* to **MSC/NASTRAN**.
- Import the new satellite model and results for this exercise by reading the MSC/NASTRAN output2 file `satellite_static.op2`.
- Create a deformed shape plot of the **Displacement** result values using Quick Plot.
- Turn off the undeformed plot of the Satellite model. Change the orientation of the deformed image of the Satellite model to an **Isometric view**. Render the deformed plot using a **Smooth shaded** render style and decrease the *Deformation Scale Factor* to **0.05**.
- Create a fringe plot of the displacement magnitudes. Make sure to turn element edges on and render the edges in black.
- Animate the fringe and deformation created in the last step by turning on the animate button. Set the animation method to Modal, the graphics to 3D and the number of animation Frames to 10.
- When you are done viewing the animation, be sure to turn off the animation by selecting the Stop Animation button
- Reset the graphics and change the view to an Iso View #3
- Create a Vector plot of the Constraint Forces. Plot each of

the vector components.

- Again, Reset the Graphics and view.
- Create a fringe plot targeting only the Bar2 elements plotting the axial (X) stress. Choose the bar's maximum stress recovery point and modify the display attributes such that the fringe on the bars is displayed wider. Finally, save the plot as bar_xx_stress.
- Create another fringe plot targeting the CQUAD4 elements. This time plot the maximum von Mises stress on the element. Change the display attribute to black element edges. Save the plot naming it shell_von_Mises
- Now post the deformed plot along with the shell_von_Mises fringe plot using Results/Post
- Modify and replot the shell_von_Mises fringe plot and change the averaging domain to None.
- Reset the graphics and view to bottom view and create a group containing only the upper platform elements. Make sure to unpost all other groups.
- Create a Tensor plot of the X, Y and Z stress components. Target only the Quad4 elements. Change the vector display such that the Spectrum is on, the components are colored by relative to the spectrum and the vectors are scaled relative to the model size.
- Replot the tensor plot, but this time specify the transformation coordinate as global.
- Create an X vs. Y graph of the von Mises stress along a path defined by two vertices of the upper platform. Use a spline curve fit and turn on the symbols.
- Create another group of just the bar elements that represent the navigational platform support structure. Unpost all other groups.
- Create a Freebody plot of the bars to determine reaction loads at the bars ends.
- Close and Quit PATRAN.

Exercise Procedure:

1. Open a new database and name it **static_results.db**.

Open a New Database

In this exercise, we will open a new database and import some model information/results data into the database. Alternatively, we could have opened the **satellite.db** and imported the results into that database. Since we have chosen to separate the static from the modal results, we have made a conscious decision to separate the model data from the results as well as separate the results solutions.

File/New...*New Database Name***static_results****OK****New Model Preference***Tolerance*◆ **Default***Analysis Code***MSC/NASTRAN****OK**


2. Import the model and results for this exercise by reading the output **satellite_static.op2** file.

Import the model and results

In this step, we will read in the model data and results data from the MSC/NASTRAN op2 results file. The model data contains enough information, namely nodes and element connectivity, to post process the results. It does not contain/import groups, element properties, materials or loads and boundary conditions. If that information is needed, we could have alternatively imported the MSC/NASTRAN input file.

◆ **Analysis***Action:***Read Output2***Object:***Both***Method:***Translate****Select Results File...**

Selected Results File

satellite_static.op2 

Note: this can be selected from the Available Files list

OK

Apply

Change the view to Isometric View and the render style to Smooth shaded using the Toolbar



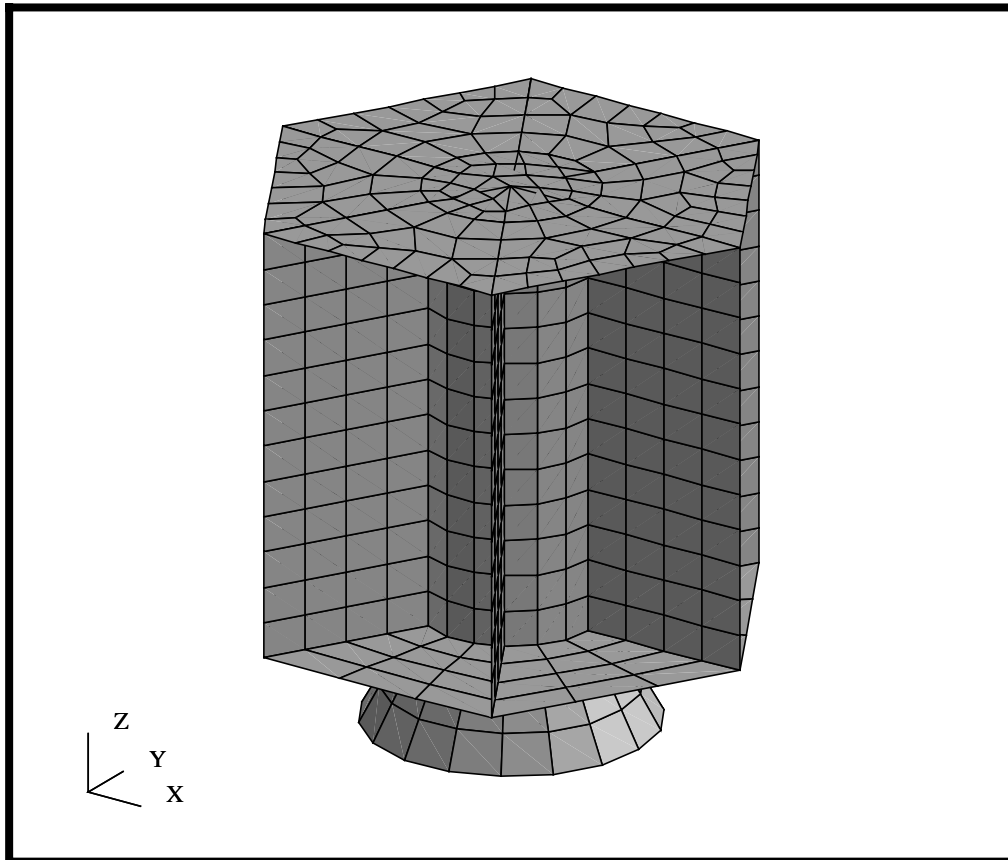
Isometric View 3



Shaded Smooth

Your model should look like the figure below.

Figure 7.1. Satellite Model Nodes and connectivity ready for post processing.



There are two approaches for post processing results in MSC/PATRAN. One approach is to use the multi-purpose (e.g. fringe, deformation and animation) **Quick Plot** form. The other is to use the response type specific forms, e.g. deformation, fringe, etc., Advanced Forms.

The Advanced Forms are designed to allow the user more flexibility/control on the manipulation and rendering of the model. However, this may require more user input compared to doing the same kind of plots in the Quick Plot form. In this exercise, you will be using both form types. We will start with the Quick Plot forms.

3. Create a deformed shape plot of the **Displacement** result values using Quick Plot.

**Create a
Deformed
Shape Plot**

◆ **Results**

Action:

Create

Object:

Quick Plot

Select Result Cases

**LAUNCH STATIC, Static
Subcase**

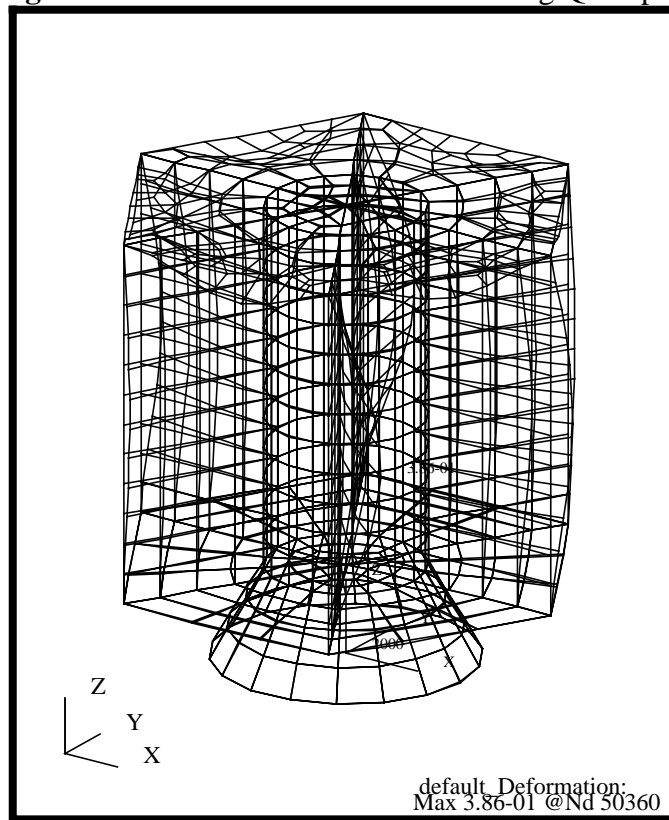
Select Deformation Result

Displacements, Translational

Apply

Your model should look like the figure below.

Figure 7.2. Default Deformation Plot using Quickplot.



4. Turn off the undeformed model in the Quick Plot Deformation Plot of the Satellite model.

By default, the deformed and undeformed plots are superimposed. The deformed model is in white and the undeformed model is blue. Also, the deformed render style is set to Wireframe. Note, the maximum deformation is indicated in the lower right hand corner of the viewport as well as on the model.

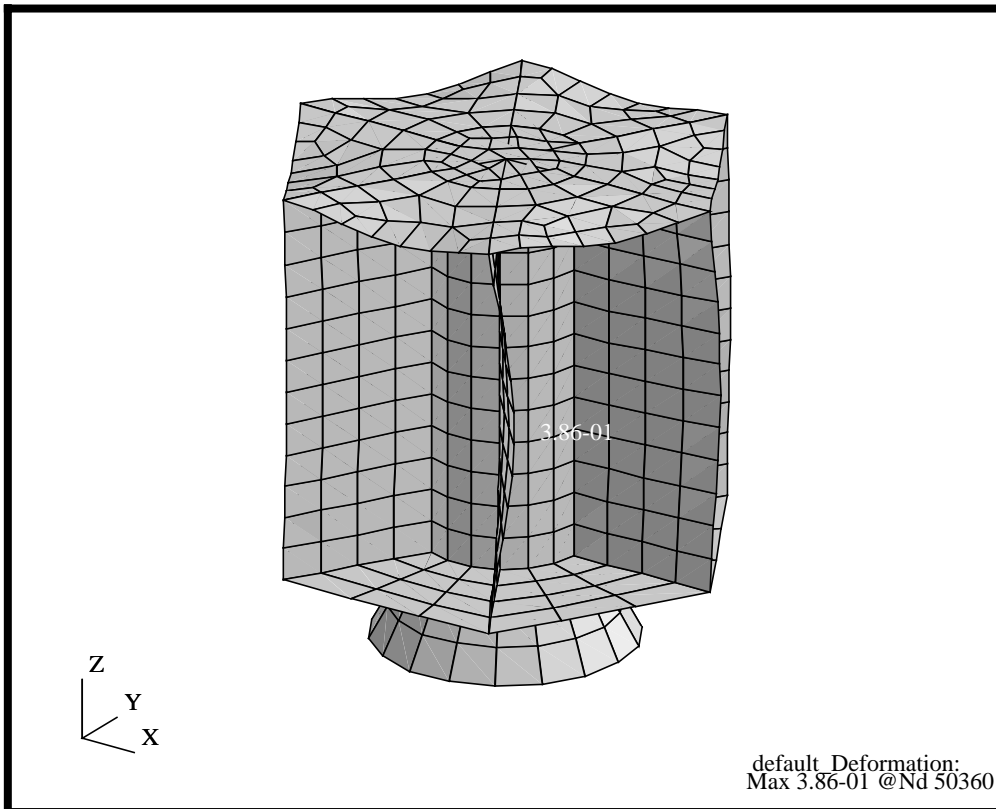
Near the top of every results form are a set of Icons which are used to control the results display. The icons and forms they control will change depending on which type of result tool. These icons control the results selection, entities the results are to be applied on (targeted entities), results display specific to the type of result tool and animation setup.

The Deform Attributes icon controls the appearance, scale factor and graphic options of any deformed plots created under Quick Plot. Turn off the undeformed plot, change the render style to Shaded and set the scale factor to 0.05 using Deform Attributes icon.

**Deform Attributes***Render Style:***Shaded***Scale Factor***0.05** **Show Undeformed****Apply**

Your model should be similar to the one shown below. You will notice the deflection is less than the previous deformation plot. Also, by turning off the undeformed entities and turning on the Shaded render style, the deformed plot becomes easier to visualize.

Figure 7.3. Deformed Plot of the Satellite using modified Display Attributes



Create a Fringe Plot of Displacements

5. Create a fringe plot of the displacement magnitudes.
Render the plot with the element edges turned off.

Another way of at looking at the displacement results is by looking at a fringe result of either a component (X, Y or Z) or the displacement magnitude. In this step, we will create a fringe plot of the displacements to better visualize numerical amount of deformation. Additionally, we will superimpose the values on top of the deformed plot.

First, change back to the Select Results form using the following Results Icon.



Select Results

Next, create the fringe plot. Under the **Results** form, select:

Action:

Create

Object:

Quick Plot

Select Result Cases

LAUNCH STATIC, Static Subcase

Select Fringe Result

Displacements, Translational

Quantity:

Magnitude

Select Deformation Result

Displacements, Translational

Apply

6. Recreate a fringe plot of the displacement magnitudes, but this time render the plot with the element edges turned on.

We will now redo the plot but changing some of the Fringe Attributes. The default fringe plot sets the display to Free Edges. We will change this to Element Edges. Select the Fringe Attributes icon from the Quick Plot form.



Fringe Attributes

Fringe Edges

Black



Display:

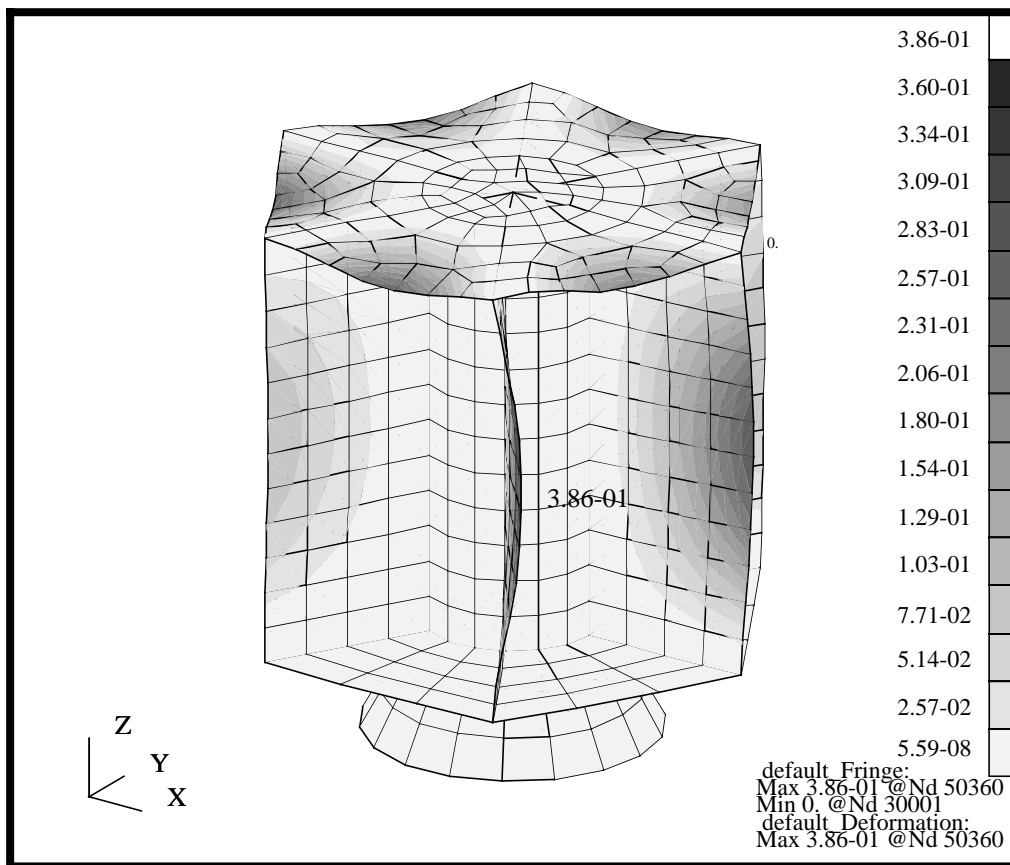
Element Edges

Apply

It should also be noted that the Deform Attributes previously set are still retained.

Your model should now look like the figure below. The fringe results are superimposed over the existing deformation. By turning on the element edges, it becomes easier to see the shape of the satellite, especially inside the bays.

Figure 7.4. A combined Deformation and Fringe Plot of the Satellite Model



7. Create a modal animation of the displacements (Fringe and Deformation).

We will now create an animation of the displacement deformation and displacement magnitude fringe plot. We will use Quick Plot to setup a simple modal animation. A modal animation is an animation sequence which varies the results values (deformation and fringe) from +Max to -Max values.

Animate the Deformation and Fringe Plots

Click on the Select Results icon in Quick Plot form.



Select Results

Now recreate the same plot, but this time enable the animation.

<i>Action:</i>	<input type="button" value="Create"/>
<i>Object:</i>	<input type="button" value="Quick Plot"/>
<i>Select Result Cases</i>	<input type="button" value="LAUNCH STATIC, Static Subcase"/>
<i>Select Fringe Result</i>	<input type="button" value="Displacements, Translational"/>
<i>Quantity</i>	<input type="button" value="Magnitude"/>
<i>Select Deformation Result</i>	<input type="button" value="Displacements, Translational"/>

Animate

Before hitting Apply and creating the Animation, let's look at the Animation Options.



Animation Options

Now verify the following options are set.

Animate Fringe

Animate Deformation

<i>Animation Method</i>	<input checked="" type="checkbox"/> Modal
<i>Animation Graphics</i>	<input checked="" type="checkbox"/> 3D
<i>Number of Frames</i>	<input type="text" value="10"/>
<input type="button" value="Apply"/>	

MSC/PATRAN will now build the frames and start the animation. The *Animation Control* form will then be displayed. Once the model is animating, you can use the middle mouse button to rotate the model

(only available when Animation Graphics is set to 3D). When you are finished, you can use the Stop Animation button on the Animation Control form to stop the animation.

Stop Animation

The Quick Plot provides a simple interface to displacement, fringe and modal animations. However, if you want to go beyond these result display types, you will need to use response specific result forms, the Advanced Forms. You will now use some of the Advanced result types to get a vector plot of the reaction forces.

8. Create a Vector plot of the Constraint forces.

We will now plot the Reaction Forces on the model. This is an example of a result plot type that can not be generated using Quick Plot.

To begin, we will start by resetting the view and cleaning up the Display. From the Main Form, use the Reset Graphics to clear the display.

Create a Vector Plot of the Constraint Forces



Reset graphics

Next, reset the view and render style using the Toolbar Icons.



Isometric View 3



Wireframe

Now create a vector plot of the reaction forces. We will specify the plotting the X, Y and Z components of the reaction forces.

Action:

Create

Object:

Marker

Method:

Vector

Select Result Case(s)

LAUNCH STATIC, Static Subcase

Select Vector Result

Constraint Forces,
Translational

Show As:

Component

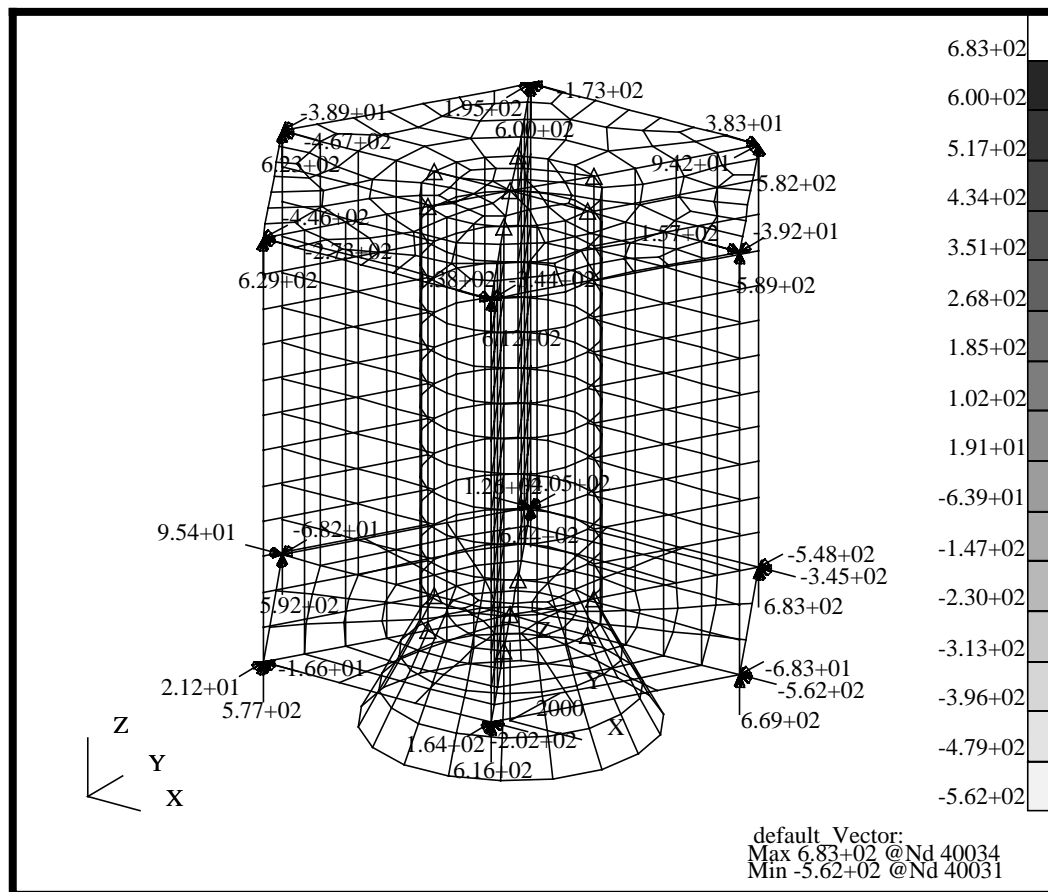
XX YY ZZ

Animate

Apply

Your model should now look like the figure shown below.

Figure 7.5. Vector Plot of the Reaction Forces for the Launch Condition.



Create a Fringe Plot of the Bar Axial Stresses

9. Create a results fringe plot of the bar axial stresses.

We will now plot the axial stresses on the bar elements and the von Mises stresses on the shells for the model. To begin, we will start by resetting the view and cleaning up the Display. From the Main Form, use the Reset Graphics to clear the display.



Reset graphics

Now create a fringe plot of the maximum axial stress on the Bar elements. Note the selection of stress recovery locations, in this step you will be selecting all four locations (Points C, D, E and F) and MSC/PATRAN will take the maximum value for each results position.

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Fringe"/>
<i>Select Result Case(s)</i>	<input type="text" value="LAUNCH STATIC, Static Subcase"/>
<i>Select Fringe Result</i>	<input type="text" value="Stress Tensor"/>
<input type="text" value="Position...(At Center)"/>	
<i>Positions</i>	<input type="text" value="At Center"/> <input checked="" type="text" value="At Point C"/> <input checked="" type="text" value="At Point D"/> <input checked="" type="text" value="At Point E"/> <input checked="" type="text" value="At Point F"/> <input type="text" value="At Z1"/> <input type="text" value="At Z2"/>
<i>Option:</i>	<input type="text" value="Maximum"/>
<input type="text" value="Close"/>	
<i>Quantity:</i>	<input type="text" value="X Component"/>

First, we will only plot the stresses for the bar elements. We will do this by selecting Target Entities icon from the results menu.



Target Entities

Now select the Bar2 entities. Also notice, the Select Element Types only list the entities in the model (Point elements, Bar2 and Quad 4 elements types).

Target Entity:

Element Types

Select Element Types

Point

Bar 2

Quad 4

Next, modify the default behavior of the Fringe plot on the bars and make the fringe for the bars “wider”. This will help to visualize the axial stresses on the elements.



Display Attributes

You can customize the display by modifying the Style and Width of the bar elements.

Style:

Discrete/Smooth

Shading:

None

Style:



(Straight continuous line)

Width:



(Widest line width)

Finally, give the plot a name using the Plot Options



Plot Options

Save Fringe Plot As:

bar_xx_stress

Apply

This will create a fringe plot for just the bar elements. By naming the plot, we can later post and unpost this plot using the results Action Post.

10. Create a results fringe plot of the quad von Mises stresses.

Now we will add the von Mises stresses for the shells on top of the existing bar axial stress fringe plot.

Create a Fringe Plot of the Quad von Mises Stresses



Select Results

Now modify the Positions to reflect those of the Quad4 type elements

Position...(At Point C)

Positions

At Center
At Point C
At Point D
At Point E
At Point F

At Z1
At Z2

Option:

Maximum

Close

Quantity:

von Mises

Again, change the target entities to select only the Quad4 entities.



Target Entities

Target Entity:

Element Types

Select Element Types

Point
Bar 2
Quad 4

Again, modify the display attributes to customize the Fringe Display.



Display Attributes

You can customize the display by modifying the Style and Width of the bar elements.

Style:

Discrete/Smooth

Shading:

None

Fringe Edges:

Black



Display:

Element Edges

Style:



(Straight continuous line)

Width:



(Thinnest line width)

Finally, give the plot a name using the Plot Options



Plot Options

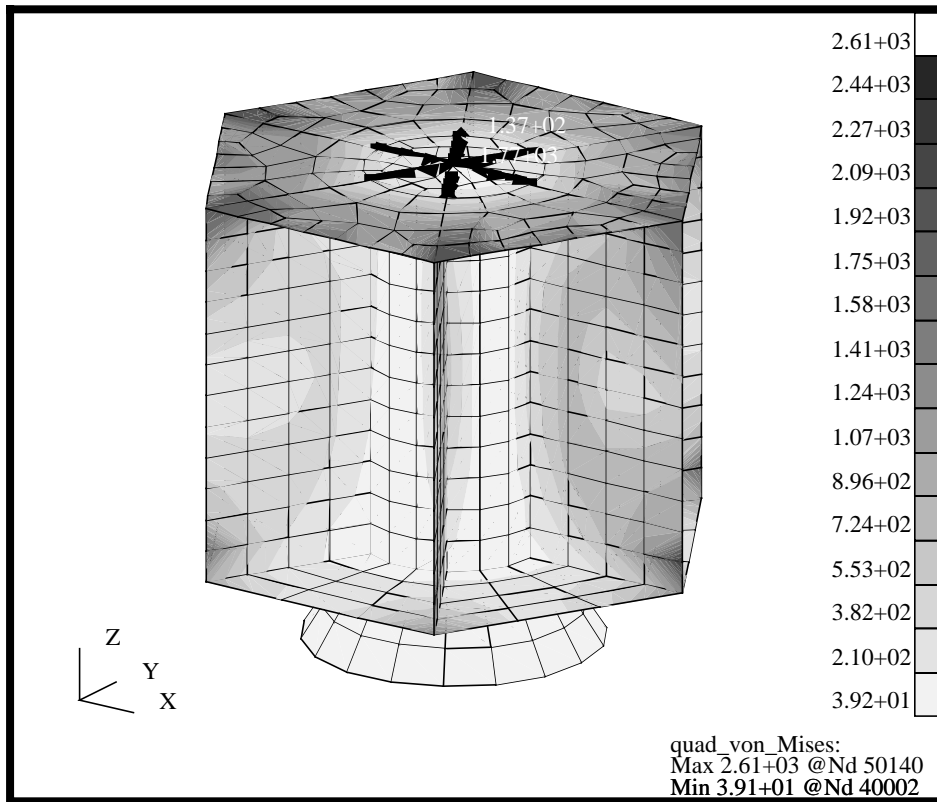
Save Fringe Plot As:

shell_von_Mises

Apply

Your model should now look like the figure below

Figure 7.6. Bar Fringes Superimposed on top Shell Stress



11. Now display the shell von Mises stresses on top of the deformed plot previously create and remove the bar axial stresses.

By “naming” the fringe plots, you can now post and unpost any plot in much the same way as you can post and unpost groups of elements. We will now post the shell_von_Mises fringe plot along with the last deformed plot. Since we did not name the deformation, the last plot of a particular plot type (in this case deformation) gets automatically saved under the name of “default”.

Action:

Object:

Existing Plot Types

Post
Plots
DEF_default_Deformation
FRI_default_Fringe
VEC_default_Vector
FRI_bar_xx_stress
FRI_shell_von_Mises

Post and Unpost Results

Apply

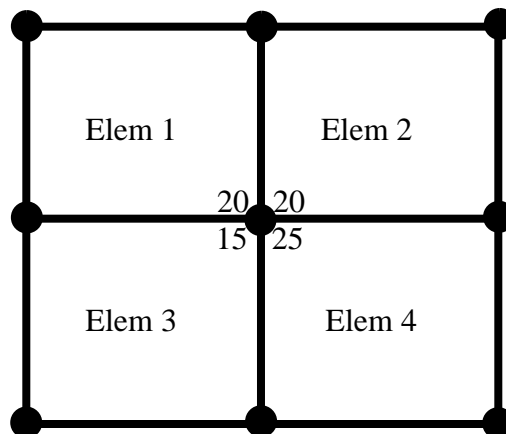
Upon Apply, you should see the same figure as before, but this time without any bar stresses and the shell fringe plotted on a deformed shape. Using Post Plots, you can mix and match the results plots.

Change the Averaging Domain for the Fringe Plot

12. Now modify the von Mises stress fringe plot and change the Averaging at element nodes.

The default “Averaging Domain” for MSC/PATRAN is to average All Element nodal results at each node. This provides a smooth contour/fringe plot. However, if you want to compare the results back to the printed results, the F06 file, you should modify the approach used to average the results. MSC/PATRAN provides several averaging methods, including All, Material, Property, Targeted Entities and None. The best way of looking at stresses is to create a fringe plot that has NO Element-to-Element Averaging at the nodes.

For example, in the figure below, there are 4 quads which all share a common node at the center. Each element contributes a result value to this node. In this case, Elements 1, 2, 3 and 4 contribute values of 20, 20, 15 and 25, respectively. The default Fringe for this case would be to yield an average, or a value of 20 at the center node. However, this value does not correspond to the maximum value of 25. If we were to change the averaging domain from All to None, no averaging would be performed and the fringe would be discontinuous across the element boundaries. Additionally, it would show the maximum value of 25 at the center node of Element 4.



In general, for well discretized models, there should be little difference between averaging domain of All and None across similar element type/property/material. If you start to see large fringe/contour jumps across elements in a critical area, this may indicate your mesh is not discretized enough.

We will now modify the Fringe values for the shell_von_Mises plot to use an Averaging Domain of None.

Action:

Modify

Object:

Fringe



Plot Options

We will select the shell von Mises plot we previously made and modify the Averaging Domain.

Existing Fringe Plots...

Fringe

default_Fringe

bar_xx_stress

shell_von_Mises

Close

Averaging Definition/Domain:

None

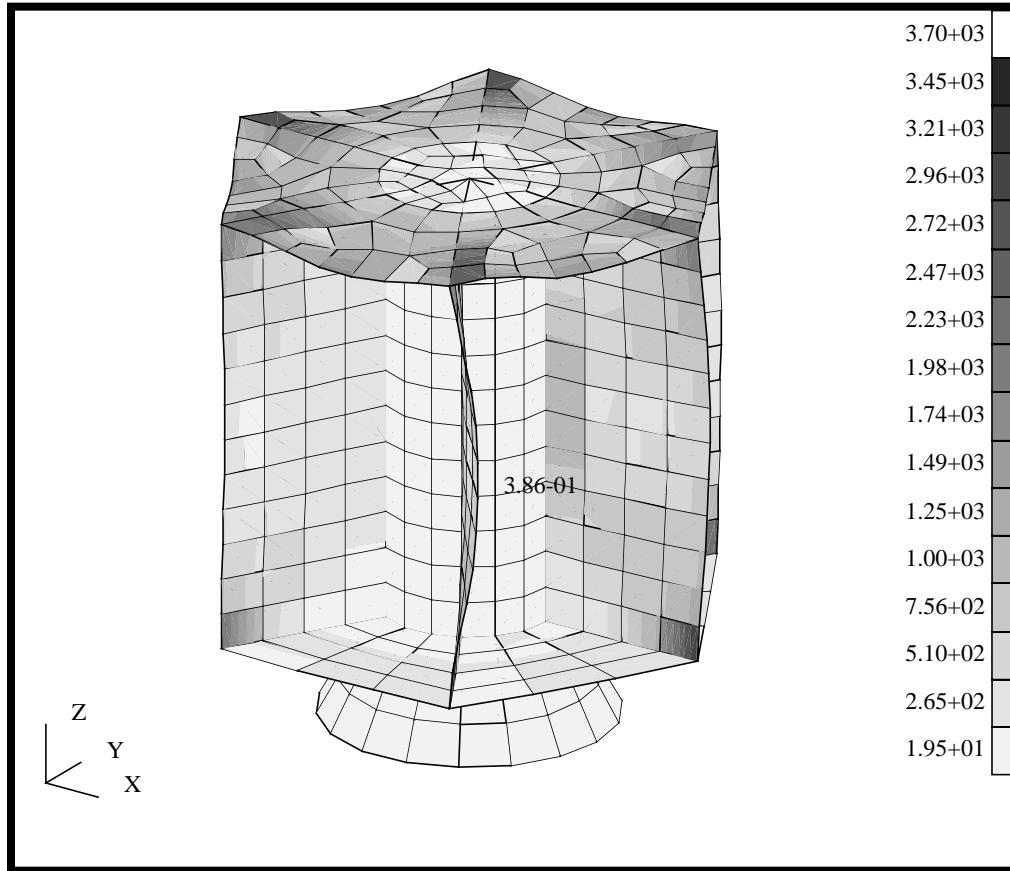
Averaging Definition/Method:

Derive/Average

Apply

Your display should be similar to the one shown below. The Fringe will indicate the location of the maximum von Mises stress.

Figure 7.7. Shell Stress Plotted with the Averaging Domain set to None



Create a Group for the Upper Platform

13. Now we will create a group for the upper platform. Lets reset the display and view and create a Upper Platform group.

First use the Main Form to reset the graphics.



Reset graphics

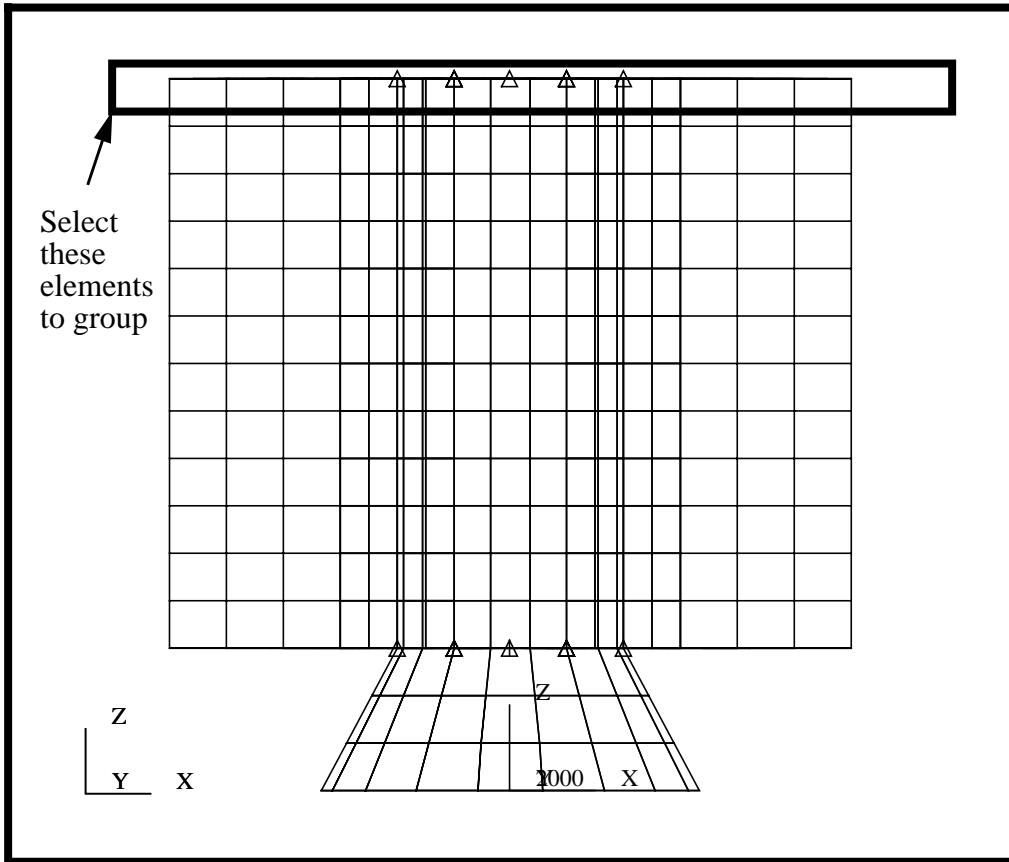
Next, reset the view and render style using the Toolbar icon.



Bottom View

Your display should be similar to the one shown below.

Figure 7.8. Elements to Select to Create Upper Platform Group.



Now create a group of the Upper Platform using the Group Create functionality.

Group/Create...

New Group Name

Upper Platform

Make Current

Unpost All Other Groups

Entity Selection

Select only the top Quads shown in the figure above

Apply

Cancel

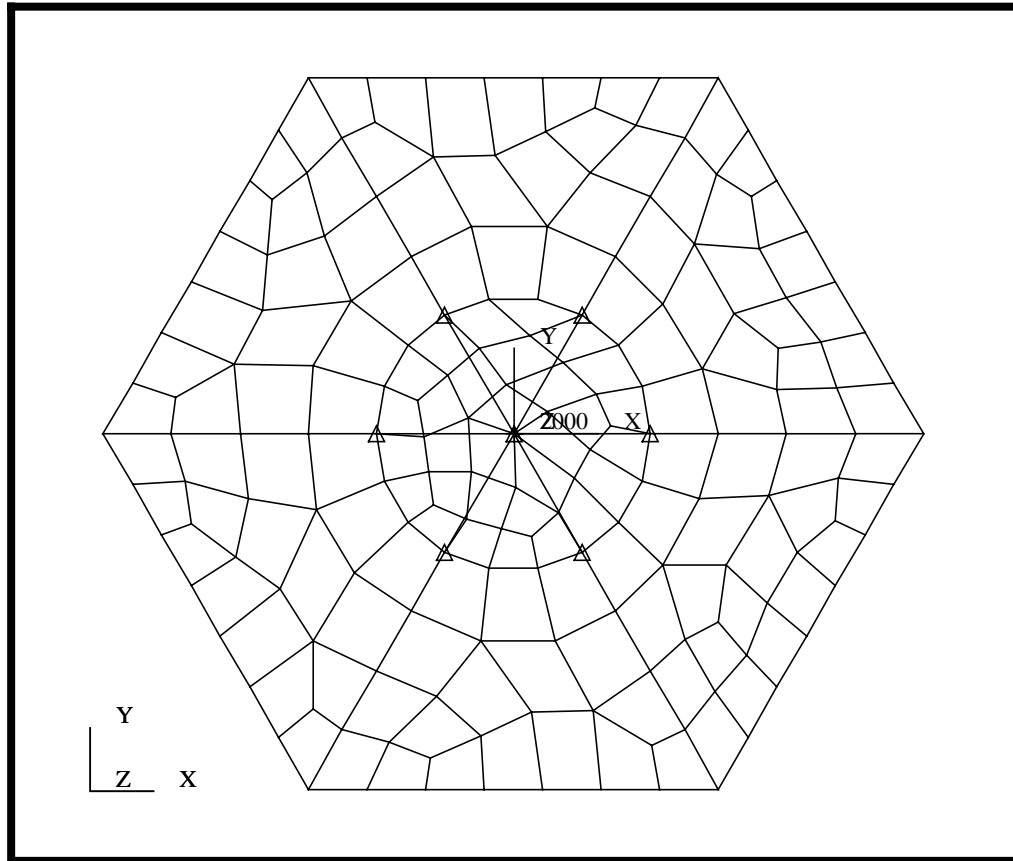
Change the view to a top view using the Toolbar Icons



Front View

Your model should look like the figure below

Figure 7.9. Front View of the Upper Platform Group



**Create a
Tensor Plot
of the Quad
Stresses/
Element CID**

14. Create a Tensor plot for the model using the default element coordinate system.

We will now look at the effect a coordinate system transformation can have on the results. We will generate a Tensor plot for the Upper Platform using the Default coordinate system. In this case, the results are based on the local element coordinate system that is aligned with the element edge direction.

For clarity, we will be displaying only the 3 principal tensile stresses components, XX, YY and ZZ, and no shears. This will help illustrate the effect result transformation has on the display/accuracy of the results.

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Marker"/>
<i>Method:</i>	<input type="text" value="Tensor"/>
<i>Select Result Case(s)</i>	<input type="text" value="LAUNCH STATIC, Static Subcase"/>
<i>Select Vector Result</i>	<input type="text" value="Stress Tensor"/>
<input type="text" value="Position...(At Z1...)"/>	
<i>Positions</i>	<input type="text" value="At Center"/> <input type="text" value="At Point C"/> <input type="text" value="At Point D"/> <input type="text" value="At Point E"/> <input type="text" value="At Point F"/> <input checked="" type="text" value="At Z1"/> <input type="text" value="At Z2"/>
<i>Option:</i>	<input type="text" value="Maximum"/>
<input type="text" value="Close"/>	
<i>Show As:</i>	<input type="text" value="Component"/>
	<input checked="" type="checkbox"/> XX <input checked="" type="checkbox"/> YY <input checked="" type="checkbox"/> ZZ <input type="checkbox"/> XY <input type="checkbox"/> YZ <input type="checkbox"/> ZX
<input type="checkbox"/> Animate	

Again, change the target entities.



Target Entities

Now select the Quad4 entities.

<i>Target Entity:</i>	<input type="text" value="Element Types"/>
-----------------------	--

Select Element Types

Point
Bar 2
Quad 4

Again, modify the display attributes to customize the Tensor Display.



Display Attributes

You can customize the display by modifying the Style and Width of the bar elements.

◆ Spectrum

Show Spectrum

Length:

Model Scaled

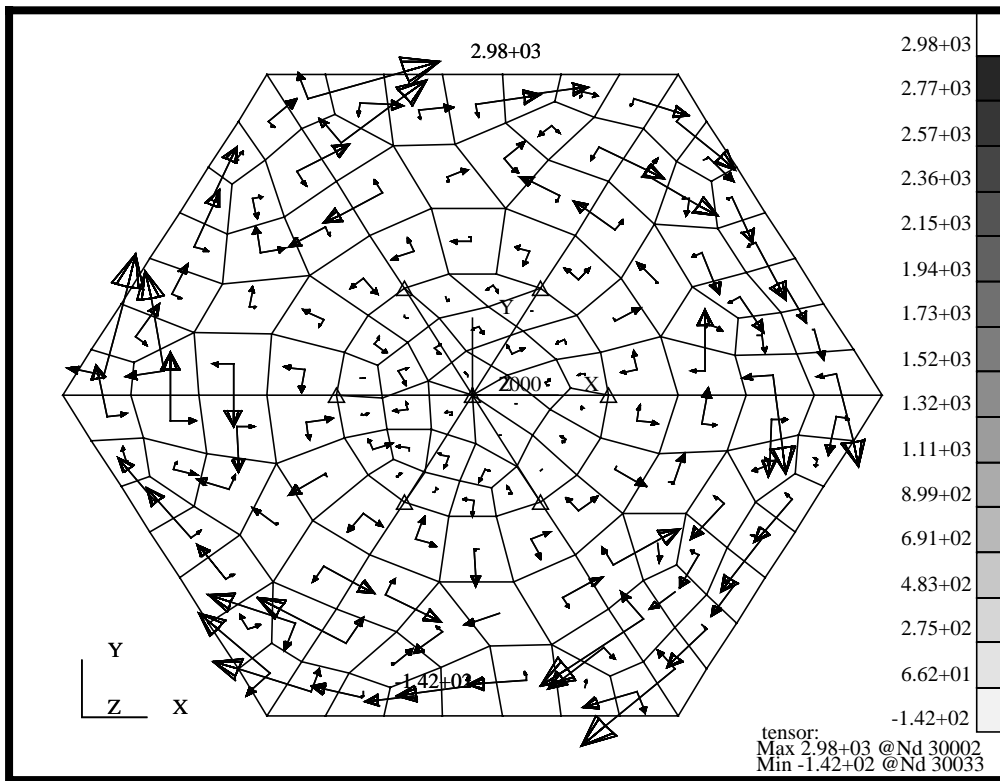
Show Tensor Box

Show Tensor Label

Apply

Your model should look like the figure below. You will notice that very few of the element edges are aligned and thus, many of the vectors are not parallel to each other. Therefore, you CANNOT sum the sum/average results from element to element. reliably.

Figure 7.10. Vector Plot of Principal Components of Stress in Element CID.



15. Create another Tensor plot for the model using the global coordinate system.

We will create another Tensor plot, but this time we will specify the global coordinate system. All other result options will remain the same as the previous plot. Change the Coordinate Transformations located under Plot Options.

Create a Tensor Plot of the Quad Stresses/ Global CID



Plot Options

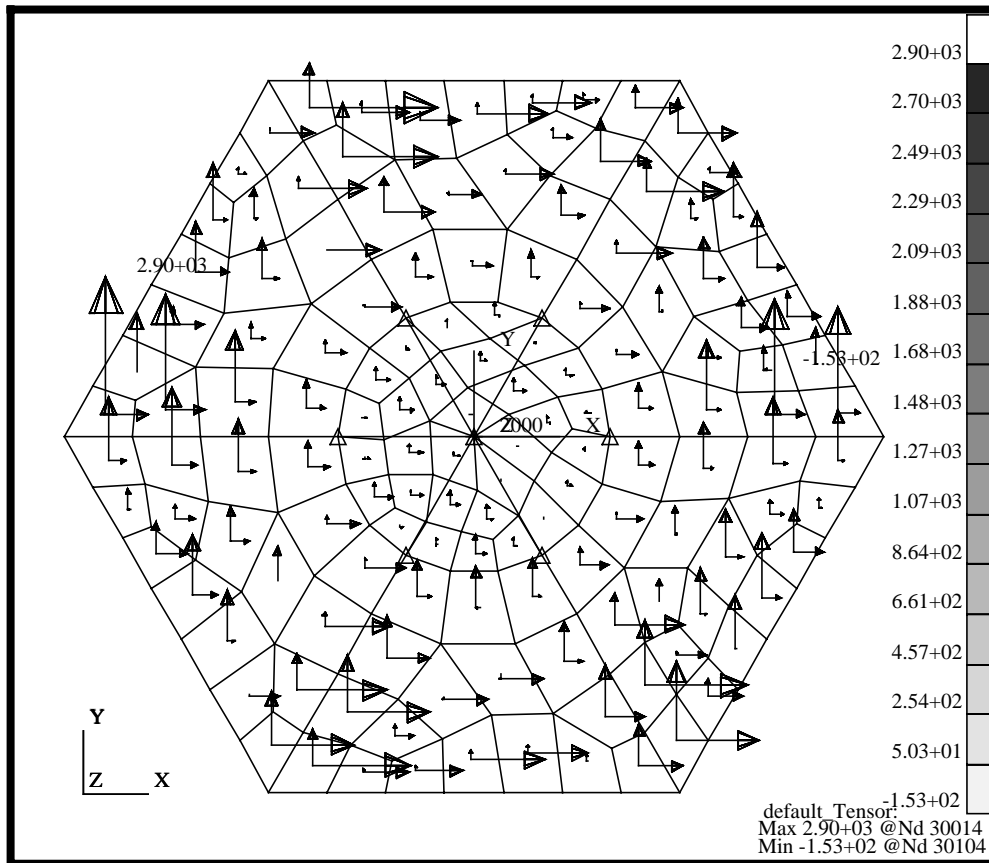
Coordinate Transformation:

Global

Apply

Your model should now look like the figure below.

Figure 7.11. Vector Plot of Principal Components of Stress in Global CID.



You will notice that the second plot has all the tensor components aligned, which is not the case for the first plot does. The stress components in the first plot have the X and Y principal stress axis alignment based on the element edge orientation. The second result plotted these align with the global coordinate system. Thus, the second plot gives you an “apples-to-apples” comparison for vectors from element to element. This does not make a difference for stress values that is based on invariants, such as von Mises stresses. However, if you are comparing stresses that are directional in nature, (i.e.: stress components: XX, YY, ZZ, XY, XZ, YZ) you will want to verify and set the coordinate transformation.

The coordinate transformation can be set for Fringe plots in much the same way. However, with fringe plots it may be even more important since the results are usually combined/averaged at the nodes. If no coordinate transformation is specified, the net result is two directional stress not aligned will be combined. Again, for invariant results, such as von Mises stresses, coordinate transformations are not important since the value is not dependent on direction.

Create an XY Plot of the Stresses

16. Finally, plot the stresses along a line that stretches from vertex to vertex horizontally across the upper platform.

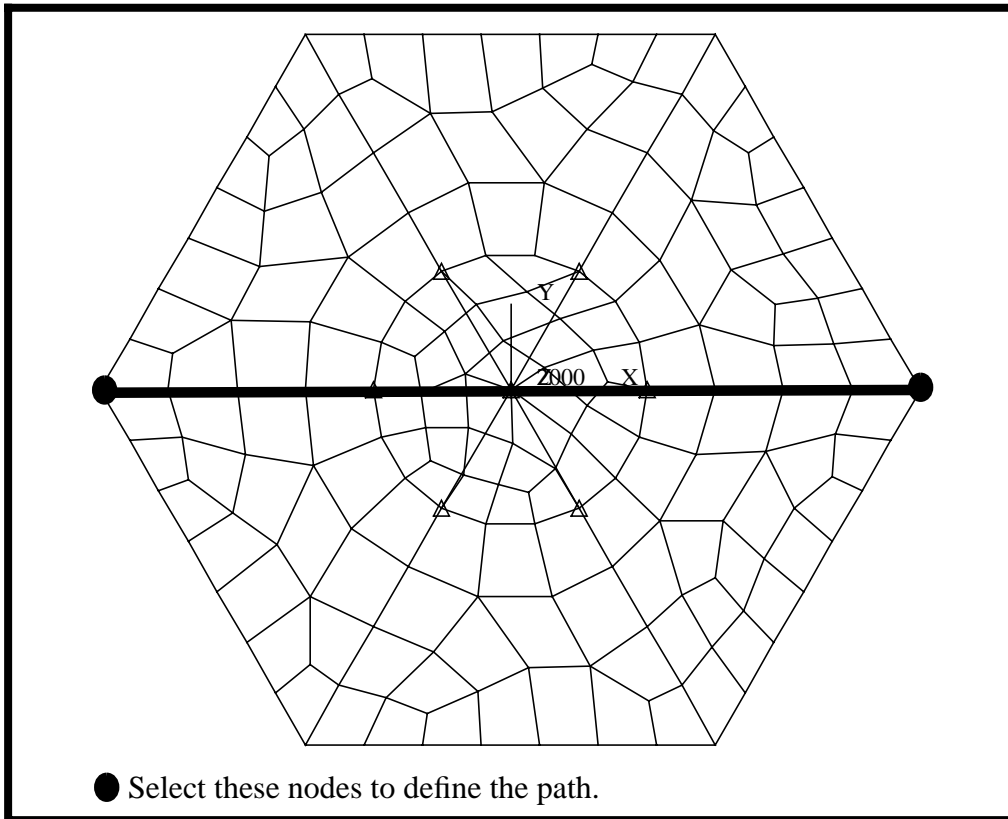
First use the Main Form to reset the graphics



Reset graphics

We will be looking at an XY Plot of the mid-span stresses of the upper platform as shown in the figure below

Figure 7.12. Path Location to Compute the Stresses Across Upper Platform.



Now create an XY plot for the von Mises stresses taken across a horizontal slice of the model as shown in the figure above.

Action:

Create

Object:

Graph

Method:

Y vs X

Select Result Case(s)

**LAUNCH STATIC, Static
Subcase**

Y:

Result

Select Y Result

Stress Tensor

Position...(At Z1...)

Positions

At Center

At Point C

At Point D

At Point E

At Point F

At Z1

At Z2

Option:

Maximum

Close

Quantity:

von Mises

X:

Path Length

You will now use Target Entities to define a path.



Target Entities

Select Path Points

Select vertex nodes shown in the figure above.

Points Per Segment

25

Again, modify the display attributes to customize the Fringe Display.



Display Attributes

You can customize the display of the Graph using Display Attributes. Make sure to turn on the symbol and change the curve fit to spline. Feel free to also modify the X and Y axis definition and format.

Curve Fit:

Spline

Show Symbol

Verify the Plot Options:



Plot Options

Coordinate Transformation:

None

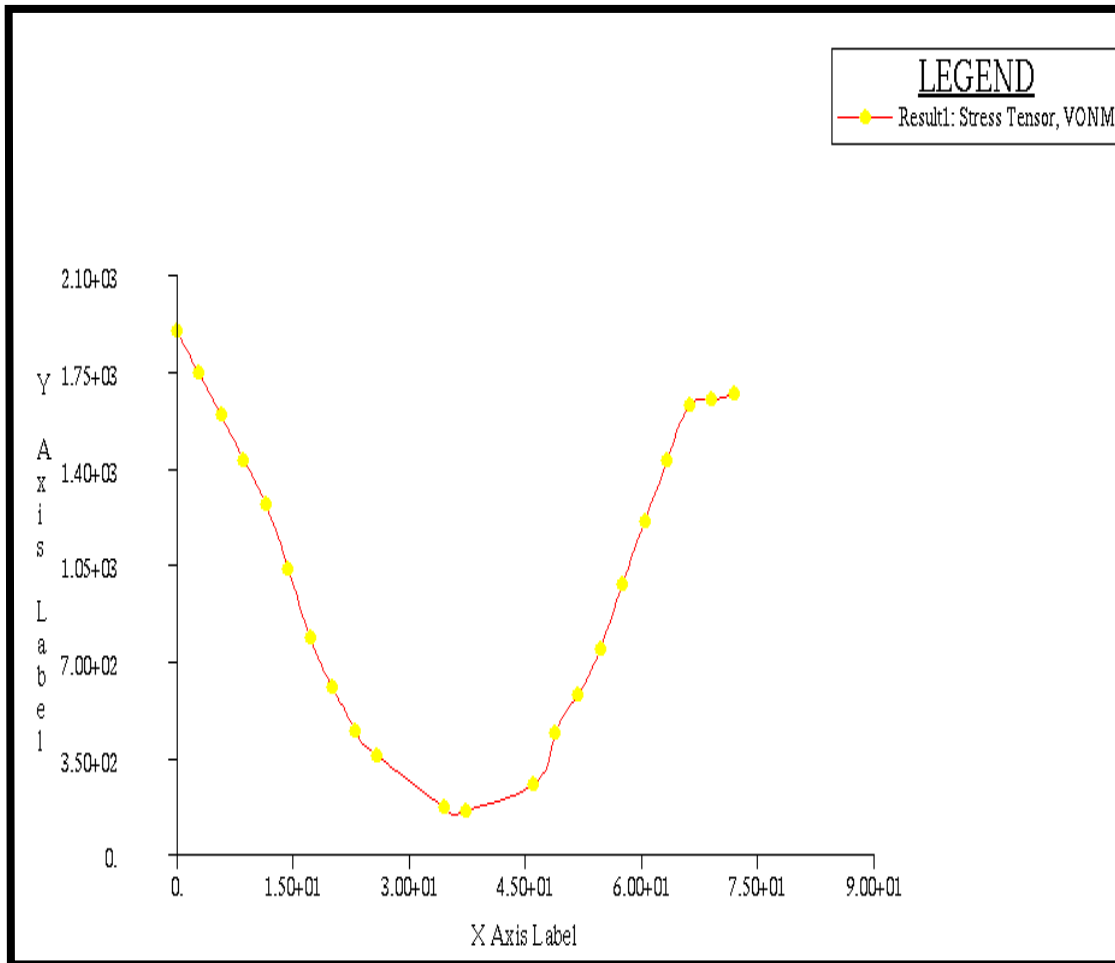
Averaging Definition/Domain:

All Entities

Apply

This will create an XY Graph of your the model as shown in the figure below

Figure 7.13. Upper Platform Mid Span von Mises Stresses.



**Create a
Group for the
Navigational
Platform
Support
Structure**

17. Create a group of the bar elements that represent the support structure for the navigational platform.

First use the Main Form to reset the graphics.



Reset graphics

Verify the view is set to a top view using the Toolbar Icons



Front View

Now, we will create a group called Nav Support Struct which will only include the bar elements.

Group/Create...

New Group Name

Nav Support Struct

Make Current

Unpost All Other Groups

We will want to make a group containing only the bar2 type elements. You will use the Select Menu to isolate only these element. From the Select Menu, select FEM Entities --> then Elements --> then Bar elements



**Select Menu:
FEM Entity**



**Select Menu:
Element**



**Select Menu:
Beam Element**

Entity Selection

Select all posted entities. With the select menu set, only bars will be selected

Apply

Cancel

18. Create a Freebody plot of the loads for the Navigational Platform Support Structure.

In this step, we will use the Freebody tool to investigate the Freebody Loads for the Navigational Platform Support Structure. The original representation of this model and support structure was fairly coarse. This is because this model represents a “Global” primary structure model. Much of the detail about the satellite panels, such as the stiffeners, was assumed to be smeared throughout the panel. The loads generated for this model will be used to help design these components and perform detailed analysis of the panels and substructure. We will now use the Freebody tool to obtain the loads which can be used in sizing the Navigational Platform Support Structure and it’s joints.

Create a Freebody Plot for the Support Structure

Lets first reorient the model using the Toolbar



Isometric View 3

Action:

Create

Object:

Freebody

Method:

Loads

Select Result Case(s)

LAUNCH STATIC

Select Fringe Result

Freebody Loads

Apply

This will create a Freebody with component vectors of the loads at each end of the bar elements. To further inspect the loads on a node by node basis, we will choose the Show Spreadsheet results icon:



Show Spreadsheet

This will provide you with each of the force components as well as the summation of all the loads.

19. To complete this exercise, you will close the database.

File/Quit

This will exit MSC/PATRAN and close your file.

**Close the
Database
and Quit
Patran**

