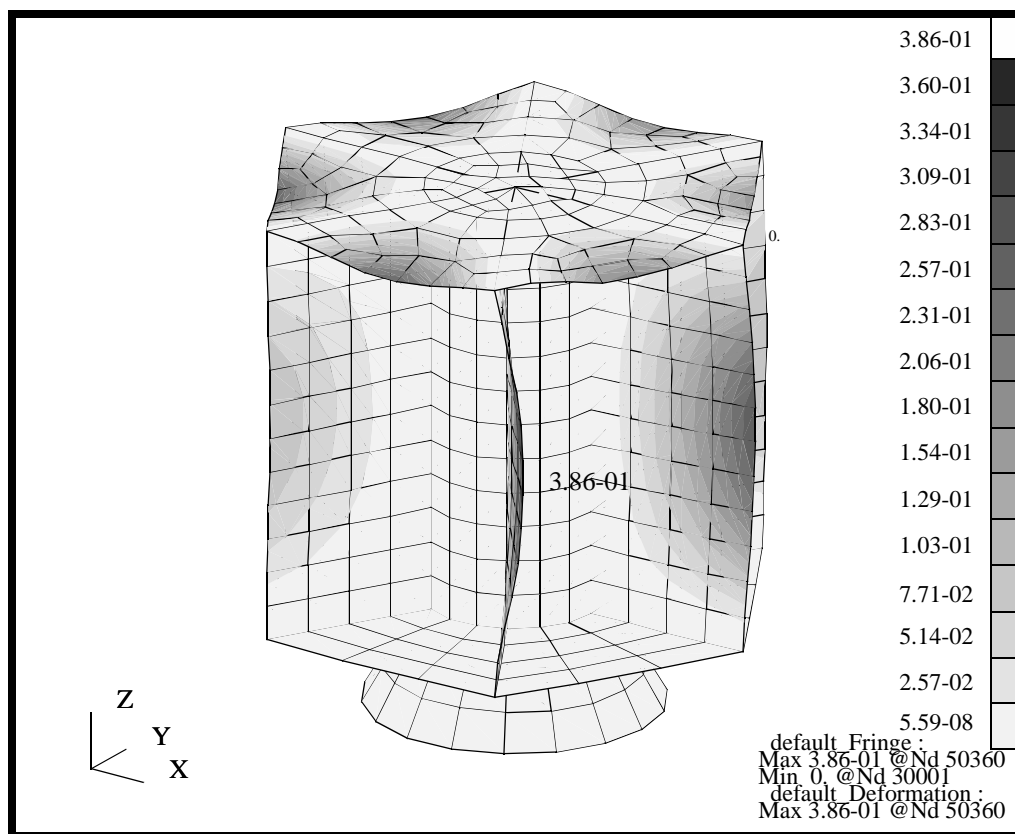


## Exercise 8

# *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 Bar 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 titles, results labels and scale factors. Finally, you will create a graph of stress versus path for von Mises stresses across the Upper platform.

## Suggested Exercise Steps:

- Start MSC.Patran 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 change the 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 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 MSC.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*

**New Model Preference**

*Tolerance*  **Default**

*Analysis Code*

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:*

*Object:*

*Method:*

*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



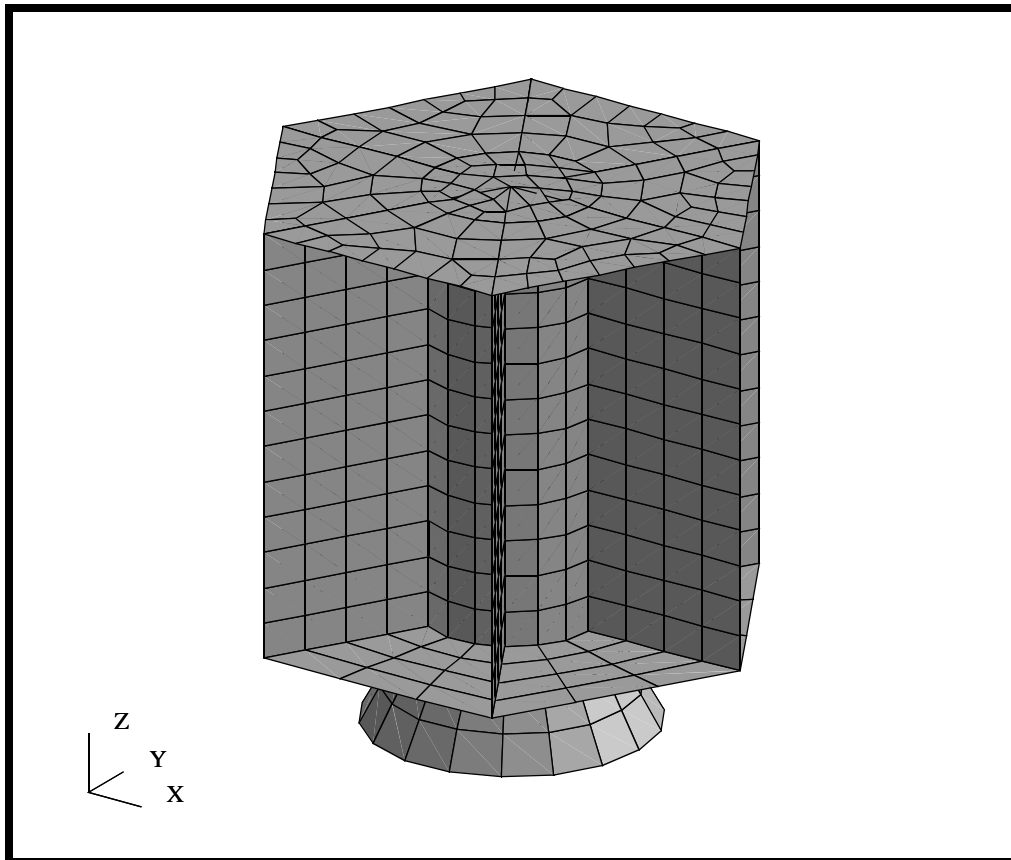
**Isometric View 3**



**Smooth Shaded**

Your model should look like the figure below.

**Figure 8.1.** 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*

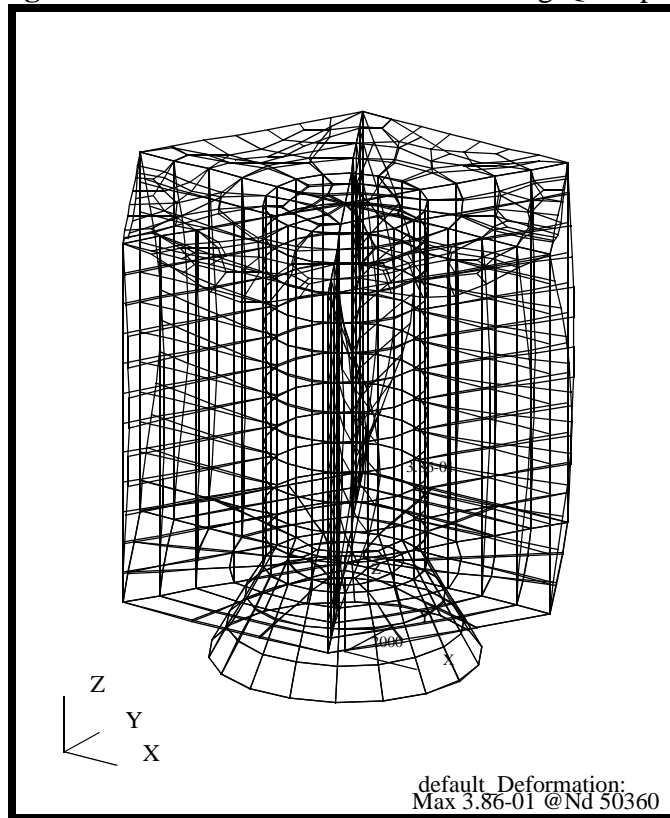
**Displacements, Translational**

**Apply**

---

Your model should look like the figure below.

**Figure 8.2.** 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

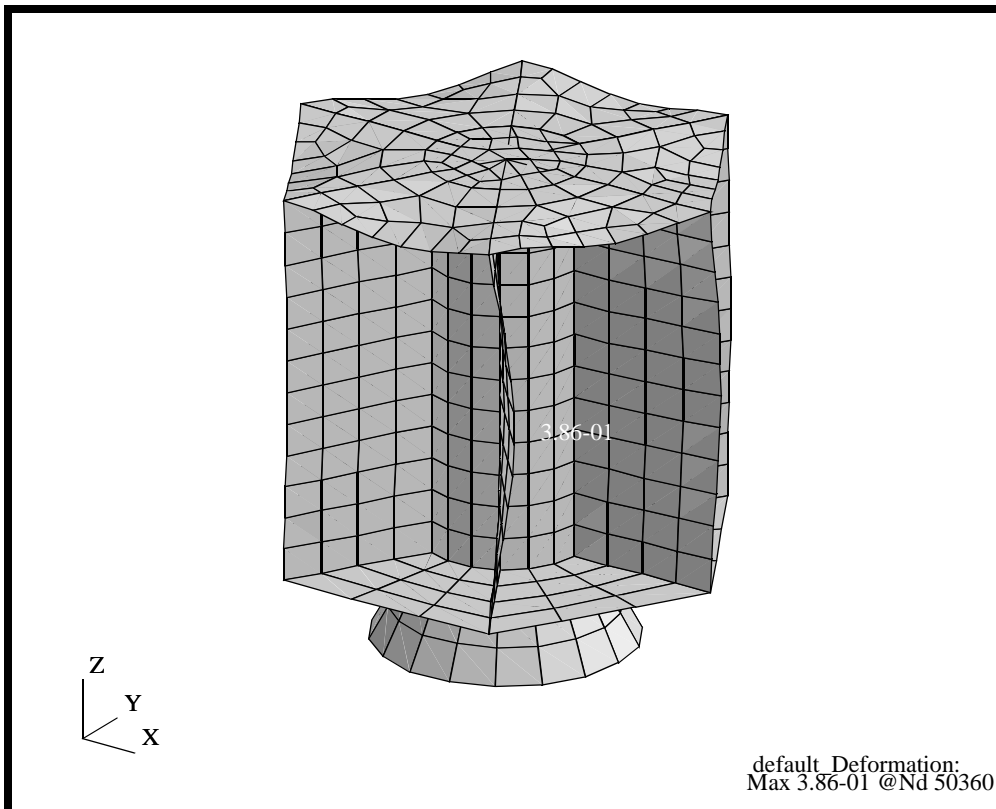
0.05

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 8.3.** 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 by 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:

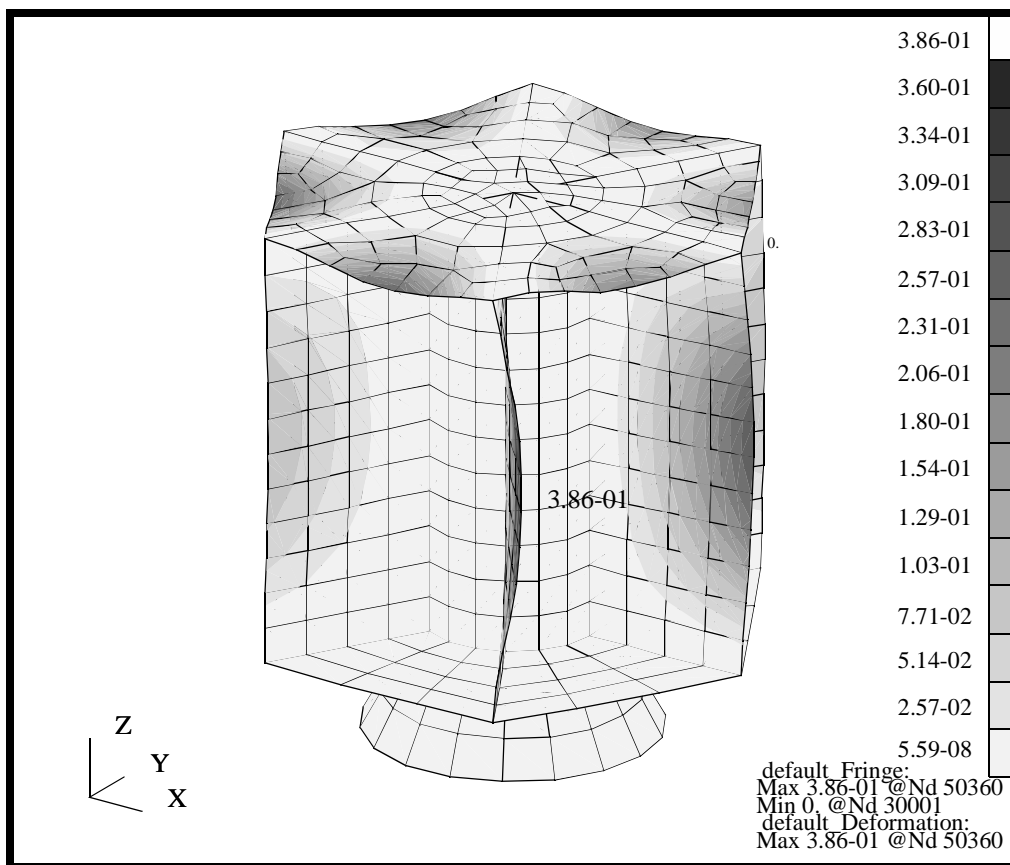
Element Edges

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 8.4.** 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 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>	<b>Create</b>
<i>Object:</i>	<b>Quick Plot</b>
<i>Select Result Cases</i>	<b>LAUNCH STATIC, Static Subcase</b>
<i>Select Fringe Result</i>	<b>Displacements, Translational</b>
<i>Quantity</i>	<b>Magnitude</b>
<i>Select Deformation Result</i>	<b>Displacements, Translational</b>

**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>	<b>◆ Modal</b>
<i>Animation Graphics</i>	<b>◆ 3D</b>
<i>Number of Frames</i>	<b>10</b>
<b>Apply</b>	

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.

<i>Action:</i>	<b>Create</b>
<i>Object:</i>	<b>Marker</b>
<i>Method:</i>	<b>Vector</b>
<i>Select Result Case(s)</i>	<b>LAUNCH STATIC, Static Subcase</b>

Select Vector Result

Constraint Forces,  
Translational

Show As:

Component

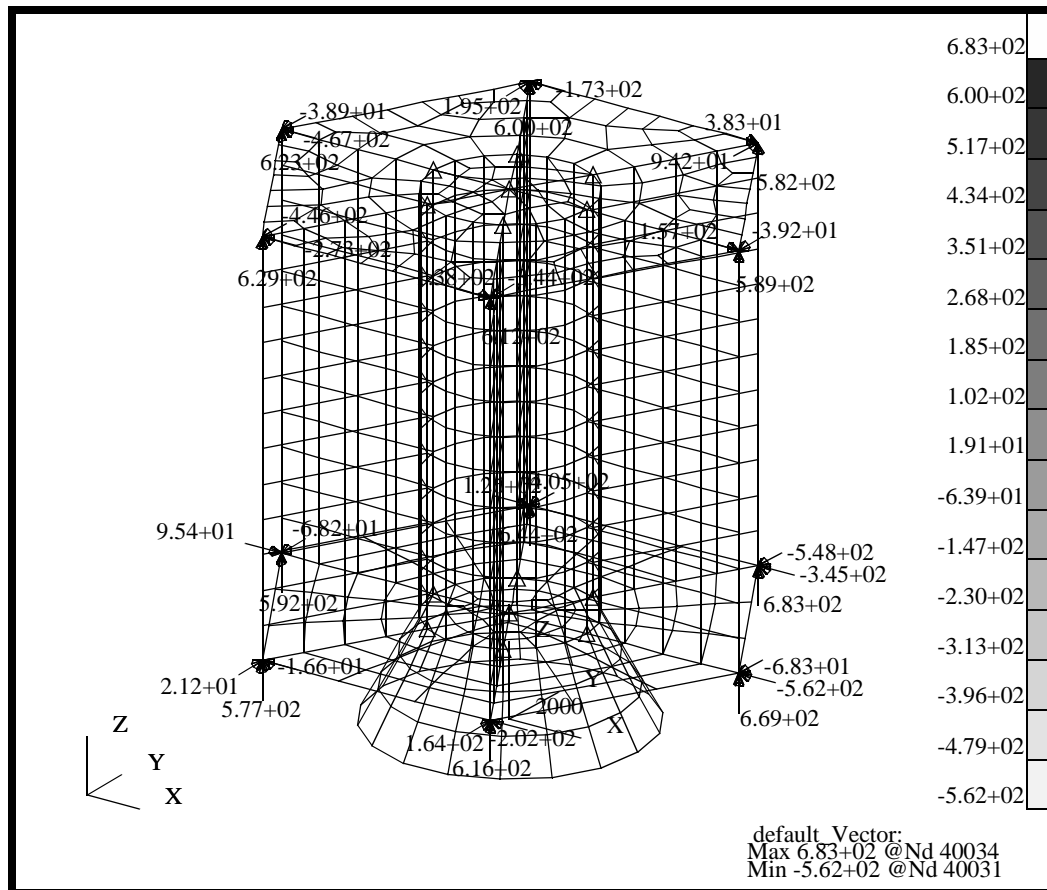
XX  YY  ZZ

Animate

Apply

Your model should now look like the figure shown below.

Figure 8.5. Vector Plot of the Reaction Forces for the Launch Condition.



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. 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 Fringe Plot of the Bar Axial Stresses**



**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="Bar Stresses, Bending"/>
<input type="text" value="Position...(At Center)"/>	
<i>Positions</i>	<input type="text" value="At Point C&lt;br/&gt;At Point D&lt;br/&gt;At Point E&lt;br/&gt;At Point F"/>
<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:*

<b>Element Types</b>
----------------------

*Select Element Types*

<b>Point</b>
<b>Bar 2</b>
<b>Quad 4</b>

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:*

<b>Discrete/Smooth</b>
------------------------

*Shading:*

<b>None</b>
-------------

*Style:*



(Straight continuous line)

*Width:*



(Widest line width)

Finally, give the plot a name using the Plot Options



### Plot Options

*Save Fringe Plot As:*

<b>bar_xx_stress</b>
----------------------

<b>Apply</b>
--------------

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

Select Result Case(s)	LAUNCH STATIC, Static Subcase
Select Fringe Result	Stress Tensor
Position...(At Point C)	
Positions	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

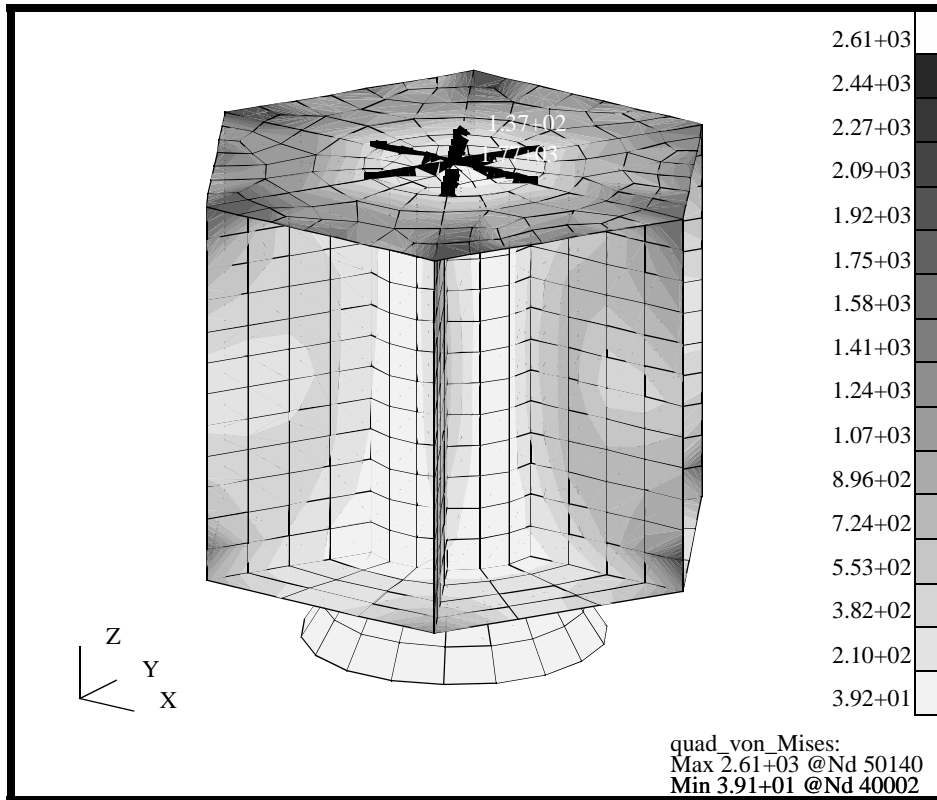
*Save Fringe Plot As:*

**shell\_von\_Mises**

**Apply**

Your model should now look like the figure below

**Figure 8.6.** 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*

<b>Post</b>
<b>Plots</b>
<b>DEF_default_Deformation</b>
<b>FRI_default_Fringe</b>
<b>VEC_default_Vector</b>
<b>FRI_bar_xx_stress</b>
<b>FRI_shell_von_Mises</b>

**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.

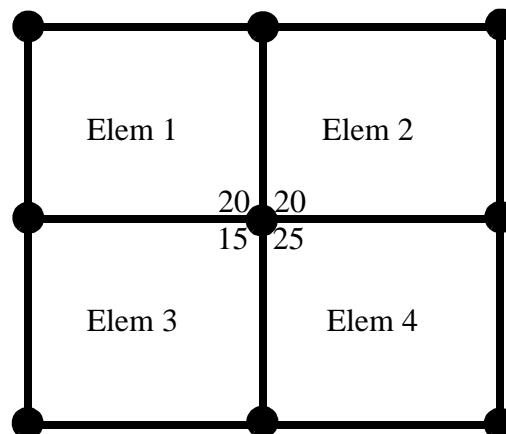
---

**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:

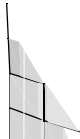
None

Averaging Definition/Method:

Derive/Average

Apply

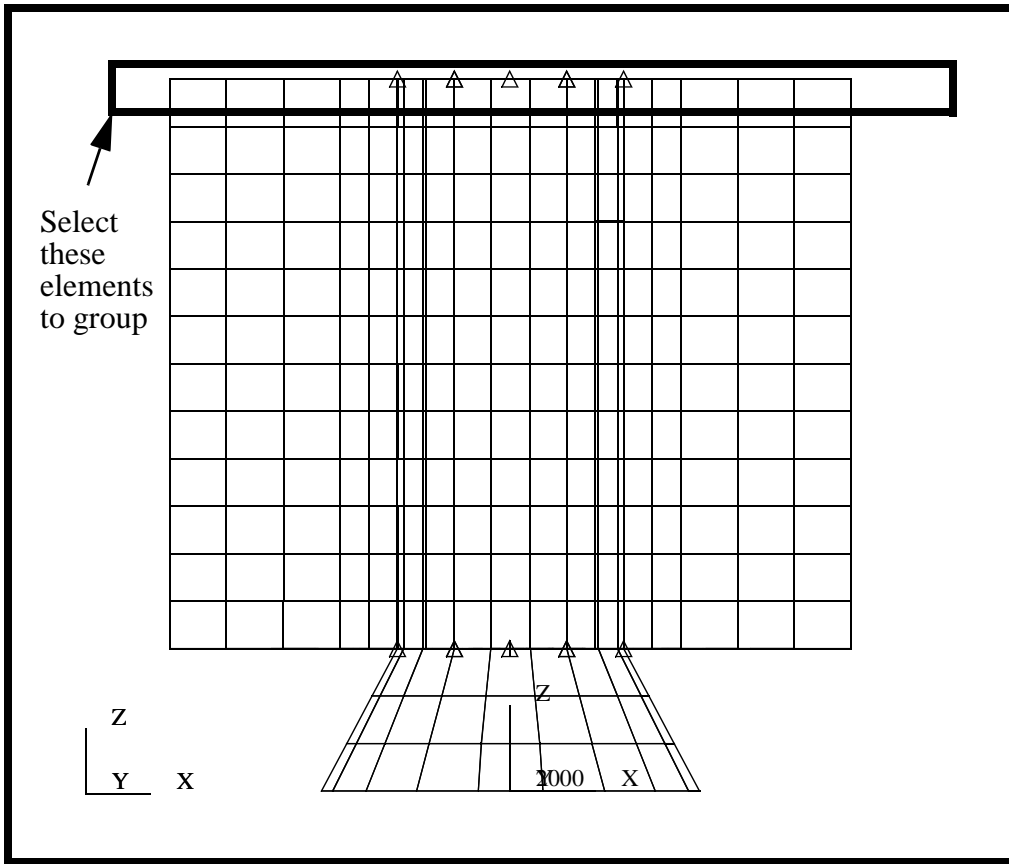
Your display should  
will indicate the lo



x

Your display should be similar to the one shown below.

**Figure 8.8.** 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**

---

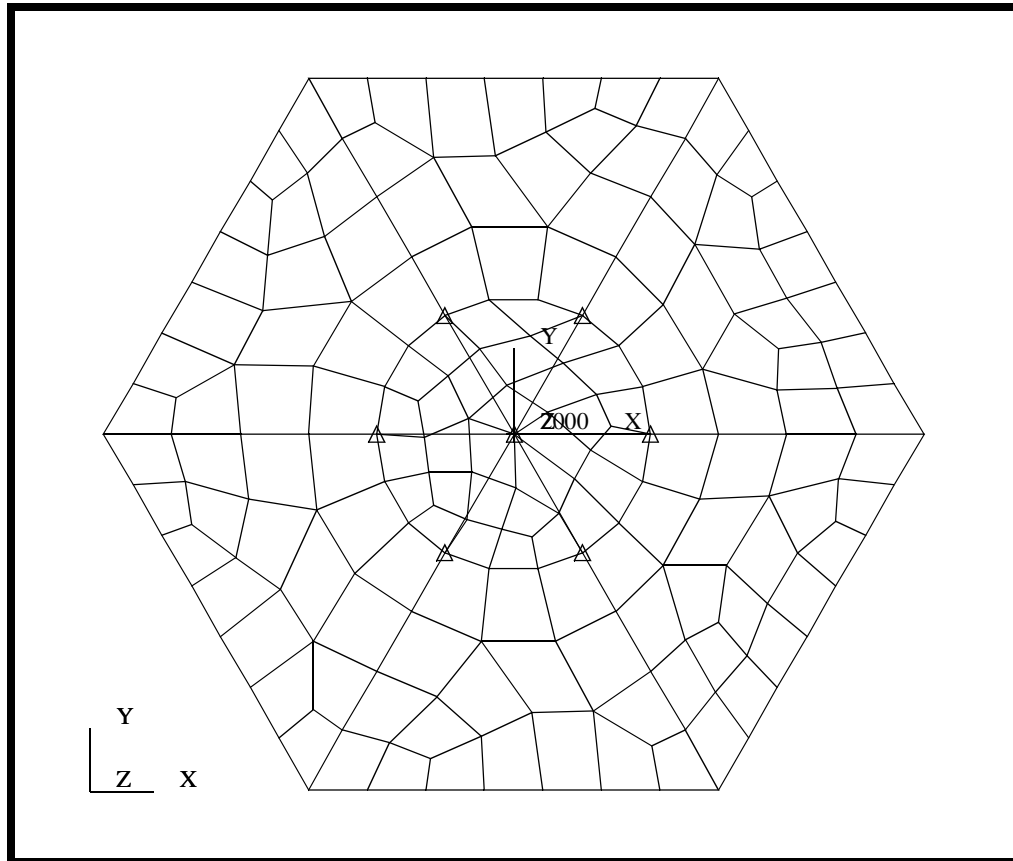
Change the view to a top view using the Toolbar Icons



**Front View**

Your model should look like the figure below.

**Figure 8.9.** 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.

## Post-Processing Static Results of a Satellite

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 Tensor Result</i>	<input type="text" value="Stress Tensor"/>
<input type="text" value="Position...(At Z1...)"/>	
<i>Positions</i>	<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>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"/> <b>Animate</b>	

Again, change the target entities.



Now select the Quad4 entities.

<i>Target Entity:</i>	<b>Target Entities</b>
<i>Select Element Types</i>	<input type="text" value="Element Types"/>
	<input type="text" value="Point"/> <input type="text" value="Bar 2"/> <input checked="" type="text" value="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**

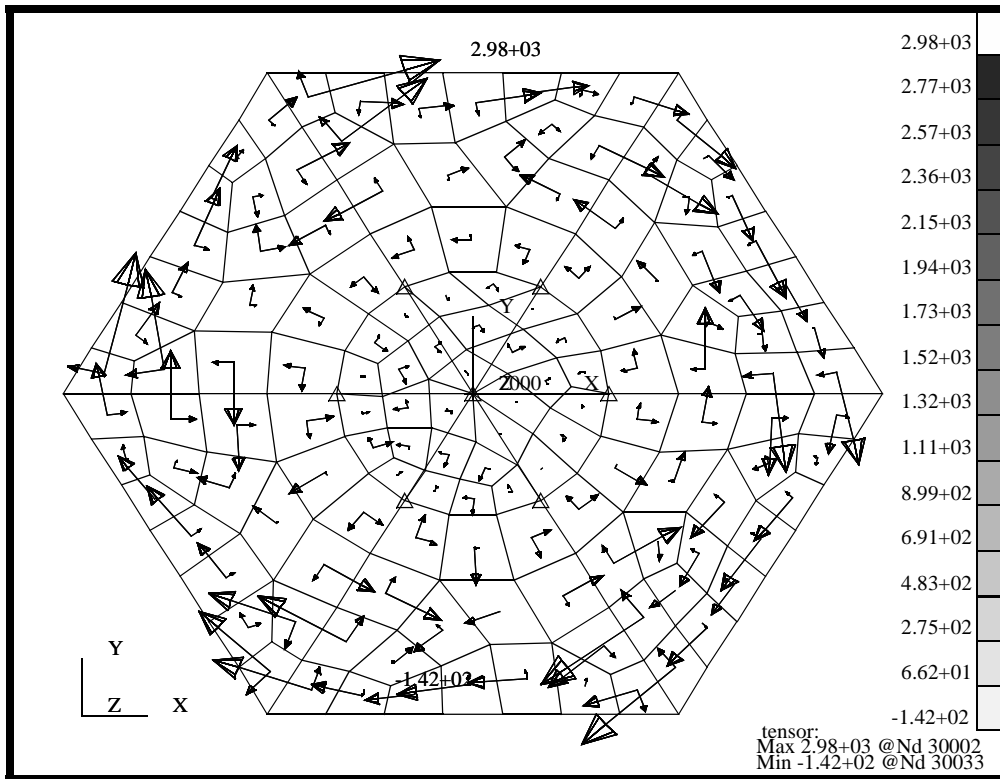
##### Show Tensor Box

##### 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 8.10.** 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**

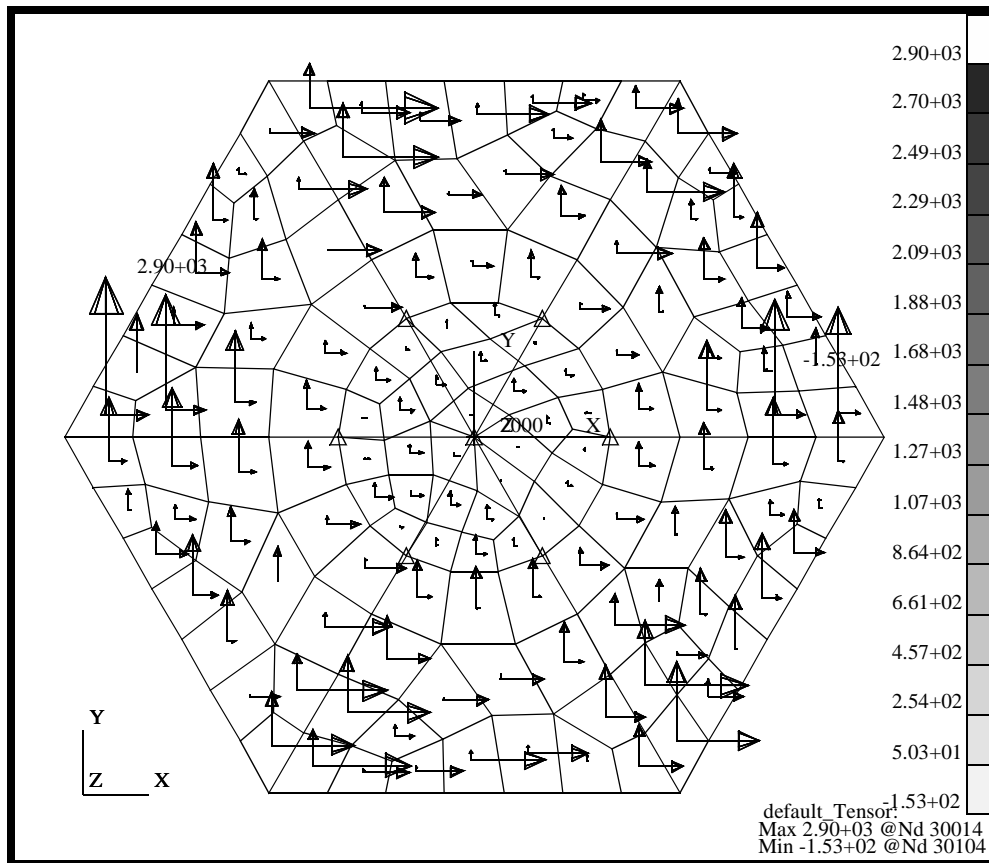
Coordinate Transformation:

**Global**

**Apply**

Your model should now look like the figure below.

**Figure 8.11.** 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. 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 is aligned 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 a two directional stress is not aligned and 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.

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

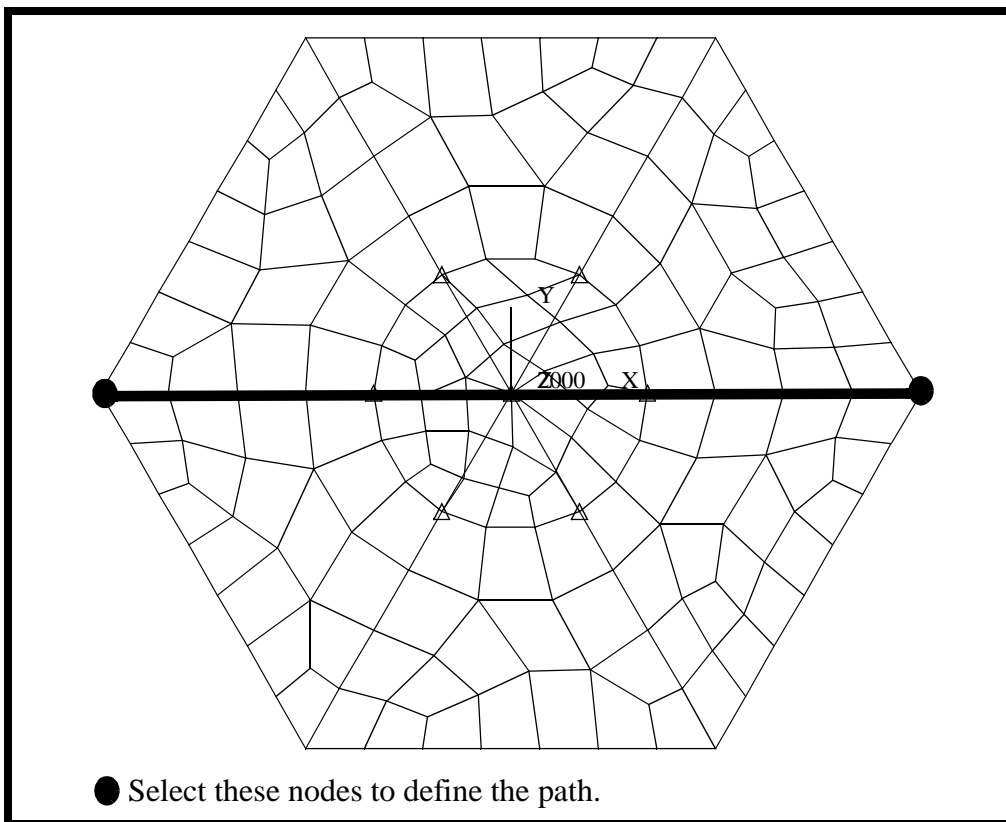
**Create an XY  
Plot of the  
Stresses**



**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 8.12.** 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.

<i>Action:</i>	<b>Create</b>
<i>Object:</i>	<b>Graph</b>
<i>Method:</i>	<b>Y vs X</b>
<i>Select Result Case(s)</i>	<b>LAUNCH STATIC, Static Subcase</b>
<i>Y:</i>	<b>Result</b>
<i>Select Y Result</i>	<b>Stress Tensor</b>
<b>Position...(At Z1...)</b>	
<i>Positions</i>	<b>At Center</b> <b>At Point C</b> <b>At Point D</b> <b>At Point E</b> <b>At Point F</b> <b>At Z1</b> <b>At Z2</b>
<i>Option:</i>	<b>Maximum</b>
<b>Close</b>	
<i>Quantity:</i>	<b>von Mises</b>
<i>X:</i>	<b>Path Length</b>

You will now use Target Entities to define a path.



### Target Entities

<i>Select Path Points</i>	Select vertex nodes shown in the figure above. 
<i>Points Per Segment</i>	<b>25</b>

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:*

**Show Symbol**

Verify the Plot Options:

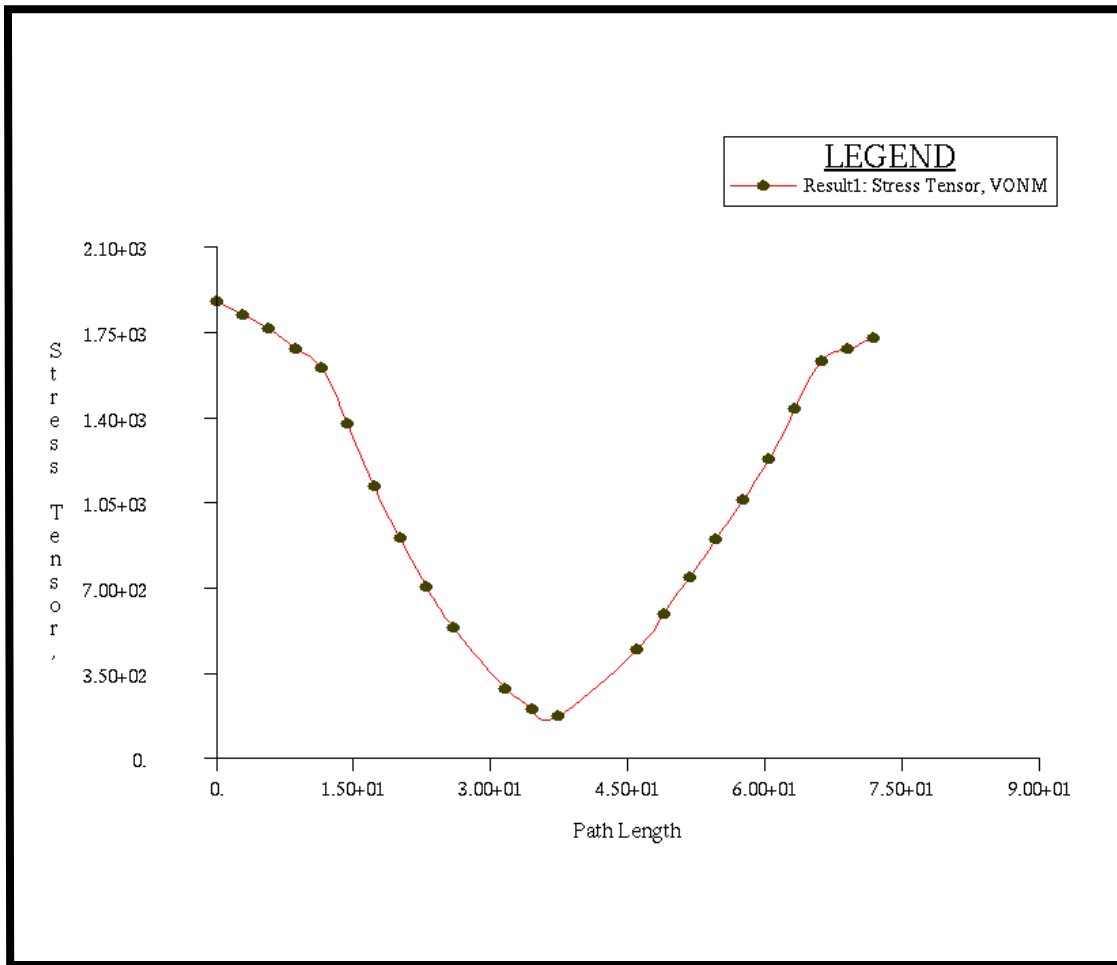
**Plot Options**

*Coordinate Transformation:*

*Averaging Definition/Domain:*

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

**Figure 8.13.** 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**

## Create a Freebody Plot for the Support Structure

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.

Lets first reorient the model using the Toolbar .



### Isometric View 3

*Action:*

**Create**

*Object:*

**Freebody**

*Method:*

**Loads**

*Select Result Case(s)*

**LAUNCH STATIC**

*Select Result Type*

**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  
MSC.Patran**

