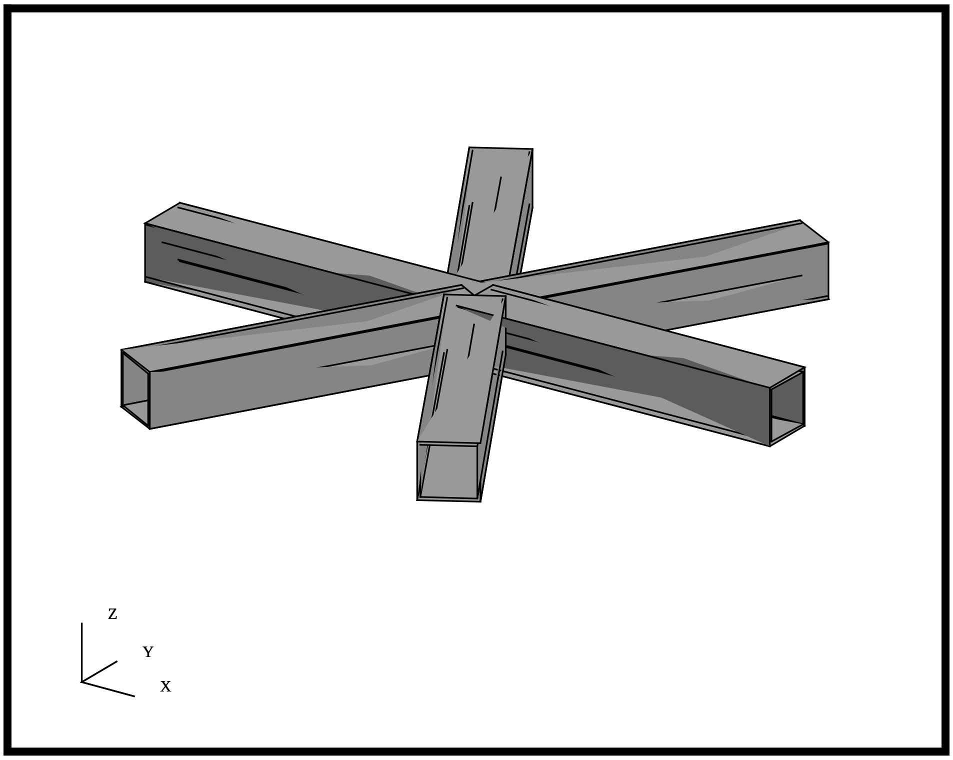
LESSON 5

Element Property Definition for the Space Satellite



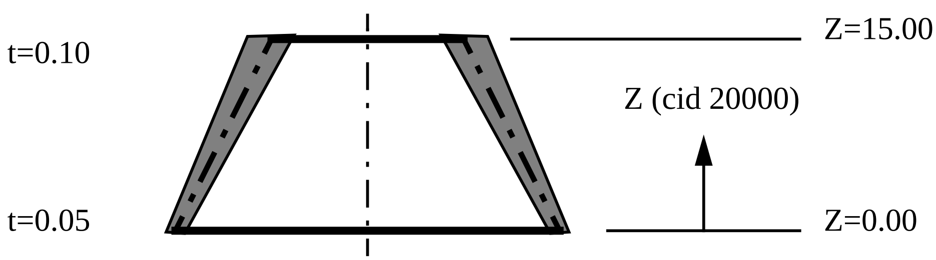
Objectives:

- Define shell, bar, and point mass element properties for the Space Satellite.
- Create and apply a **Field** to describe a spatially varying thickness for the Adapter Cone.
- Verify Mass Properties for the structure.

Model Description:

In this exercise, you will create Element Properties for the Space Satellite. These Properties will include properties for 2-D shell elements, 1-D bars, and 0-D concentrate masses. Additionally, we will be creating a field to define a thickness variation in the Adapter Cone. The thickness of the cone will be 0.05 at its base and will vary linearly to a thickness of 0.10 at its top dimension.

Figure 5.1. Adapter Cone's specifications - 2D slice



thickness(Z) = 0.05 + 0.05 * Z / 15.0

t(Z=0) = 0.05
t(Z=15) = 0.10

We will also be controlling the Property ID's. From within PATRAN, you can define the ID of an Element Property or Material by defining a trailing number. If this number is not already defined for another property or material that number will be used. The Property assignments are listed in the table below.

PID #	Component	Type	Units
10000	Central Cylinder	2D Shell	0.250 in
20000	Adapter	2D Shell	field
30000	Upper Platform	2D Shell	0.100 in
39000	Navigational Platform	0D Mass	40 lbs

39100	Navigational Platform Support	1D Bar	Hollow Box 2” x 2” t=0.1 in
40000	Lower Platform	2D Shell	0.350 in
49000	Propulsion Block	0D Mass	60 lbs
50000	Shear Panels	2D	0.125 in
59000	Science Platforms	0D Mass	20 lbs

Suggested Exercise Steps:

- Start MSC/PATRAN and open the **`satelite.db`** file.
- Use a PCL Function to define a spatially varying field named **Thickness_variation**, using the equation described in Figure 5.1. Also make sure the definition is based on CID 20000.
- Post and define each of the 2D Shell Element Properties that listed in the table above.
- Post and define each of the 1D Beam element properties that listed in the table above.
- Post and define each of the 0D Mass element properties that listed in the table above.
- Display the Mass property and compare with the Mass hand calculation.
- Close and Exit MSC/PATRAN.

Exercise Procedure:

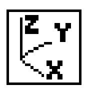
*Note: In most MSC/PATRAN forms, the default setting for the **Auto Execute** button is on; thus, you do not need to press **Apply.***

1. Start MSC/PATRAN and open the `satelite.db` file.

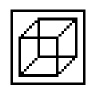
File/Open...

Existing Database Name	`satelite.db`
OK	

2. Before we get started, let's define a common render style and view using the Toolbar icons.



Isometric View 3



Wireframe

Fields to Define a Spatially Varying Load

3. Create a spatially varying scalar field named **thickness_variation** that will later be used to define the Adapter Cone's thickness. Use the scalar field described in Figure 5.1.

◆ **Fields**

Action:	**Create**
Object:	**Spatial**
Method:	**PCL Function**
Field Name	**thickness_variation**
Field Type	◆ **Scalar**
Coordinate System Type	◆ **Real**
Coordinate System	**Coord 20000**
Scalar Function	**0.05+0.05*'Z/15.0**

Note: Remember to precede the independent variable, capital Z, with a single quote.

Apply

Verify the Field

4. Lets verify the thickness variation we just defined.

Action:	**Show**
Select Field to Show	**thickness_variation**

Specify Range...

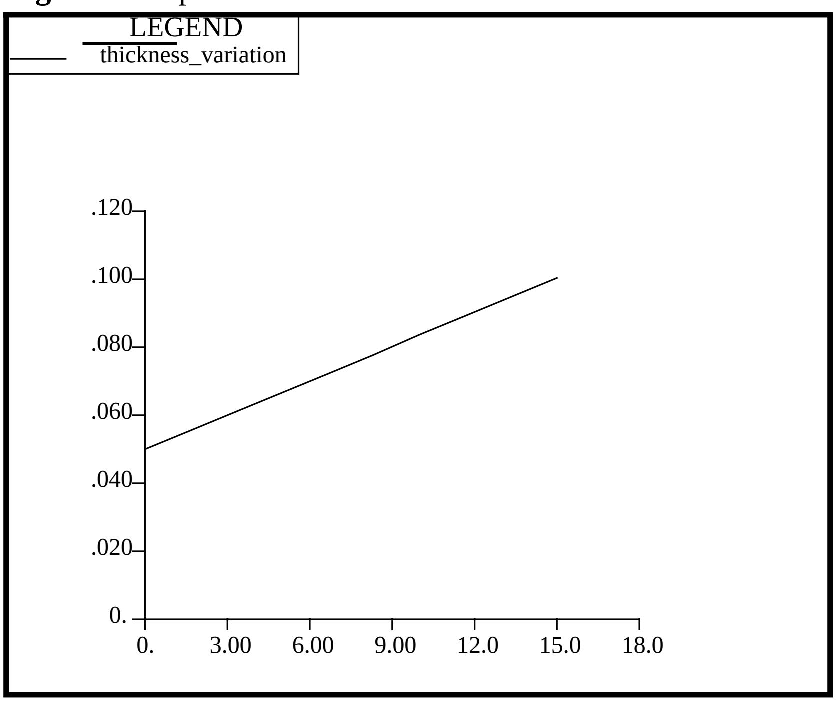
Minimum	**0.0**
Maximum	**15.0**
Number of Points	**10**

OK

Apply

You will now get an XY Plot of the data as well as a Table defining the evaluated values. The XY Plot should look like the figure below.

Figure 5.2. Spatial Field Variation



To unpost the XY Window, select the following button on the Fields Form.

Unpost Current XY Window

Plotted Curves window **Cancel**

Defining Shell Elements

5. In this section, we will create properties for all 2D Shell elements. To do this, we will post each group, then define the appropriate element properties. The 2D Shell element properties are given in the table below. It should also be noted, by ending the name with an ID, that ID will be used when the model is written out.

Property Name	Material	Thickness
Central Cylinder.10000	Aluminum	0.250 in
Adapter.20000	Titanium	field
Upper Platform.30000	Aluminum	0.100 in
Lower Platform.40000	Aluminum	0.350 in
Shear Panels.50000	Aluminum	0.125 in

Let's start by posting the Central Cylinder group.

Group/Post...

Select Groups to Post

Adapter
All Fem
All Geometry
Central Cylinder
Lower Platform
Navigational Platform
Propulsion Block
Science Platforms
Shear Panels
Upper Platform
default_group

Apply

Cancel

Now, let's define the element property for the Central Cylinder. The Central Cylinder will have an ID of 10,000, thus the name Central Cylinder.10000. Additionally, it is made of aluminum and a 0.25 inches thick.

◆ **Properties**

Action:	**Create**
Dimension:	**2D**
Type:	**Shell**
Property Set Name	**Central Cylinder.10000**

Input Properties...

In the *Input Properties...* form, click in the *Material Name* data box. The material properties available for selection will appear in the Material Property Sets list.

Click on **aluminum** under the *Material Property Sets* box. The selected material name will appear with the prefix "**m:**" in the *Material Name* data box.

Material Name	**m:aluminum**
Thickness	**0.250**

OK

Select Members	Select all posted surfaces

Add

Apply

Repeat this process, posting the group then assigning the element properties, for the Upper Platform, Lower Platform and Shear Panels. DO NOT define a Property for the Adapter at this time. Note the use of the property numbering encoded syntax.

Applying a Spatially Varying Thickness

6. Now, we will define the element properties for the Adapter. The Adapter has a variable thickness. We will use the previously create field, Thickness_variation, to define this variation.

Let's start by posting the Adapter group.

Group/Post...

Select Groups to Post

Adapter
All Fem
All Geometry
Central Cylinder
Lower Platform
Navigational Platform
Propulsion Block
Science Platforms
Shear Panels
Upper Platform
default_group

Apply

Cancel

Now, let's define the element property for the Adapter. The Adapter will have an ID of 20,000, thus the name Adapter.20000. Additionally, it is made of titanium and we will use a field to define its thickness.

Action:	**Create**
Dimension:	**2D**
Type:	**Shell**
Property Set Name	**Adapter.20000**

Input Properties...

In the *Input Properties...* form, click in the *Material Name* data box. The material properties available for selection will appear in the Material Property Sets list.

Click on **titanium** under the *Material Property Sets* box. The selected material name will appear with the prefix "**m:**" in the *Material Name* data box.

Material Name **m:titanium**

In a similar method we used to define the material, we will define the thickness field for the Adapter. Click in the *Thickness* data box. The fields available for selection will appear in the Field Definitions list.

Select the thickness_variation field from the list. The selected material name will appear with the prefix “**f:**” in the *Thickness* data box indicating a spatial varying field has been applied.

Thickness	**f:thickness_variation**
OK	
Select Members	Select all posted surfaces
Add	
Apply	

7. Verify the thickness variation on the Adapter Cone.

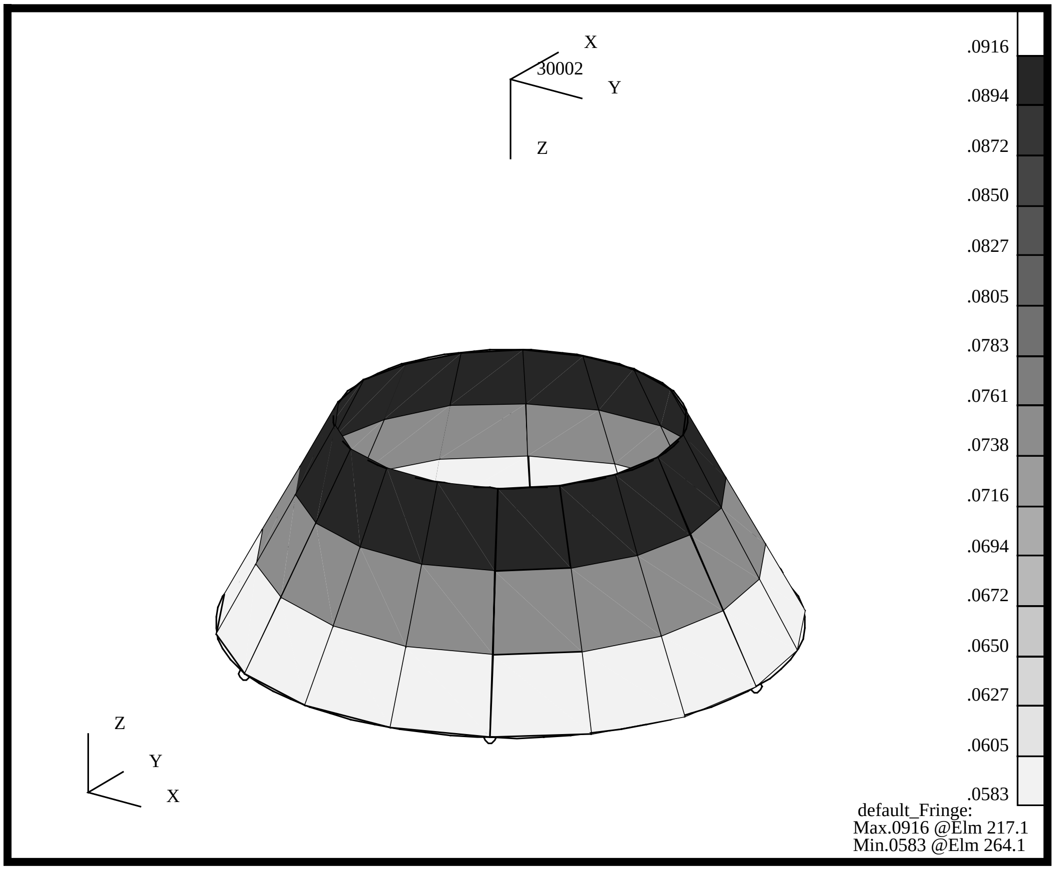
Verify the Spatially Varying Thickness

We will now use the Properties Show utilities to verify the spatially varying thickness of the Adapter Cone.

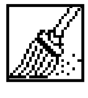
Action:	**Show**
Existing Properties	**Definition of XY Plane** **Mass** **Material Name** **Property Set Name** **Section Name** **Thickness**
Display Method	**Scalar Plot**
Select Groups	**Adapter**
Apply	

This will create a element fill plot of the Adapter Cone thickness. You will notice each element is uniform. Your model should look like the figure below.

Figure 5.3. Thickness Variation as Applied to the Adapter Cone



Property show is a good tool to verify various property assignments, including, bar XY plane definition, material assignment, field assignment, etc. All of the verification checks can be displayed in a number of graphical as well as tabular forms. Before we proceed, lets clear the plot by using reset graphics from the Main Form.



Reset graphics

Create 1-D Bar Properties

8. Create the Bar properties for the Navigational Platform. We will use a PBARL type MSC/NASTRAN element property and define a box section that is 2 x 2 with a thickness of 0.10. Also, the navigational platform will have a PID of 39100.

Let's first post the Navigational Platform.

Group/Post...

Select Groups to Post

Adapter
All Fem
All Geometry
Central Cylinder
Lower Platform
Navigational Platform
Propulsion Block
Science Platforms
Shear Panels
Upper Platform
default_group

Apply

Cancel

Now lets define the element property for the Navigational Platform support structure. The structure is modeled with 1-D beams which has a 2x2 box cross section and a wall thickness of 0.1 inches.

◆ **Properties**

Action:	**Create**
Object:	**1D**
Type:	**Beam**
Property Set Name	**np_support.39100**
Options	**General Section**
	Standard Formulation

Input Properties...

In the *Input Properties...* form, click in the *Material Name* data box. The material properties available for selection will appear in the Material Property Sets list.

Click on **aluminum** under the *Material Property Sets* box. The selected material name will appear with the prefix "**m:**" in the *Material Name* data box.

Material Name	**m:aluminum**

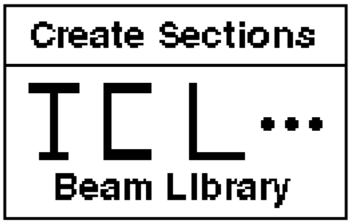
Now change the options menu on the right side of the Section Name window from Properties to dimensions. This will tell MSC/PATRAN we are defining a MSC/NASTRAN Section using section Properties by dimensions, intern, this will write out a PBARL card.

[Section Name] **Dimensions**

Finally, we will define the Bar orientation, which is in +Z direction

Bar Orientation **<0, 0, 1>**

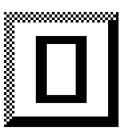
We will now define a box beam section using the beams dimensions. Select the following Icon to open the Beam Library.



Define a standard hollow box section using one of the standard shapes.

Action:	**Create**
Type:	**Standard Shape**
New Section Name	**box**

Now select the following icon form the list of beam sections to define a hollow box section.



Hollow Box Section

Define the dimensions for the section, 2 x 2, t=0.1.

W	**2.0**
H	**2.0**
t1	**0.1**

t2	**0.1**

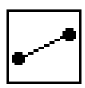
Let's use the Calculate/Display functionality to see the section properties and figure.

Calculate/Display

A Section Display window should have now appeared on the screen showing you the section and the relevant section properties. Once you have reviewed the information, you can close the form and apply the property.

Section Display Window	**Close**
Beam Library	**OK**
Input Properties Window	**OK**

Let's define the Application Region. We want to assign these properties to the bar elements since there is no available geometry. First, click in the Select Members databox, the Select Menu will then be displayed. Select the Beam Element icon from the Select Menu.



Select Menu:
Beam element

Finally, select the All beam elements in the Viewport and create the property.

Select Members	Select all displayed beam elements

Add

Apply

Displaying the Beam Sections

9. Now, let's turn on the Display for the Beams so we can visually verify the section we have just created.

Before we get started with this step, turn off the coordinate frame display.

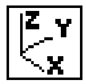
Display/Coordinate Frames...

Post/Unpost Coord. Frame(s)	Select Coordinate Frames 20000 and 30000

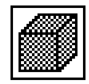
UnPost

OK

Make the following changes to your viewport using the following Toolbar icons.



Isometric View 3



Shaded Smooth

Finally, let’s display the sections using full section display.

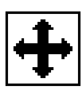
Display/Load/BC/Elem. Props...

Beam Display **3D:Full-Span**

Apply

Cancel

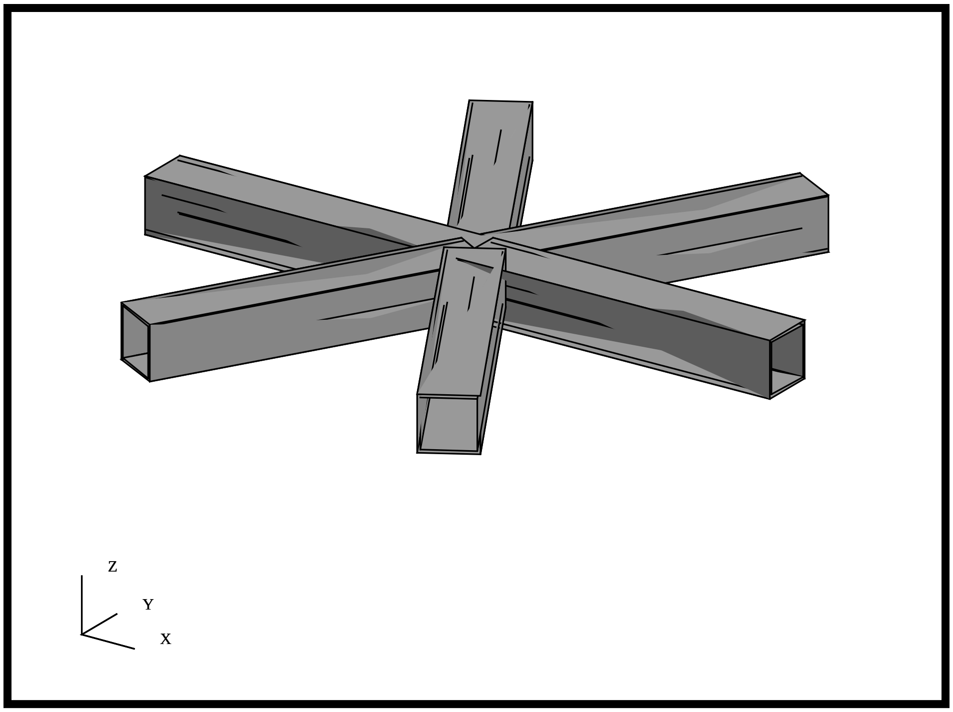
You may need to select Fit View from the Toolbar to properly see the beam sections.



Fit view

Your viewport should be similar to the figure below.

Figure 5.4. Hollow rectangular sections representing the Navigational Platform mounting structure as displayed in MSC/PATRAN



Defining Concentrated Masses

10. We will now turn our attention to defining the last of our properties, the mass elements. These will be defined as 0D element properties.

Again, each of the mass elements have a specific numbering requirements, as follows:

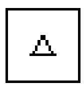
Property Set Name	Component	Type	Mass
np_mass.39000	Navigational Platform	Lumped Mass	40 lbs
pb_mass.49000	Propulsion Block	Lumped Mass	60 lbs
sp_mass.59000	Science Platforms	Lumped Mass	20 lbs

We will start with the Navigational Platform concentrated mass.

Action:	**Create**
Dimension:	**0D**
Type:	**Mass**

Property Set Name	**np_mass.39000**
Option(s):	**Lumped**
Input Properties...	
Mass:	**40.0**
OK	

Again, let's define the Application Region. We want to assign these properties to the Point elements. First, click in the Select Members databox, the Select Menu will then be displayed. Select the Point Element icon from the Select Menu.



Select Menu:
Point element

Finally, select the All point elements in the Viewport and create the property.

Select Members	Select all Posted point elements
Add	
Apply	

Repeat this procedure for the remaining concentrated masses listed in the table above. Remember to post the Propulsion Block and Science Platforms groups before assigning the properties. Also, make sure to select all 12 Science Platforms point elements.

Mass Property Calculations

11. Now that we have completed the last of the property assignments, we will now compute the mass for the entire model and compare it back to hand calculations.

First start by posting the All Fem group.

Group/Post...

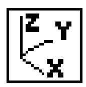
Select Groups to Post

- **Adapter**
- **All Fem** (selected)
- **All Geometry**
- **Central Cylinder**
- **Lower Platform**
- **Navigational Platform**
- **Propulsion Block**
- **Science Platforms**
- **Shear Panels**
- **Upper Platform**
- **default_group**

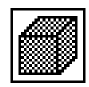
Apply

Cancel

Next, reset the model orientation and render style using the following Toolbar icons.



Isometric View 3



Shaded Smooth

Now, lets compute the mass of the vehicle. Since we defined all the densities and masses in units of weight, the resulting quantities will be in units of lbs.

Tools/Mass Properties...

Action:	**Show**
Dimension:	**3D**
Define Region...	
Region:	**All**
Include:	**FEM**
OK	
Relative to Coordinate Frame	**Coord 0**
Density/Concentrated Mass	**Use Element Properties**
Thicknesses/Areas/NSM	**Use Element Properties**

■ **Plot Principal Axes at CG**

■ **Create Princ. Coord Frame**

□ Write to Report

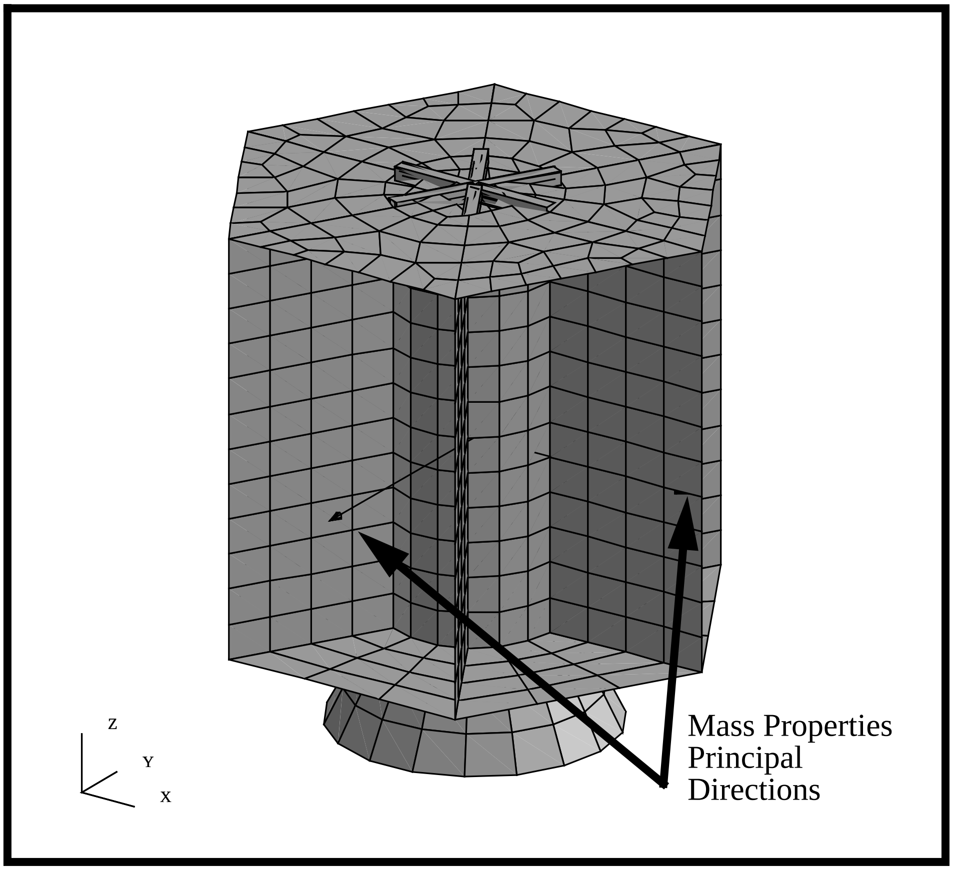
Apply

Upon Apply, a spreadsheet will appear which will include the mass property of the vehicle of 741.1 lbs. You can also look at a variety of other mass property calculations, such as inertia tensors.

Mass Property Display Window **Cancel**

Your model should be similar to the display below

Figure 5.5. Principal Mass Properties Coordinate System for Satellite



As requested, an additional coordinate system will be automatically generated at the model Center of Gravity and Displayed.

The equivalent hand calculation is given in the table below.

Component	Qnty	Calculation	Item Mass
Central Cylinder	1	0.250 in * 6 * 754 in^2 * 0.101 lb/in^3	114.23
Adapter	1	0.075 in * 6 * 284.4 in^2 * 0.16 lb/in^3	20.48
Upper Platform	1	0.100 in * 3367.1 in^2 * 0.101 lb/in^3	34.01
Navigational Platform Support Bars	6	0.76 in^2 12 in * 0.101 lb^in3	5.53
Lower Platform	1	0.350 in * 3367.1 in^2 * 0.101 lb/in3	117.85
Shear Panels	6	0.125 in * 1440 in^2 * 0.101 lb/in^3	109.8
Science Platforms	12	20 lbs	240 lbs
Navigational Platform	1	40 lbs	40 lbs
Propulsion Block	1	60 lbs	60 lbs
TOTAL			**741.90 lbs**

This hand calculation based on geometric areas is in very good agreement with the MSC/PATRAN Mass Property calculation of the Satellite FEM model of **741.10 lbs**.

Close the Database and Quit Patran

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

File/Quit

This will exit MSC/PATRAN and close your file. Do not delete the database from your directory since you will use it for future exercises.