LESSON 3

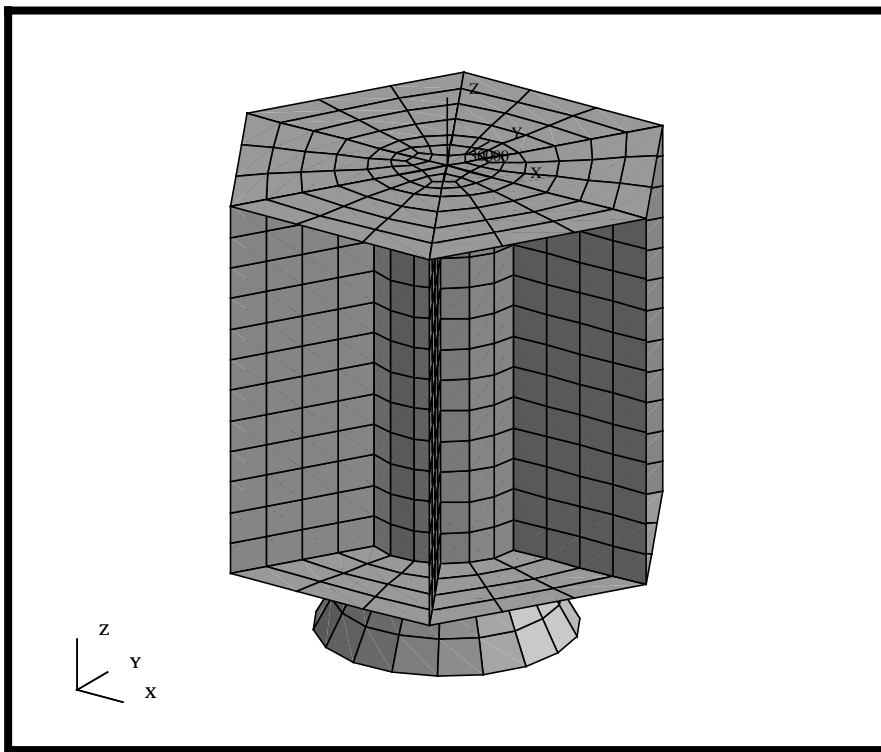# *Building the Finite Element Model of a Space Satellite*

Objectives:

- Apply mesh & MPC's on a Space Satellite

- Perform Model and Element Verification.

- Learn how to create 0-D, 1-D and 2-D elements

*Finite Element Model of a Space Satellite*

Model Description:

In this exercise, you will define a finite element mesh for the Space Satellite model that you developed earlier. You will use mesh seeding to create a controlled mesh. Several components of the satellite model will be idealized as concentrated masses; namely the Propulsion Module, Science Platforms and Navigational Platform. The concentrated masses will be modeled with Point Elements. You will also apply rigid elements to the model to connect concentrated masses to the model.

Additionally, there will be two coordinate systems used to define the On-Orbit and Launch conditions. The specific information regarding to these is given in the following table.

	Coord. Frame ID	Type	Origin Location
Launch	20,000	Rect.	Model Centerline, Base of Adapter
On-Orbit	30,000	Rect.	Center of Upper Platform

(table cont)	**Y-Axis**	**Z- Axis**
Launch	Global X-Axis, Shear Panel #1's Radial Orientation	Global Z-Axis, Along model Longitudinal Axis
On-Orbit	Global X-Axis, Shear Panel #1's Radial Orientation	Global Z-Axis, Along model Longitudinal Axis

Figure 3.1. Locations of the concentrated masses.

MASS LOCATIONS FOR SPACE SATELITE
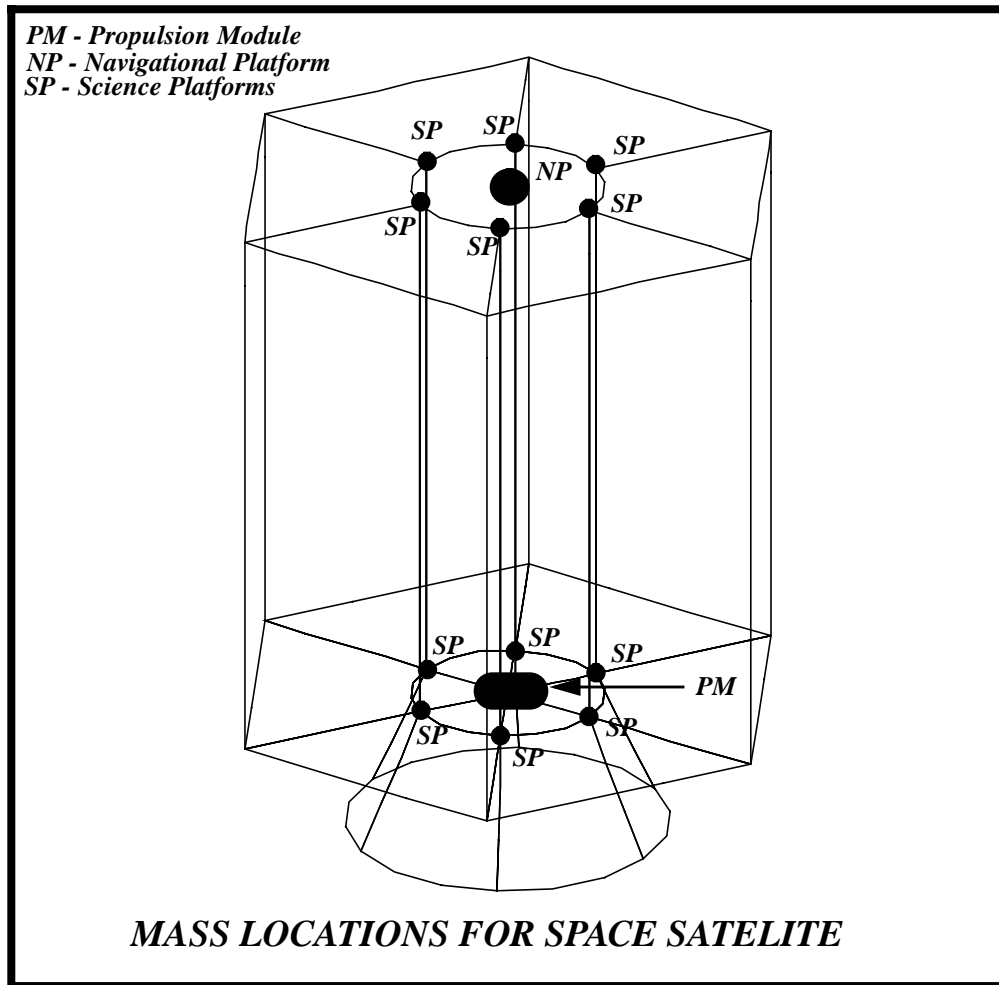
Suggested Exercise Steps:

■ Start MSC⁄PATRAN and open the **satellite.db** file.

■ Post the Upper Platform group and create a point that is located 5 units above its center.

■ Create a coordinate system, called 30000, located at the center of the Upper Platform.

■ Create a coordinate system, called 20000, located at the base of the Adapter Cone by transforming the Upper Platform coordinate system 75 units in -Z direction.

■ Post All Geometry group and update it to contain the newly

created Geometry.

■ Post and mesh the Central Cylinder, Adapter, and Shear Panel groups using Quad4 and IsoMesh with the global edge length of 5 units.

■ Post and mesh the Lower and Upper Platform groups using Quad4 and IsoMesh mesher with the global edge length of 6 units.

■ Create a group for all finite elements in the model.

■ Verify and Equivalencing the mesh.

■ Create a Group that will contain the elements representing the Science Platforms.

■ Post the Upper Platform with Science Platform group (current group). Then create 6 point elements on each Platform as indicated in Figure 3.1.

■ Repeat the previous step for the Lower Platform and Science Group (current group).

■ Use the Group menu to create a group that will contain the elements representing the Propulsion Block.

■ Post the Lower Platform group along with Propulsion Block group (current group).

■ Create a Point Element at the center of the Lower Platform.

■ Create a Rigid (Fixed) MPC by defining the Independent node at the center and the Dependent Nodes as shown in Figure 3.8.

■ Use the Group menu to create a group that will contain all the elements that will be used to model the Navigational Platform.

■ Post the Upper Platform group along with the Navigational Platform group.

■ Create a point element at the center for the Upper Platform. Also, create a bar elements connecting this mass to the upper platform as shown in figure 3.10.

■ Post the All Finite Element group and update it to contain all the newly created bar and mass elements as well as the

Propulsion Block rigid element.

■ Perform a duplicate element verification for the model to insure you have not created duplicate mass or bar elements.

■ Close and Exit the 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.***

Open the Satellite Database

1. Start MSC/PATRAN and open the **satellite.db** file.

File/Open...

Existing Database Name | **satellite.db** |

| **OK** |

2. Before we get started, change the view and render style of the model using the Toolbar icons shown below.

Isometric View 3   **Wireframe**

Create a Point above the Upper Platform

3. In this step, we will define a point just above the Upper Platform, which will use in a later part of this exercise.

To reduce the amount of clutter on the screen, post the only Upper Platform.

Group/Post...

Select Groups to Post

| **Adapter** |
| **All Geometry** |
| **Central Cylinder** |
| **Lower Platform** |
| **Shear Panels** |
| **Upper Platform** |
| **default_group** |

| **Apply** |
| **Cancel** |

Prior to apply, only the Upper Platform should be highlighted. Once applied, the Upper Platform will be the only entity displayed in your graphics window.

To better visualize the points, increase the point size using the toolbar icon to increase point size

Point Size

Create a point located 5 units (Z=65) above the center of the Upper Platform. We will use the Point-Transform method to create this point.

◆ **Geometry**

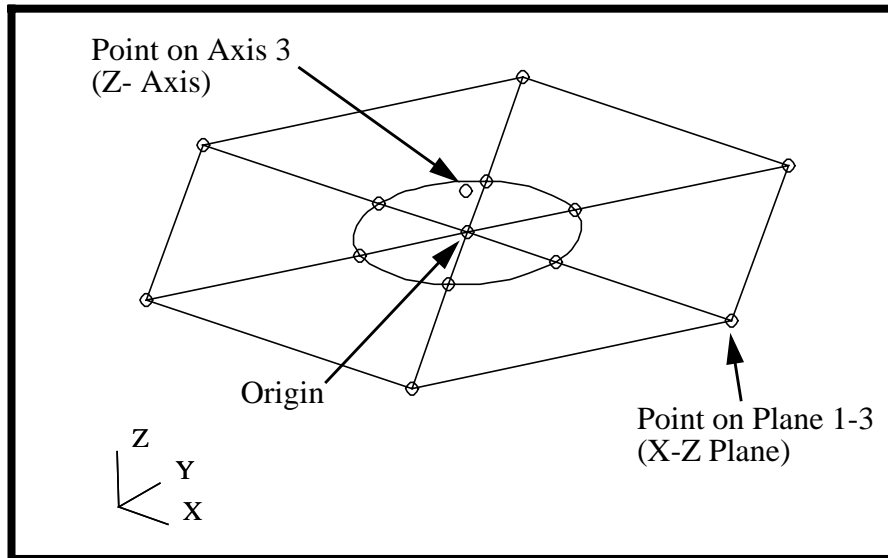|	
Action:	**Transform**
Object:	**Point**
Method:	**Translate**
Translation Vector	**< 0, 0, 5>**
Repeat Count	**1**
Point List	Pick the point located at the center of the Upper Platform

4.   In this step, we will create the On-Orbit (CID = 30,000) and Launch (CID = 20,000) coordinate systems.

Create the On-Orbit Coordinate Systems

Figure 3.2. Points used to create the On-Orbit Coordinate system

All coordinate systems in MSC/PATRAN are defined under the Geometry menu. We will start by defining the On-Orbit coordinate system (CID = 30,000).

Action:	**Create**
Object:	**Coord**
Method:	**3 Point**
Coord ID List	**30000**
Type:	**Rectangular**
Origin	See Figure 3.2 ➤
Point on Axis 3	See Figure 3.2 ➤
Point on Plane 1-3	See Figure 3.2 ➤

Create the Launch Coordinate Systems

5.   Create the Launch Coordinate system, (CID = 20,000).

Next, we will create the Launch coordinate system, CID = 20,000). The new coordinate system, the Launch coordinate system, will be located along the model centerline at the base of the Adapter Cone (Z=-15). We will create this coordinate system by transforming the On-Orbit coordinate system a distance of 75 units in the -Z direction.

Action:	**Transform**
Object:	**Coord**

Method:	**Translate**
Coord ID List	**20000**
Translation Vector	**< 0, 0, -75 >**
Repeat Count	**1**
Coordinate Frame List	**Coord 30000**

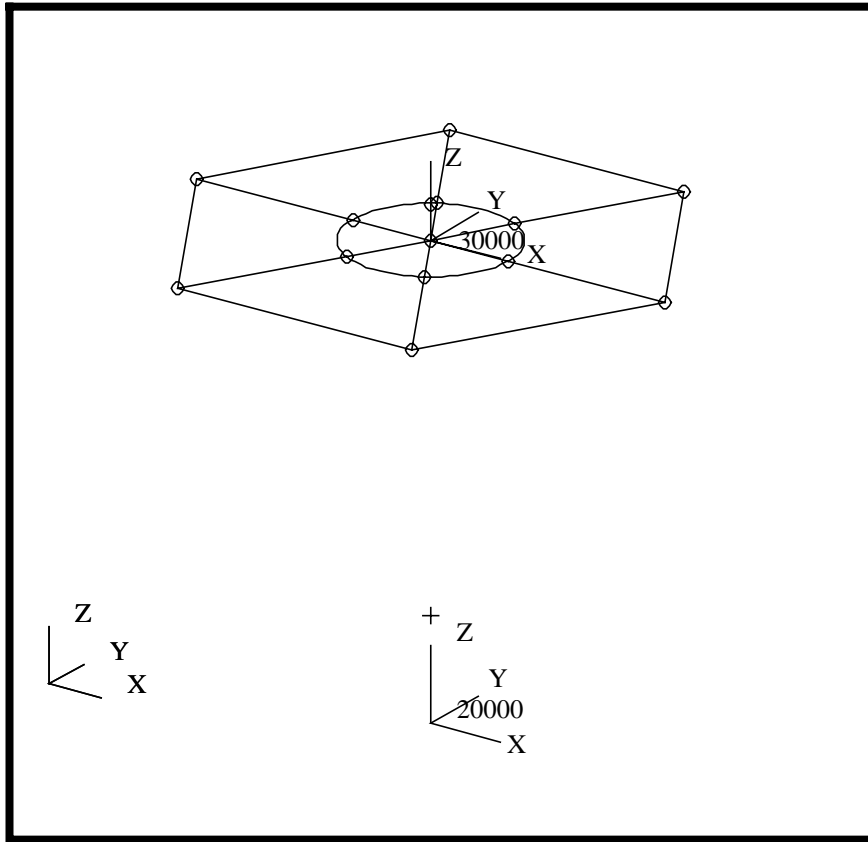In order to see the New Coordinate System, you may need to do a Fit View using the Toolbar icon

Fit View

Toggle off the point size by selecting the following Icon from the Toolbar

Point Size

Your model should look like the figure below

Figure 3.3. The Launch and On-Orbit coordinate systems.

6. Update the All Geometry group to add the recently created points.

Group/Post...

Select Groups to Post

Adapter
All Geometry
Central Cylinder
Lower Platform
Shear Panels
Upper Platform
default_group

Apply

Action:

All Geometry

OK

Modify

Add

7.   We will now turn our attention to creating the elements for the Finite Element model. We will define elements for each of the concentrated masses, bar elements, and rigid elements. In addition, we will mesh all the surfaces.

Create the Shell FEM Mesh

We will post each group and mesh the surfaces, paying careful attention to the mesh size. By using a group by group approach, we can assure that each group contains the geometry and finite elements for each of the associated parts. Let's start by posting the Central Cylinder group.

Group/Post...

Select Groups to Post

Adapter
All Geometry
Central Cylinder
Lower Platform
Shear Panels
Upper Platform
default_group

Apply
Cancel

Under the Finite Elements Menu, we will use the default element **Topology (Quad4)** and **Mesher (Isomesh)** to mesh all the Central Cylinder surfaces. We will set the Global Edge Length to a 5 units.

◆   **Finite Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Surface**
Global Edge Length	5
Element Topology	Quad4
Mesher	◆ **IsoMesh**
Surface List	Select all posted surfaces

Apply

Repeat this procedure to mesh the following components with the meshing parameters provided in the table below. For all the groups listed below, we will use the default ISOMESHER and QUAD4 element topology.

Component	Element Topology	Mesh Size
Adapter	Quad4	5
Shear Panels	Quad4	5
Lower Platform	Quad4	6
Upper Platform	Quad4	6

Create a Group for the All FEM

8. In this step, we will create a group for all finite elements.

Group/Create...

New Group Name   **All Fem**

☐ **Make Current**
☐ **Unpost All Other Groups**

Group Contents: **Add All FEM**

Apply

Cancel

Again, to better visualize the finite elements, turn on the Smooth Shaded render style and change to Isometric view using the Toolbar icons.
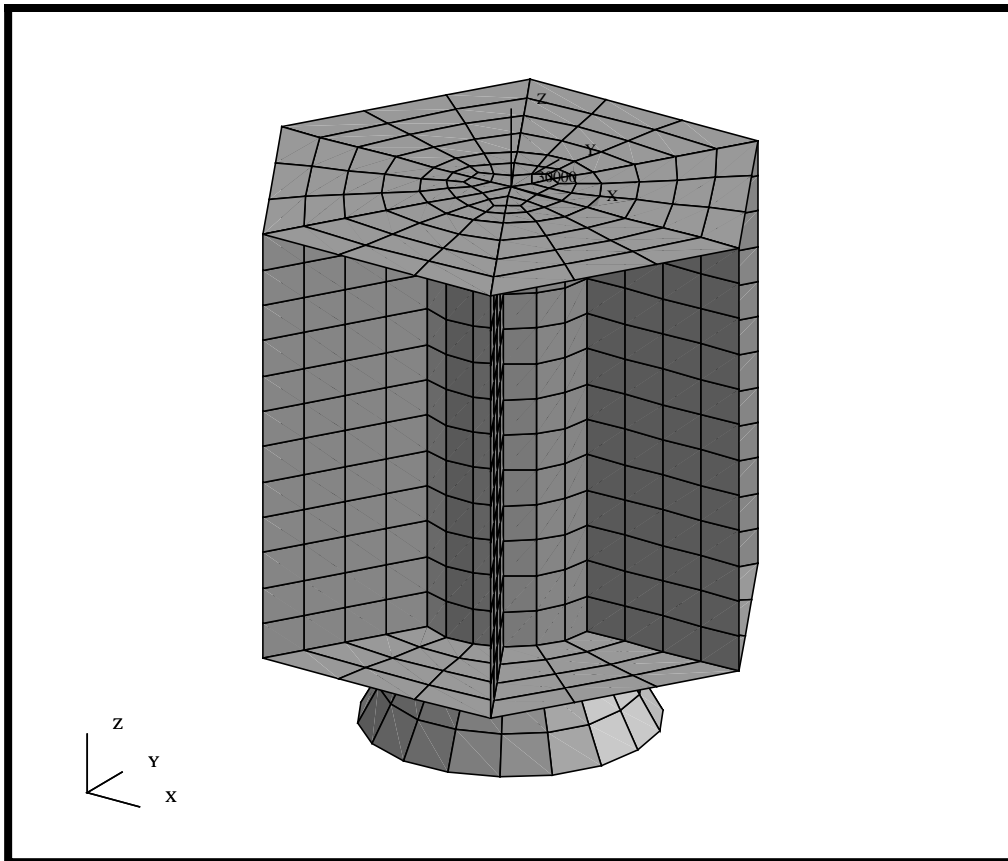
Isometric View 3   **Smooth Shaded**

Your model should look like the figure below.

Figure 3.4. Shell Mesh of the Satellite model.

Let's change the render style back to wireframe using the Toolbar icon shown below.

Wireframe

9. We will perform a verification on the model and check for internal free edges, which indicate where cracks exist in your model.
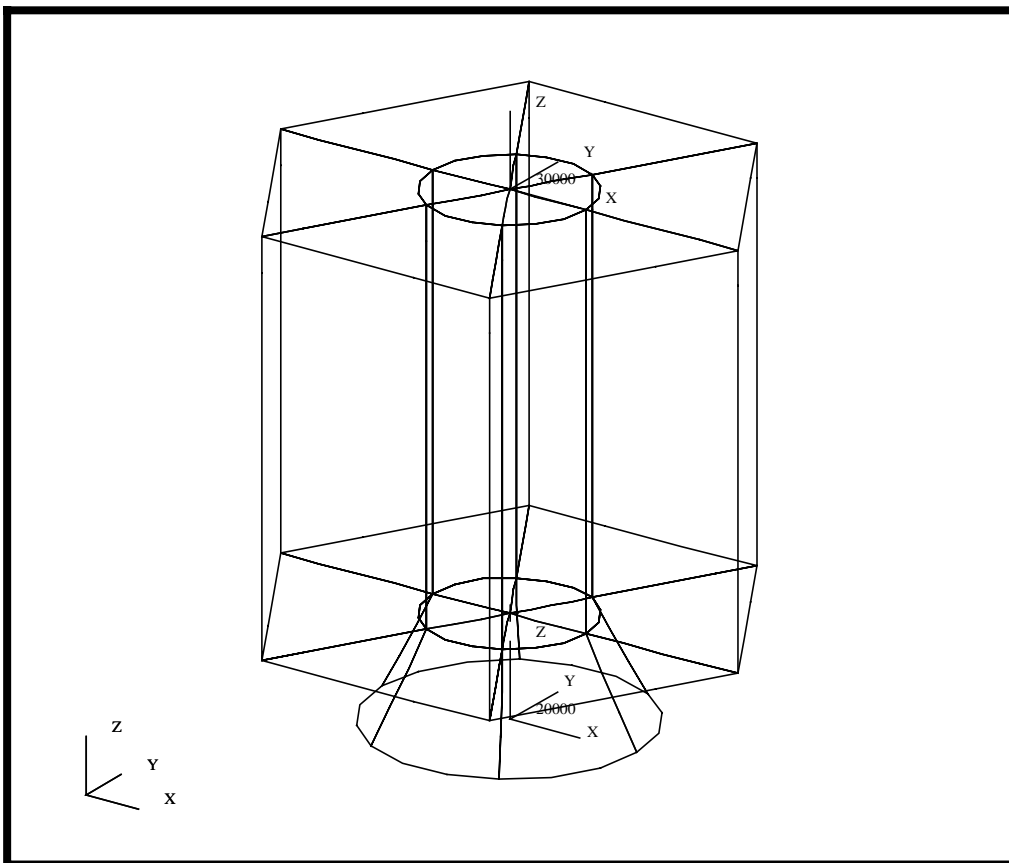
Action: | **Verify**

Object: | **Element**

Test: | **Boundaries**

Apply

MSC/PATRAN will render your model's free edges as yellow lines. Your model should look like the figure below.

Figure 3.5. Verify element boundaries of the shell mesh before equivalencing

Notice that prior to equivalencing, all geometric boundaries appear as free edges (cracks) in your model. **MSC/PATRAN** defines free edges as an edge that is shared by only one element. Click on the **Reset Graphics** button to re-render your model in its original render style.

Reset Graphics

10. Equivalence the Satellite model and then verify the element boundaries.

By equivalencing the model, all duplicate nodes will be removed. Hence, the finite element model represents the object as a single congruent mesh.

Equivalence and Verify Element Boundaries

Action: **Equivalence**

Object: **All**

Method: **Tolerance Cube**

Apply

As the equivalencing process proceeds, the locations which have been modified will be identified by circles. Verify the Element Boundaries again.
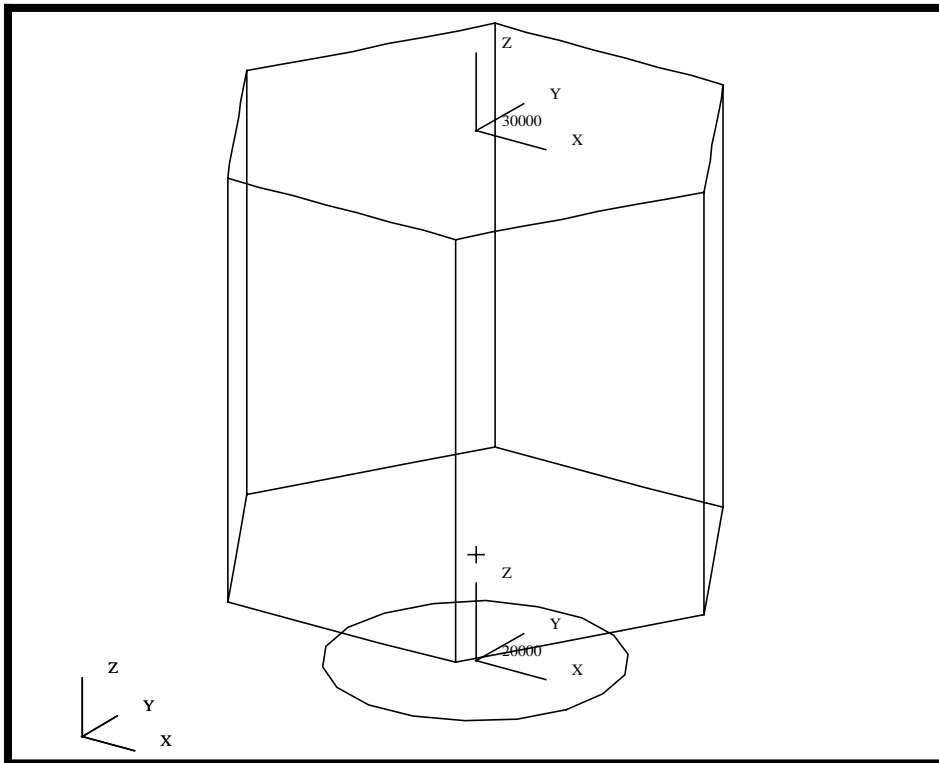
Action: **Verify**

Object: **Element**

Test: **Boundaries**

Apply

Now you should only see the exterior edges of the satellite, as shown in the figure below

Figure 3.6. Verify element boundaries of the satellite after equivalencing.

Make sure to reset the graphics to re-render the model before you proceed.

Reset Graphics

11. We will now model the concentrated masses for the Science Platforms. The Science Platforms, Propulsion Module and Navigational Platform will be idealized as concentrated masses. Each of the concentrated masses sets will be put into a group, which will aids us when we do the Property assignment.

The Science Platform will be represented by12 concentrated masses located at the vertices where the Shear Panels, Upper/Lower Platforms and Central Cylinders intersection as shown in Figure 3.7.

Figure 3.7. Point Mass locations for Science Platforms.
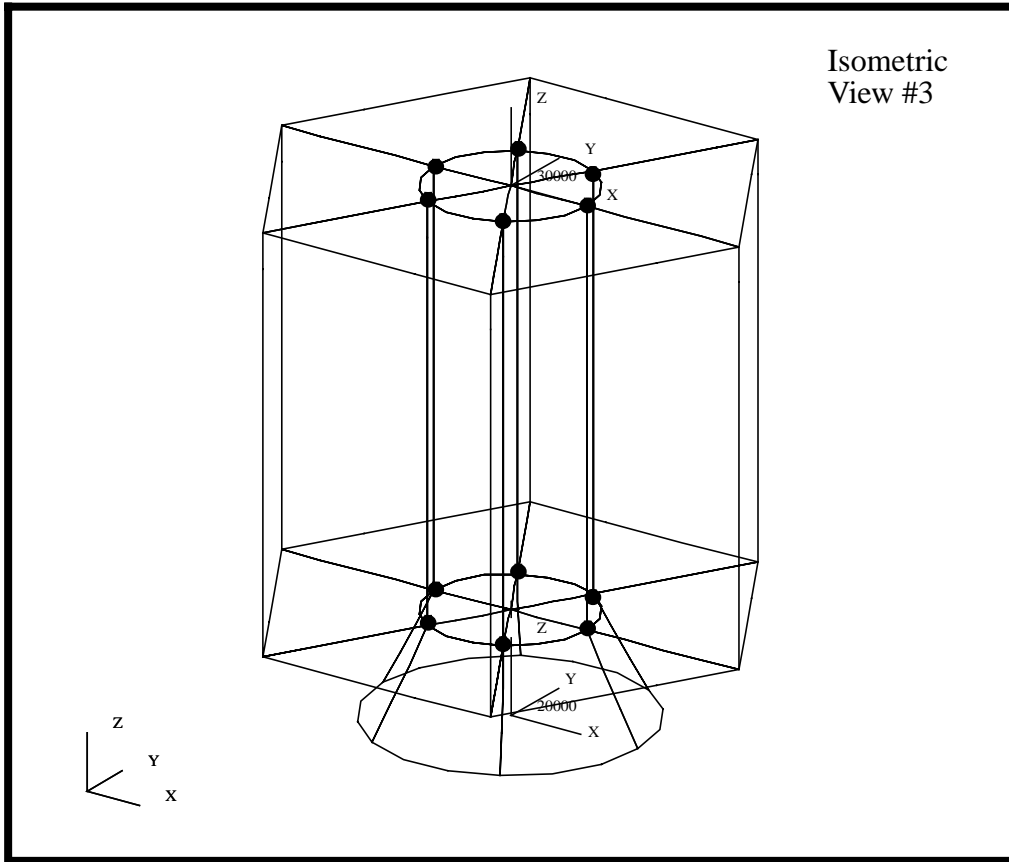
We will start by creating a Science Platform group that contains the mass for all 12 Science Platforms and then posting the Upper Platform.

Group/Create...

Action:	**Create**
New Group Name	**Science Platforms**
	☑ **Make Current**
	☑ **Unpost All Other Groups**
Group Contents:	**Add Entity Selection**

Apply

Action:	**Post**

Select Groups to Post

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

<u>Note</u>: You need to hold the
Control key down to select two
groups at the same time.

Apply
Cancel

Change the view of the model to the Front view using the Toolbar
icons.

Front view

We will first create the 6 Point Elements located on the Upper
Platform.

Action:	**Create**
Object:	**Element**
Method:	**Edit**
Shape	**Point**
Topology	**Point**
Node 1 =	See Figure 3.7

Again, there is no need to hit Apply as long as the Auto Execute is on.
You will see a orange triangle appear on the screen confirming the
creation of the point element.

*Repeat this step for the other 5 elements located on the Upper
Platform.*

Unpost the Upper Platform Group and Post the Lower Platform group as follows making sure the Science Platforms group remains current.

Group/Post...

Action:

Post

Select Groups to Post

Adapter **All Fem** **All Geometry** **Central Cylinder**
Lower Platform **Science Platforms**
Shear Panels **Upper Platform** **default_group**

Apply

Cancel

Now repeat the step above to create the 6 Point Elements associated to the Lower Platform.

12. We will now model the concentrated mass and rigid element for the Propulsion Block. The Propulsion Block will be put into a group, which aids us when doing the Property assignment.

Create the FEM for the Propulsion Platform

The Propulsion Block (PB) is located at the center of the Lower Platform. Since the PB is much stiffer than the structure around it, we will assume the PB is rigid. We will create a rigid element (RBE2) to 'wagon wheel' tie the lumped mass to the inner edge of the Lower Platform.

We will create a Propulsion Block group that contains the concentrated mass and Rigid Element.

Group/Create...

Action:

Create

New Group Name

Propulsion Block

■ **Make Current**
■ **Unpost All Other Groups**

Group Contents

Add Entity Selection

Apply

Action:	Post

Select Groups to Post

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

Apply
Cancel

Now create the Point Element located at the center of the Lower Platform (reference figure 3.8).

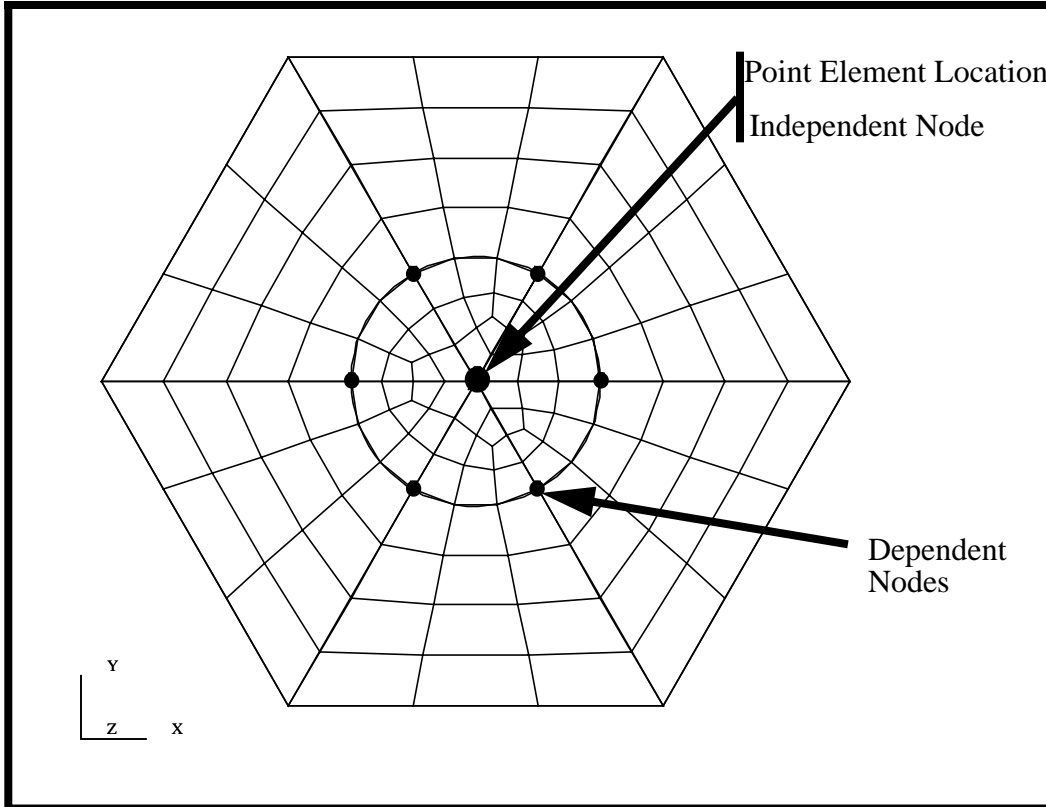| *Action:*	Create
Object:	Element
Method:	Edit
Shape:	Point
Topology:	Point
Node 1 =	Node located at the center of the Lower Platform

Next, we will create the a rigid element to tie the center point of the Propulsion Block to the inner edge of the Lower Platform/Shear Panel/ Central Cylinder intersection.

Figure 3.8. The Dependent and Independent nodes.

Make sure to turn off the Auto Execute button prior to picking any Nodes.

Action:	**Create**
Object:	**MPC**
Type:	**Rigid (Fixed)**

Define Terms...

☐ **Auto Execute**

◆ **Create Dependent**

Node List ☐ See Figure 3.8 ↖

Apply

◆ **Create Independent**
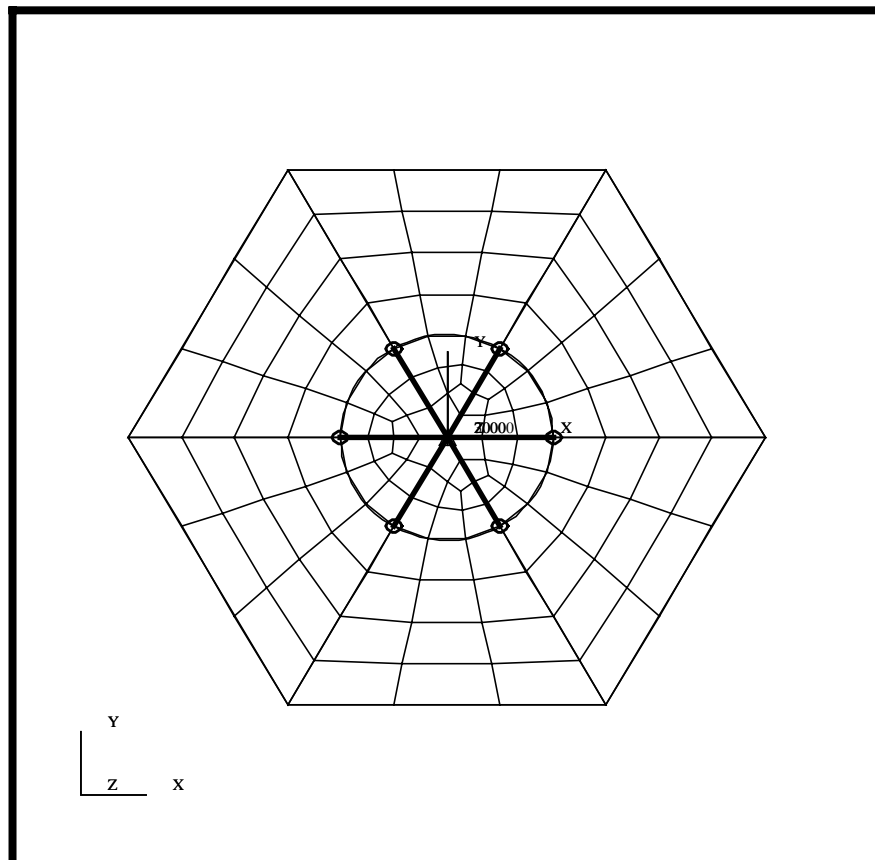
Node List ☐ See Figure 3.8 ↖

Apply

Cancel

Apply

Your model should now look like the figure below

Figure 3.9. Lower Platform with Propulsion Block modeled.

Finite Element Model of a Space Satellite

13. We will now model the concentrated mass and Bar elements for the Navigation Platform. The Navigation Platform will be put into a group, which aids us when doing the Property assignment.

The Navigation Platform (NP) is located at the center of the Upper Platform. Since the NP attach structure will be modeled with beam elements, we will manually create 6 *Bar2* elements. These elements will attach the center node/concentrated mass to the six Shear Panel/ Central Cylinder/Upper Platform intersection points.

We will create a Navigational Platform group that will contains the Navigation Platform mass and *Bar2* elements.

Group/Create...

Action:	**Create**
New Group Name	**Navigational Platform**
	■ **Make Current**
	■ **Unpost All Other Groups**
Group Contents	**Add Entity Selection**

Apply

Action:	**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

Create the Point Element located at the center of the Upper Platform.

Action:	**Create**

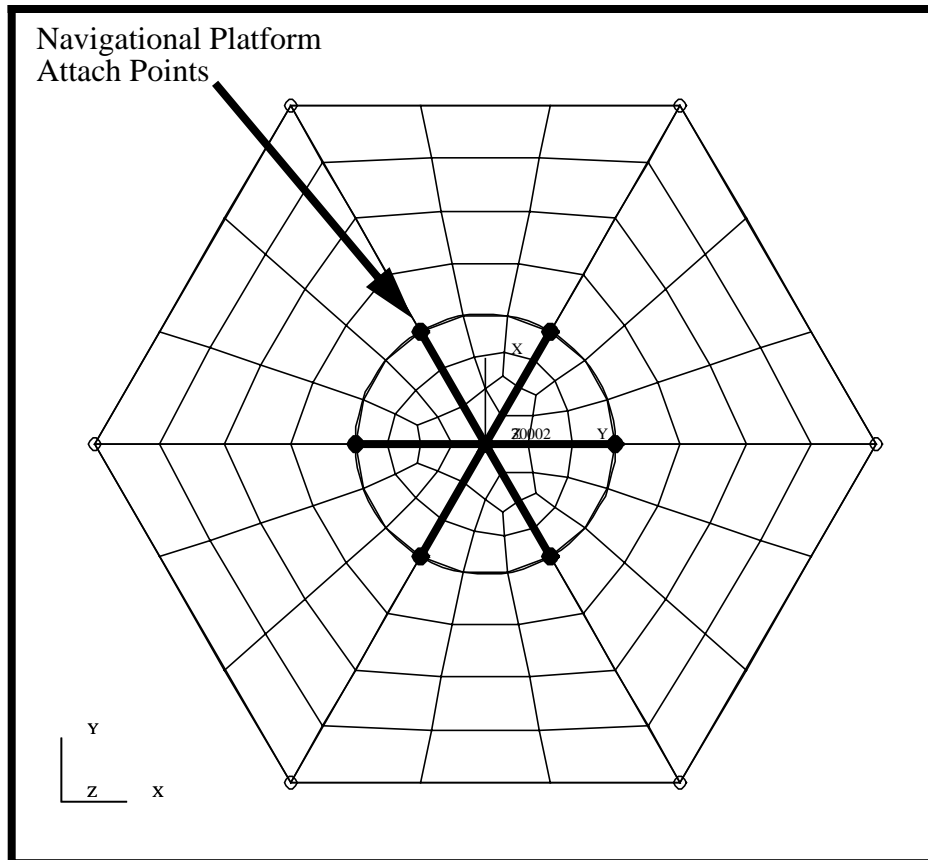| _Object:_ | **Element** |
Method:	**Edit**
Shape:	**Point**
Topology:	**Point**
Node 1 =	Node located at the center of the Upper Platform

Now create the attach structure going from the concentrated mass to each of the six Shear Panel/Central Cylinder/Upper Platform intersection points. We will manually create single elements as shown in the figure below

Figure 3.10. The Navigational Platform attachment structure

Action:	**Create**
Object:	**Element**
Method:	**Edit**

Shape:	**Bar**
Topology:	**Bar2**
Pattern:	**Standard**
Node 1 =	Node located at the center of the Upper Platform ➤
Node 2 =	See Figure 3.10 ➤

Again, you will not need to hit Apply as long as the Auto Execute is on. Once the bar is created, you will get a yellow line showing the connectivity. *Repeat this step for the remaining 5 bars*.

14. Before we finish, let's modify the All Fem group and add the newly created elements.

Update the All Fem 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

Action:	**Modify**

Change Target Group...

Existing Groups

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

Cancel

All FEM

Add

OK

Change the view to Isometric view and the render style to Smooth shaded using the Toolbar icons.
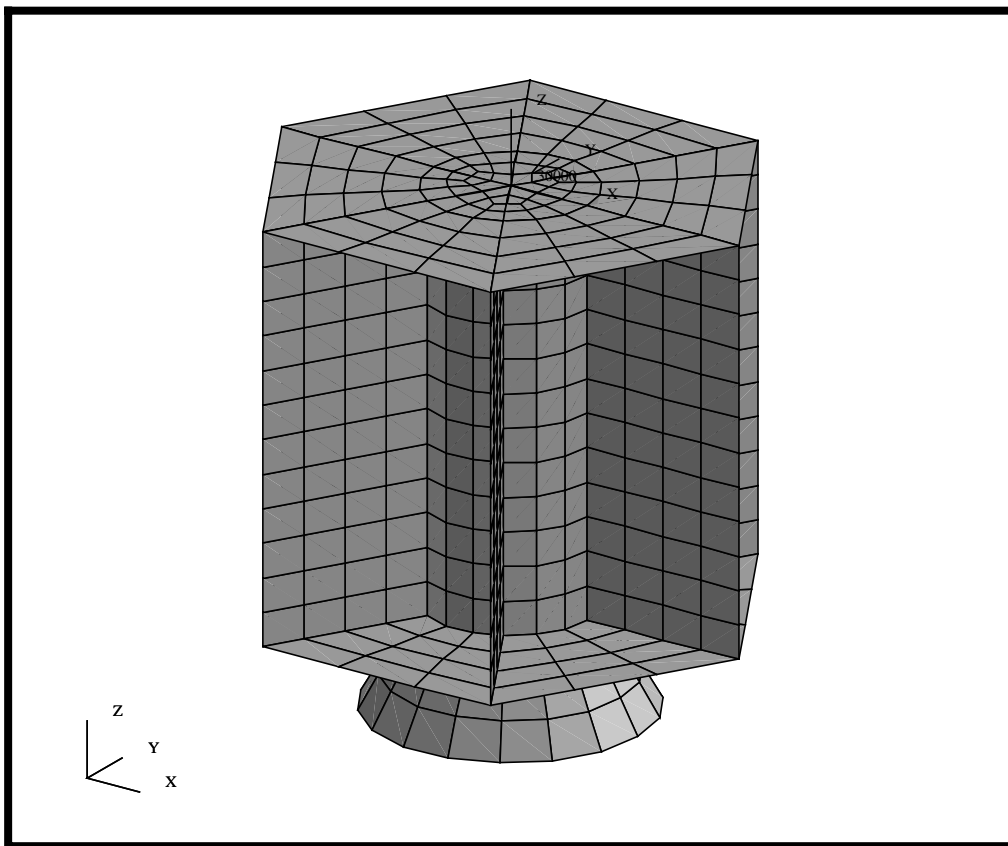
Isometric View 3

Smooth shaded

Finally, your Model should now look like the figure below

Figure 3.11. The Satellite FEM model

Perform a Duplicate Element Check

15. We will perform a final verification on the model and check for duplicate elements which may have inadvertently been created.

Action: | **Verify** |

Object: | **Element** |

Test: | **Duplicates** |

Test Control

Display Only

| **Apply** |

Upon completion, you should see the following message in the history window of the *MAIN* form.

$# There are NO duplicate elements in the model.

If you had duplicate elements, you would perform the following step.

Test Control

Delete Duplicates

▉ **Delete Higher ID**

| **Apply** |

Close the Database and Quit Patran

16. 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.

It should be noted that all changes made to the database are automatically saved and there is no need to perform a save operation.