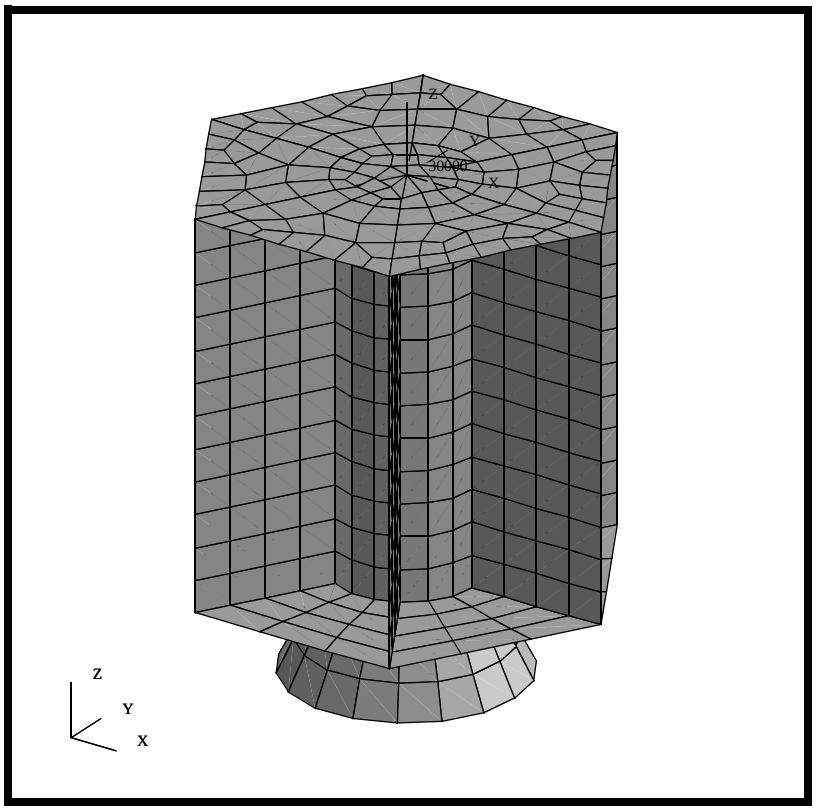
Exercise 4

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 control mesh parameters and IDs.

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 and geometry associativity to create a controlled/congruent 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. Finally, you will control element and node IDs using a combination of model creation techniques and finite element renumbering options.

Since this model will be coupled with other models, the element and node numbering need to be controlled so that none of the elements/ nodes from associated components overlap. The other components are being developed by other subcontractors and the definition of these IDs is controlled by the agency for which this vehicle is being developed. The following table lists the finite element numbering requirements.

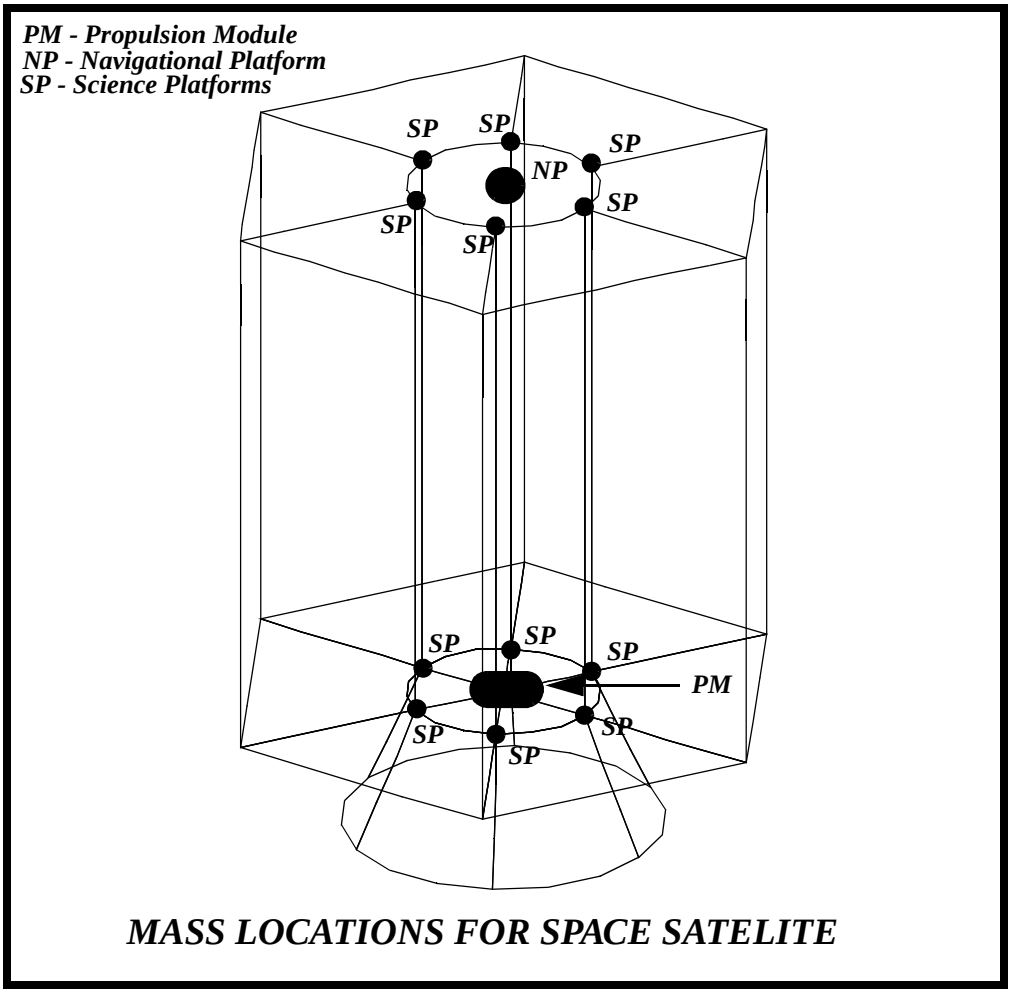
Component	**Grid ID Range**	**Element ID Range**
Central Cylinder	10,000-19,000	10,000-19,000
Adapter	20,000-29,000	20,000-29,000
Upper Platform	30,000-39,000	30,000-39,000
Lower Platform	40,000-49,000	40,000-49,000
Shear Panels	50,000-59,000	50,000-59,000
Navigational Platform	N/A	39001
Navigational Platform Tiedown	N/A	39101-39106
Propulsion Block	N/A	49001
Propulsion Block Attachment	N/A	49002
Science Platforms Located on each Shear Panel	N/A	59101- 59106 59201- 59206

Additionally, there will be two coordinate systems used to define the On-Orbit and Launch conditions. The specific information pertaining 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 4.1. The concentrated mass locations.



Suggested Exercise Steps:

- Start MSC.Patran and open the **`satellite.db`** file.
- Post the Upper Platform and associate a point on its surface.
- Post the Central Cylinder, the Shear Panels, and the Upper Platform. Associate all their curves and surfaces.
- 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 the Central Cylinder and mesh all surfaces using the IsoMesher and Quad4 type elements with a Global Edge Length of 5. The Node and Element IDs will start with an ID of 10001.
- Repeat the above step for the following groups:

Component	**Starting Grid/ Element ID**	**Mesh Size**
Adapter	20,001	5
Shear Panels	50,001	5
Lower Platform	40,001	6

- Post the Upper Platform and mesh its surface using Paver Mesher with the Global Edge Length of 5.
- Renumber the Node and Element IDs, starting with IDs of 30001, for the Upper Platform.
- Create a group that contains all the Finite Elements.
- Equivalence and Verify the Finite Elements model.
- 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 4.1.
- 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 4.11.
- 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 4.13.
- 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 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

Associate a Point to the Upper Platform

3. We will start by associating a point to the Upper Platform. This point will represents a tie in for the Navigational Platform concentrated mass. This association guarantees that the mesher will place a node at this point.

To reduce the amount of clutter on the screen, we will first use Group/Post to display only the 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 entities displayed in your graphics window.

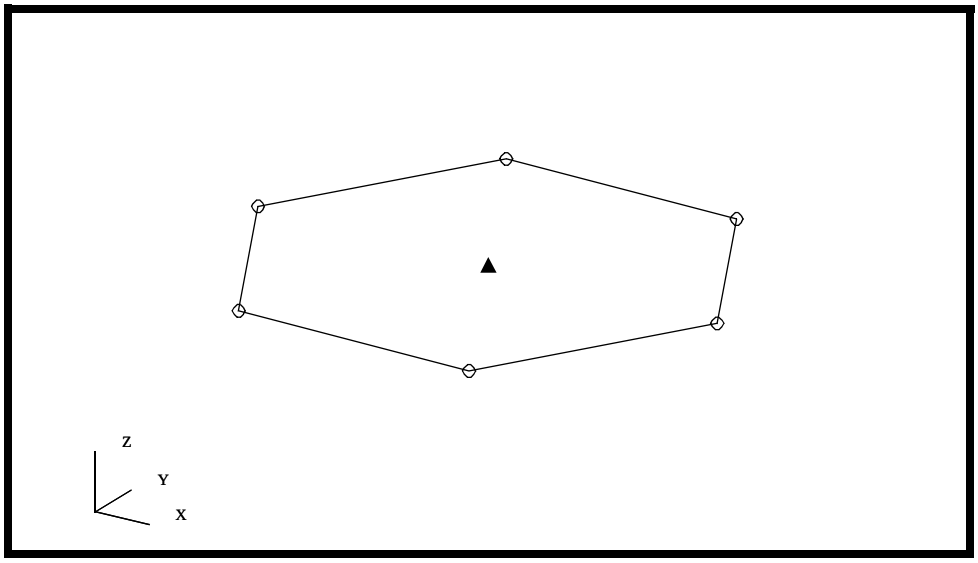
From the Geometry menu, we will associate a point to the surface representing the Upper Platform.

Geometry

Action:	**Associate**
Object:	**Point**
Method:	**Surface**
Point List	Pick the point at the center of the Upper Platform
Surface List	Pick the surface representing the Upper Platform

Your model should now look like the figure below.

Figure 4.1. The Upper Platform with a center point associated.



Toggle off the point size by selecting the following icon from the Toolbar.

Point Size

Associate Surface Edges to the Upper Platform

4. In this step, we will associate the upper edges of the Central Cylinder and Shear Panel to the Upper Platform. By making this association, the mesher will put element edge boundaries along these lines.

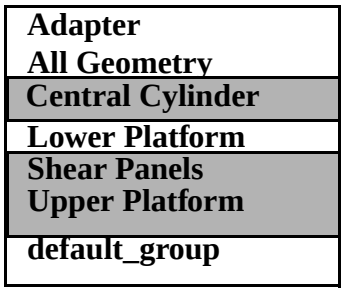
This step was done intentionally to illustrate the associate functionality (Hard curves + points). It should be noted, if all the surfaces representing the Lower Platform were transformed up during the geometry creation step, then this step would not be necessary because the geometry would already be congruent. Topological congruency is requred to insure that the meshes of adjacent surfaces and solids will be created such that However, in more complex projects, geometry comes from many sources and often times a simple geometric association will save time.

We will now post the additional groups that need to associate the curve to the surface. Highlight the groups shown below.

Note: To select non-continuous list of items from a list box (as in this case), use the <Ctrl> Key while picking, this is also called “Block Selecting”.

Group/Post...

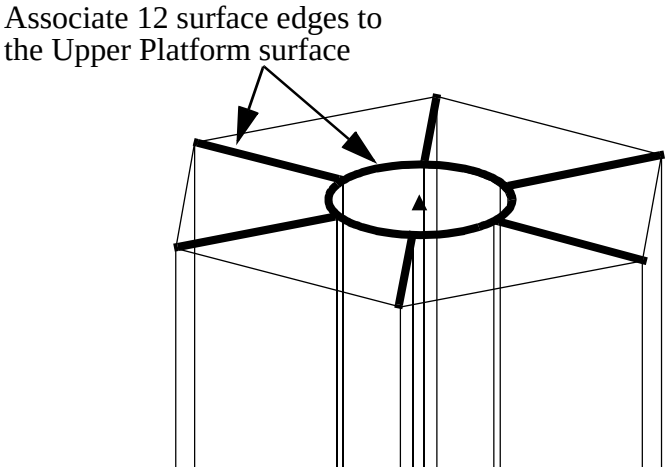
Select Groups to Post



Apply

Cancel

Figure 4.2. Central Cylinder and Shear Panel Edges to associate.



To accomplish this, we will use Associate from the **Geometry** menu.

Action:	**Associate**
Object:	**Curve**
Method:	**Surface**
Curve List	See Figure 3.2
Surface List	Pick the surface representing the Upper Platform

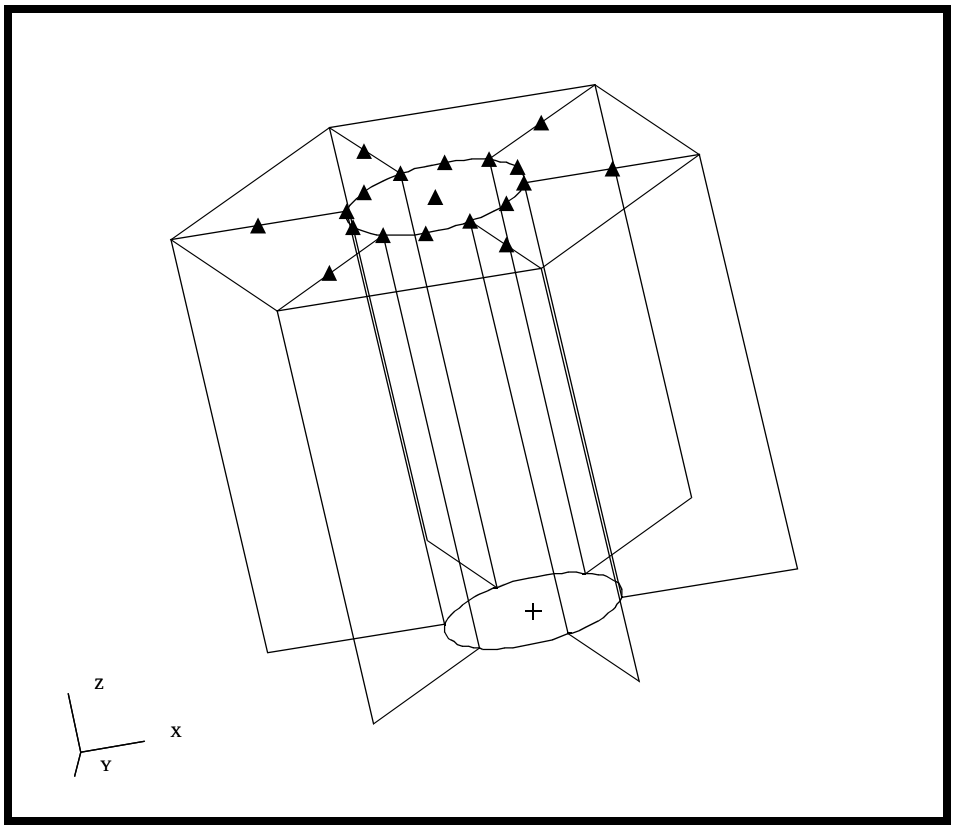
A small triangle on the curve or point will indicate the association has been made. You may need to zoom-in on the area shown in the figure below. We can do this using the View Corners icon from the Toolbar and selecting the area you wish to zoom-in on.

View Corners

Repeat this operation for each edge shown in the Figure 3.2.

When you get done, your model should look like the following.

Figure 4.3. Surface edges and points that have been associated to the Upper Platform indicated by triangles.



Create a Point above the Upper Platform

5. In this step, we will define a point just above the Upper Platform.

To reduce the amount of clutter on the screen, we will first use Group/Post to display only the Upper Platform.

Group/Post...

Select Groups to Post

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

Apply

Cancel

Before we get started, let's change the view and increasing the point size using the Toolbar icons.

Isometric View 3

Point Size

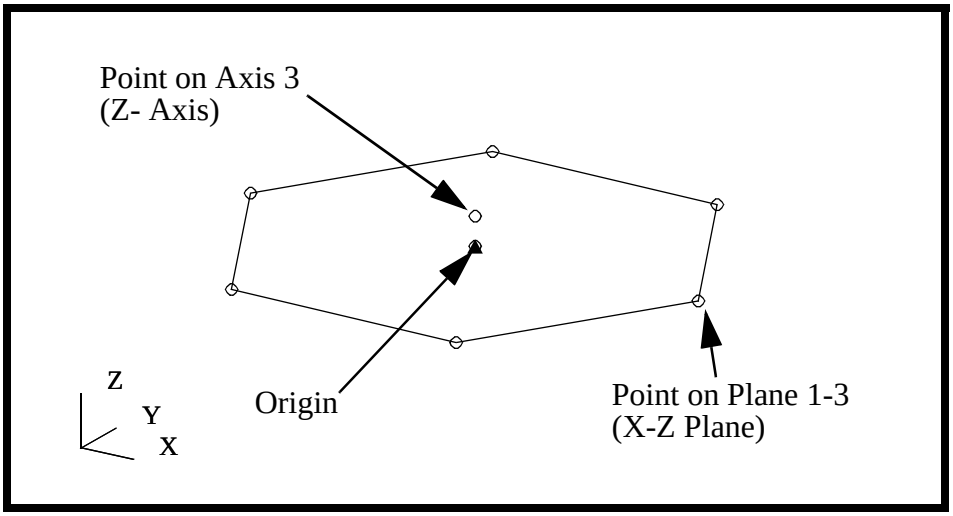
Create a point located 5 units (Z=65) above the center of the Upper Platform. We will use the Point-Transform method to create this point.

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

Create the On-Orbit Coordinate Systems

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

Figure 4.4. 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 3.4
Point on Axis 3	See Figure 3.4
Point on Plane 1-3	See Figure 3.4

Create the Launch Coordinate Systems

7. 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.

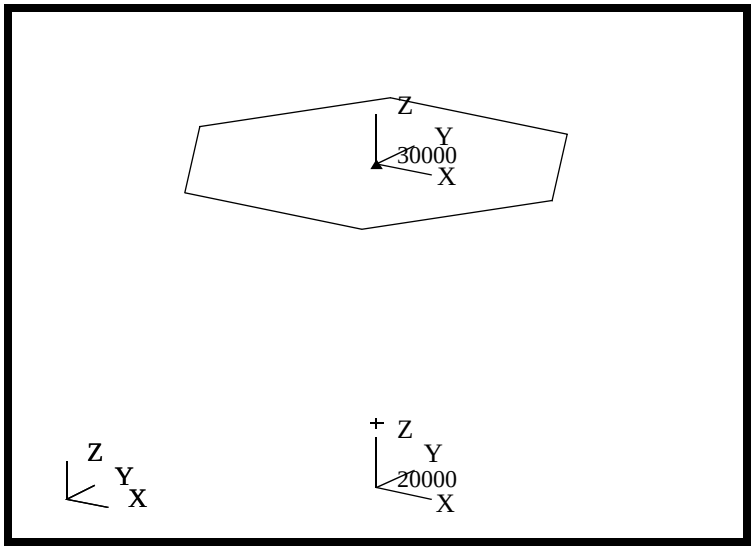
Fit View

Toggle off the point size by selecting the following icon from the Toolbar.

Point Size

Your model should look like the following below.

Figure 4.5. The Launch and On-Orbit coordinate systems.



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

Update the All Geometry Group

Group/Post...

Select Groups to Post

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

Apply

Action: **Modify**

All Geometry **Add**

OK

Create the Shell FEM Mesh

9. 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.

We will post each group and mesh the surfaces, paying careful attention to the mesh size, element and node ID's. 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. If you forget to define the element or node IDs, you can use **Renumber/ Node** and **Renumber/Element** to correct the IDs. Let's start with the Central Cylinder.

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 define a surface mesh for the Central Cylinder with a global edge length of 5 units. The Node and Element Range for this component is between 10,000 to 19,000. We will start with a node and element ID of 10,001 for convience.We will use the default element **Topology (Quad4)** and **Mesher (Isomesh)**.

Finite Elements

Action: **Create**

Object: **Mesh**

Type: **Surface**

Node ID List **10001**

Element ID List **10001**

Global Edge Length **5**

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 ISOMESH and QUAD4 element topology. DO NOT mesh the Upper Platform at this time.

Component	Starting Grid/ Element ID	Mesh Size
Adapter	20,001	5
Lower Platform	40,001	6
Shear Panels	50,001	5

We will now mesh the Upper Platform using the Paver Mesher. The Upper Platform must be meshed with the Paver for two reasons:

1) Because it is a trimmed surface.

2) Because it has Geometric Points and Curves associated with the *interior* (Hard Geometry).

If either one of the above conditions is true, the Paver must be used. It should be noted, the previous Geometric Associations (Hard Curves and Hard Points) will be followed by the PAVER. Do not worry about the Element and Node ID's at this time, we will go back and renumber them once we have completed the meshing.

Let's start by posting the Upper Platform group.

Group/Post...

Select Groups to Post

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

Apply

Cancel

Let's mesh the surface using the Paver Mesher.

Action: **Create**

Object: **Mesh**

Method: **Surface**

Global Edge Length **5**

Mesher **Paver**

Surface List Select the Upper Platform surface

Apply

We will now renumber the nodes and elements of the Upper Platform.

Action: **Renumber**

Object: **Node**

Start ID or List of New ID's **30001**

Node List Select All Upper Platform Nodes

Apply

Action: **Renumber**

Object: **Element**

Start ID or List of New ID's **30001**

Element List Select All Upper Platform elements

Apply

Create a group for the All FEM

10. 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

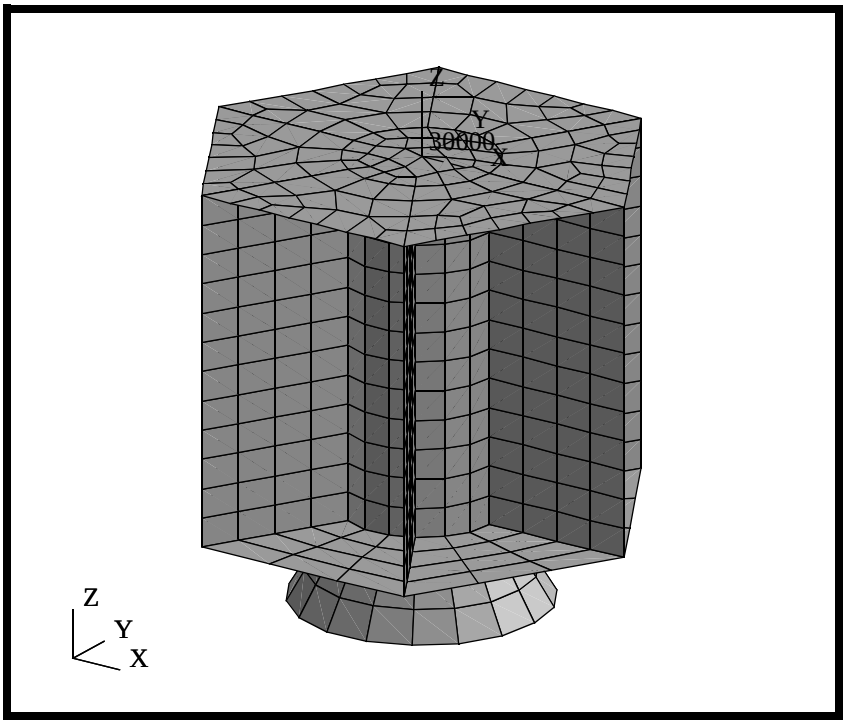
Change the view and the render style back to Isometric view and Smooth shaded, respectively, using the Toolbar icons.

Isometric View 3

Smooth Shaded

Your model should now look like the figure below.

Figure 4.6. Shell Mesh of the Satellite model.



Let’s change the render style back to Wireframe using the Toolbar icons shown below.

Wireframe

Verify Element Boundaries

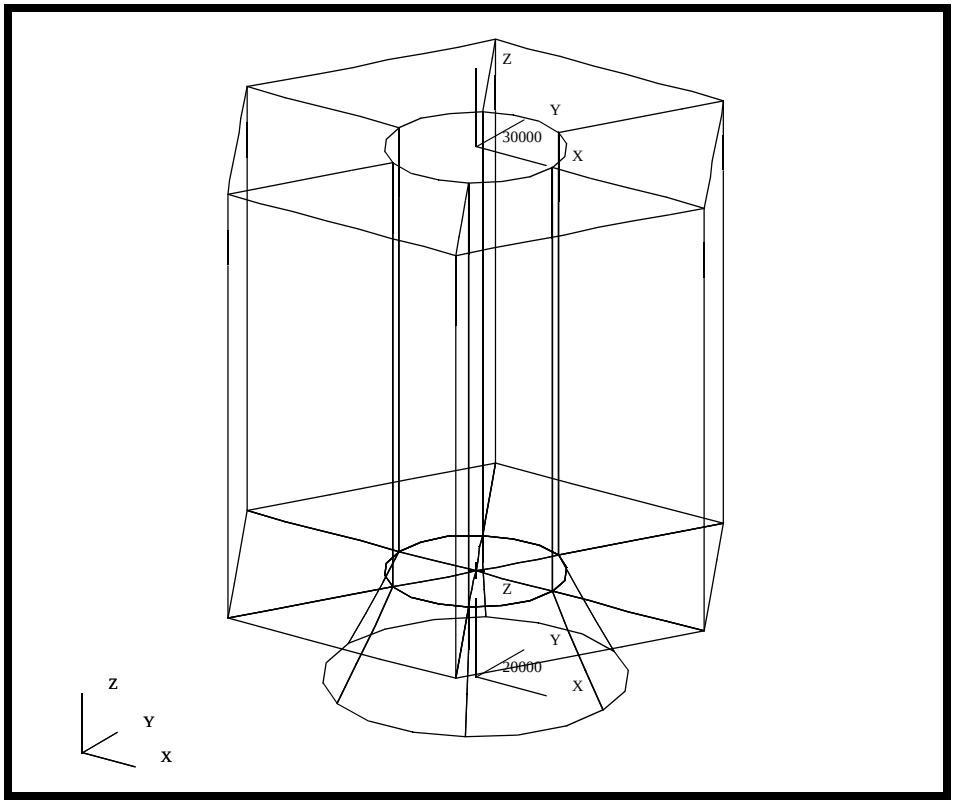
11. We will perform a verification on the model and check for internal free edges, which indicate where cracks exist in your model.

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 4.7. 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

Equivalence and Verify Element Boundaries

12. 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.

Action:	**Equivalence**
Object:	**All**
Method:	**Tolerance Cube**

Apply

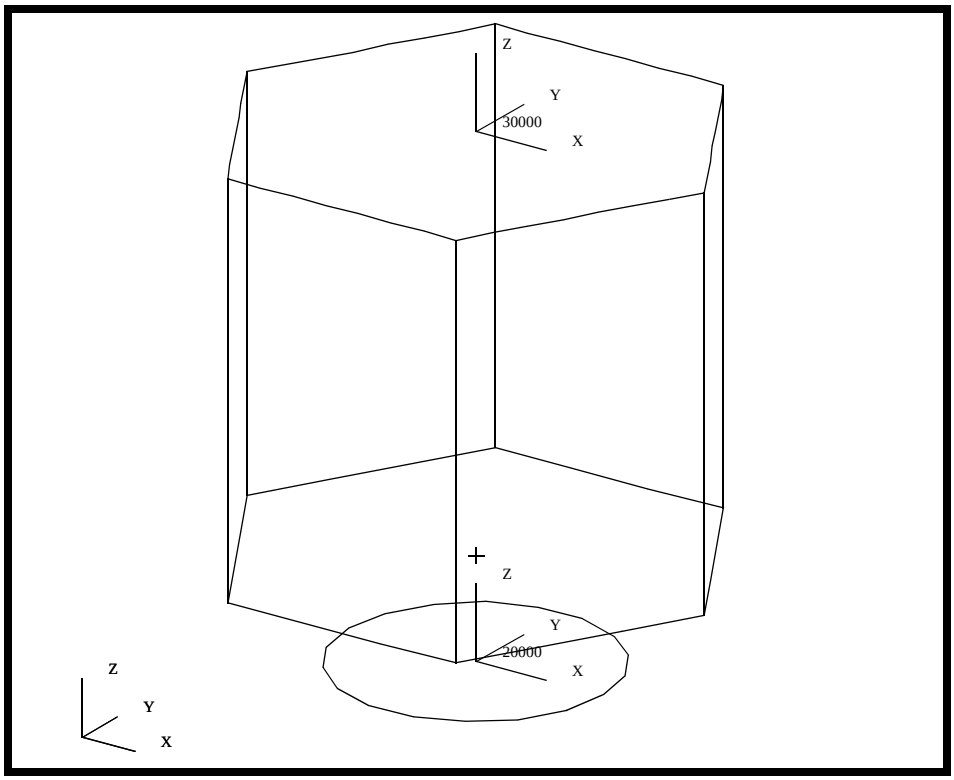
As the equivalencing process proceeds, the locations which have been modified will be identified by circles. Verify the Element Boundaries again.

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 4.8. Verify element boundaries of the satellite after equivalencing.



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

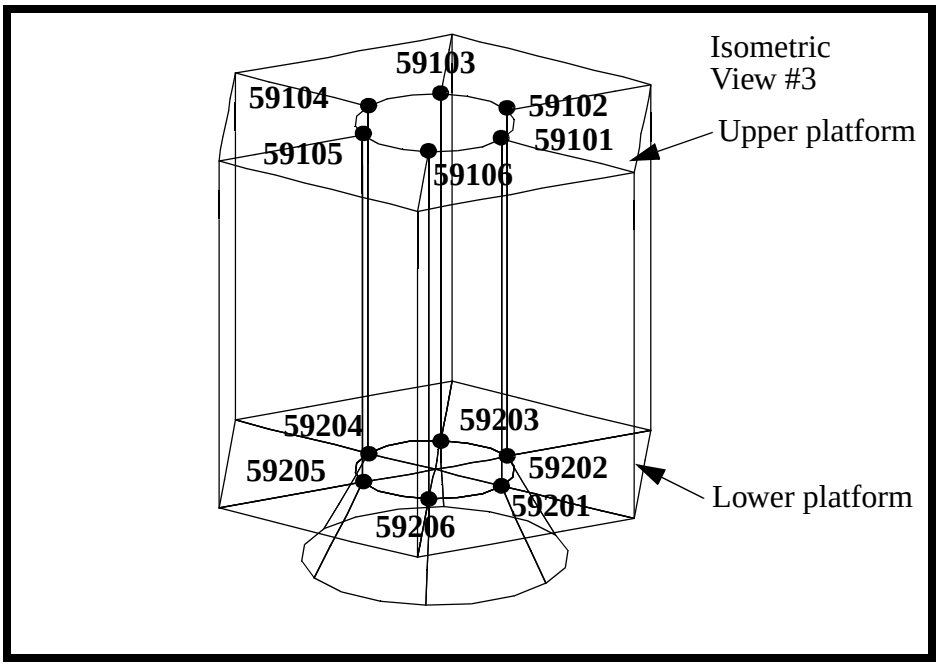
Reset Graphics

Create the FEM for the Science Platforms

13. We will now model the concentrated masses. The Science Platforms, Propulsion Module and Navigational Platform will be idealized as concentrated masses. Each set of masses will be put into a group, which will aid 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 intersect as shown in Figure 4.10. Additionally, this figure shows the element ID specification for these masses.

Figure 4.9. The mass locations and IDs 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
Lower Platform

Note: You need to hold the *Control key* down to select two groups at the same time.

Apply

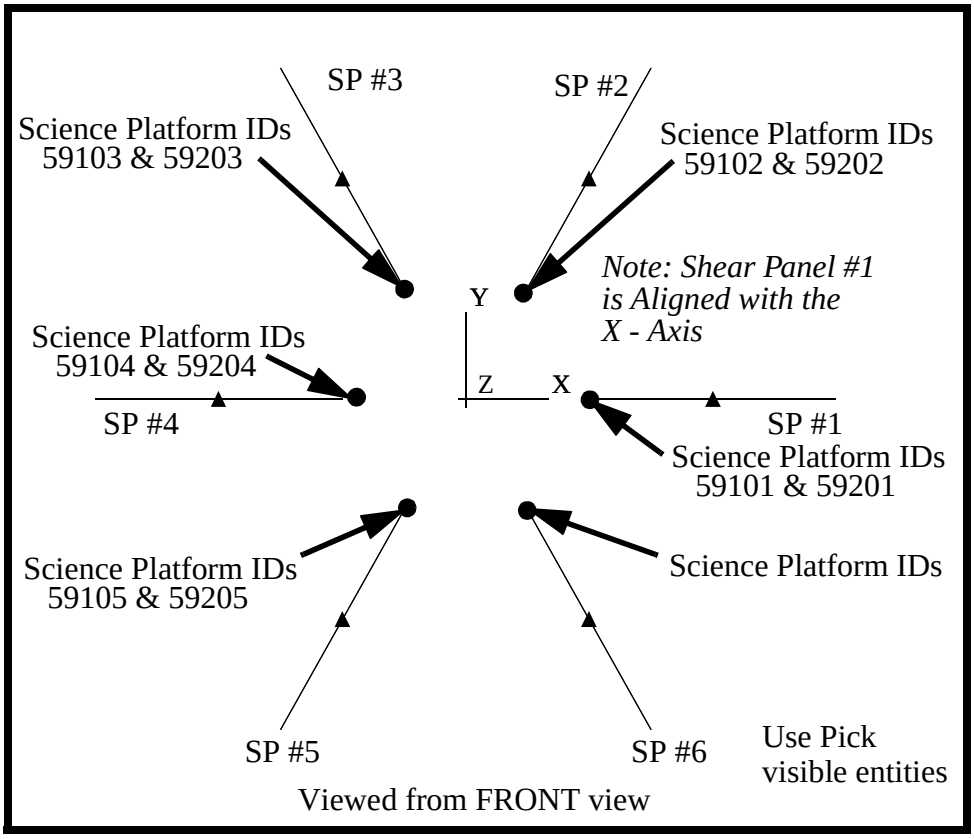
Cancel

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

Front view

Now create 12 Point Elements. These elements will be used to model the masses which are located at the vertices, where the Shear Panels, Platforms and Central Cylinders intersect as shown in the Figure 4.9 and 4.10.

Figure 4.10. The Shear Panel numbering as viewed from Front view. These will be used to define the IDs for Science Platforms.



We will first create the 6 Point Elements located on the Upper Platform (ID's 59101, 59102,..., 59106).

Action:	**Create**
Object:	**Element**
Method:	**Edit**
Element ID List	**59101**
Shape:	**Point**
Topology:	**Point**
Node 1 =	See Figure 4.10

Repeat this step for the other 5 elements located on the Upper Platform. Also note, the Auto Execute is ON, therefore you do not need to hit Apply.

Unpost the Upper Platform group and post the Lower Platform group as follows.

Group/Post...

Action: **Post**

Select Groups to Post

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

Apply

Cancel

Turn the model around 180 degrees using the toolbar icon Rear View and keep pick visible entities turned on.

Front view

Now repeat the steps to create the 6 Point Elements associated to the Lower Platform using the ID range 59201, 59202, ... 59206..

Turn Pick visible entities off, and rotate model using toolbar icon Front view

Front view

Create the FEM for the Propulsion Platform

14. We will now model the concentrated mass and rigid element for the Propulsion Block. The Propulsion Block will be put into a group, which will aid us when we do the Property assignment.

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 (49001) located at the center of the Lower Platform (reference figure 4.11).

Action: **Create**

Object: **Element**

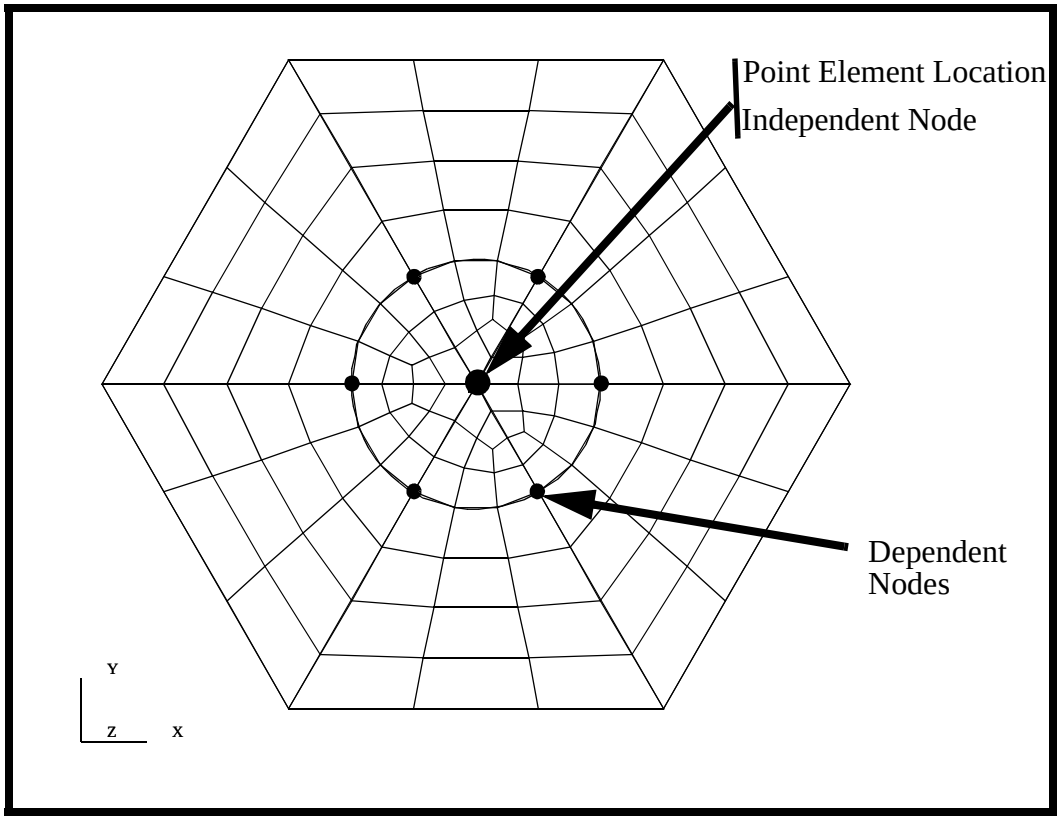
Method: **Edit**

Element ID List **49001**

Shape:	**Point**
Topology:	**Point**
Node 1 =	Node located at the center of the Lower Platform

Next, we will create 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 4.11. The Independent and Dependent nodes.



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

Action:	**Create**
Object:	**MPC**
Type:	**Rigid (Fixed)**
MPC ID	**49002**

Define Terms...

To better work with the form, you will want to move the form so it does not obscure your viewport and turn off the Auto Execute button in the define terms form.

Auto Execute

Create Dependent

Node List: See Figure 4.11

Note: Hold the *Shift key* down and then select all the Dependent Nodes

Apply

Create Independent

Node List: See Figure 4.11

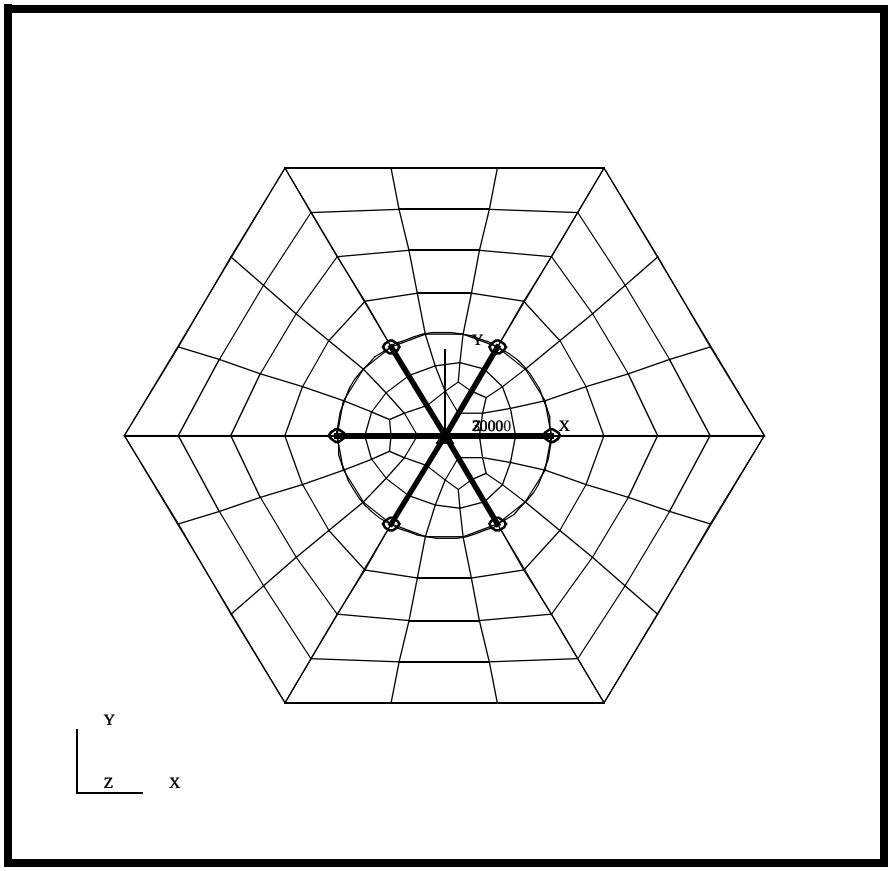
Apply

Cancel

Apply

Your model should now look like the figure below.

Figure 4.12. Lower Platform with Propulsion Block modeled.



Create the FEM for the Navigational Platform

15. We will now model the concentrated mass and Bar elements for the Navigational Platform. The Navigational Platform will also be put into a group, which will aid us when we do the Property assignment.

The Navigational Platform (NP) is located at the center of the Upper Platform. This Point was create by the mesher when we associated the point to the surface. The NP attachment 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 Shear Panel/Central Cylinder/Upper Platform intersection point.

We will create a Navigational Platform group that contains the Navigational 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 (39001) located at the center of the Upper Platform.

Action: **Create**

Object: **Element**

Method: **Edit**

Element ID List **39001**

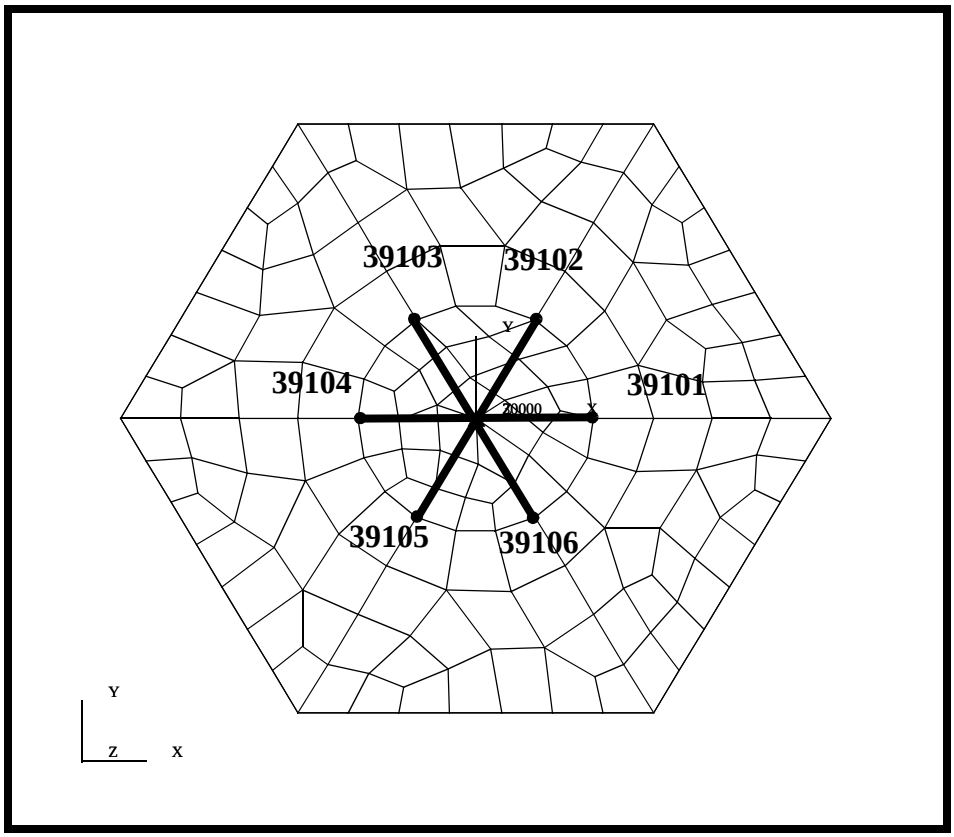
Shape: **Point**

Topology: **Point**

Node 1 = Node located at the center of the Upper Platform

Now create the attachment structure (ID = 39101 to 39106) going from the NP concentrated mass to each of the Shear Panel/Central Cylinder/Upper Platform intersection point. We will manually create Bar2 elements as shown in the Figure 4.13.

Figure 4.13. The Navigational Platform attachment structure. The Bar2 elements and IDs of the Upper Platform.



Action:	**Create**
Object:	**Element**
Method:	**Edit**
Element ID List	**39101**
Shape:	**Bar**
Topology:	**Bar2**
Pattern:	**Standard**
Node 1 =	Node located at the center of the Upper Platform

Node 2 = See Figure 3.13

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 for the remaining 5 bars and paying attention to the element numbering.*

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

Update the All Fem Group

Group/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**

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

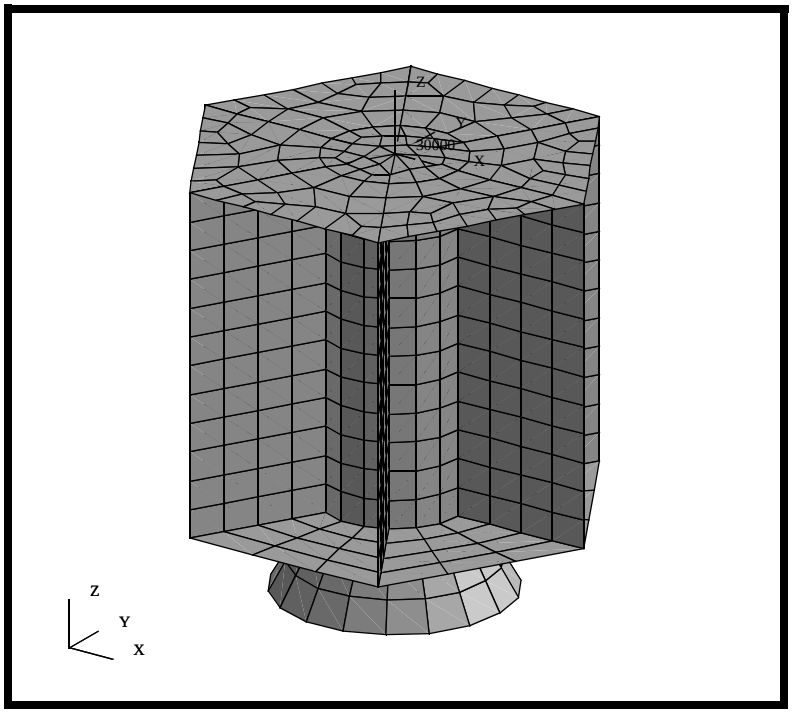
Change the view to Isometric view and the render style to Smooth shaded using the Toolbar icons.

Isometric View 3

Smooth shaded

Your model should be similar to the display below.

Figure 4.14. The Satellite FEM model.



Perform a Duplicate Element Check

17. 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.

$# There are NO duplicate elements in the model.

If you have duplicate elements, perform the following steps.

Test Control

Delete Duplicates

Test: **Delete Higher ID**

Apply

Close the Database and Quit MSC.Patran

18. 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.