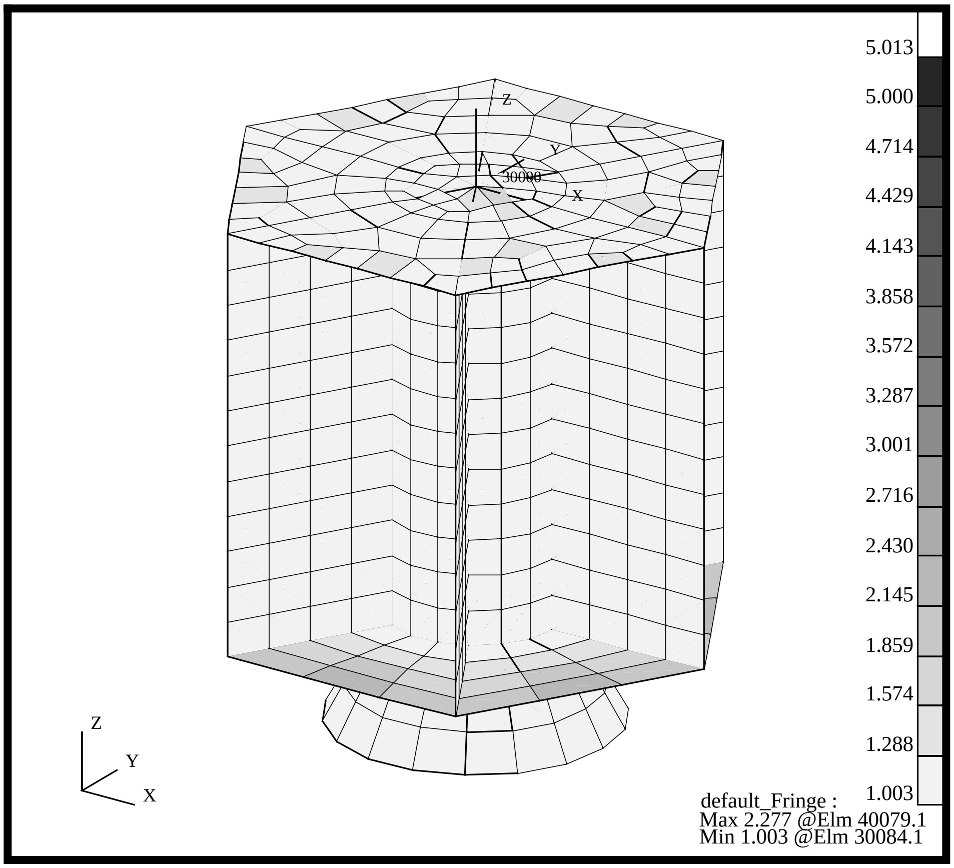
LESSON 4

Materials, Load Cases and LBC Assignment



Objectives:

- Perform a final verification of Quad elements.
- Prepare the model for analysis by creating material properties.
- Apply Loads and Boundary conditions and create two Loading conditions.

Model Description:

This exercise continues to prepare the Satellite model for analysis. You will check the Quad elements aspect ratio and define element and material properties. Finally, you will assign Loads and Boundary conditions to the model and setup two Loading conditions, one for a Modal analysis and one for a Static analysis.

Suggested Exercise Steps:

- Start MSC/PATRAN and open the **`satelite.db`** file.
- Verify the Quad element's aspect ratio using a threshold value of 5.0.
- Change the analysis coordinate system (Coord 20000) to the Launch CID by using FEM/Modify/Node.
- Create an Isotropic material, named **titanium**, which uses a Linear Elastic Constitutive Model. The material's Elastic Modulus, Poisson's Ratio and Density are 16E6, 0.16 and 0.27, respectively
- Create a Isotropic material named **aluminum**, which uses Linear Elastic Constitutive Model. The material's Elastic Modulus, Poisson's Ratio and Density are 10.5E6, 0.33 and 0.101, respectively.
- Create a simply supported displacement constraint, named **Launch constraint,** for all vertices on the outside edges of the Upper and Lower Platform. The displacement will be fixed in translation and free in rotation (i.e. ball joint).
- Define 3 inertial loadings according to the table below.

Load Name	**Trans Accel**	**Analysis Coordinate System**
Launch 1g + X	<386.4, 0, 0>	Coord 20000
Launch 1g + Y	<0, 386.4, 0>	Coord 20000
Launch 1g + Z	<0, 0, 386.4>	Coord 20000

- Create two Load cases with the following properties.

Load Case Name	**LBC Scale Factor**	**Loads & Boundary Conditions**
Launch Modal LC	1.0	launch constraint
Launch Static LC	1.0 2.0 2.0 -10.0	launch constraint Launch 1g + X Launch 1g + Y Launch 1g + Z

- 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.

Open the Satellite Database

File/Open...

Existing Database Name `satelite.db`

OK

2. Before we get started, let’s post the All Fem group.

Group/Post...

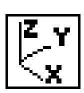
Select Groups to Post

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

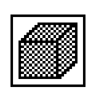
Apply

Cancel

Change the model view and render style using the following Toolbar.



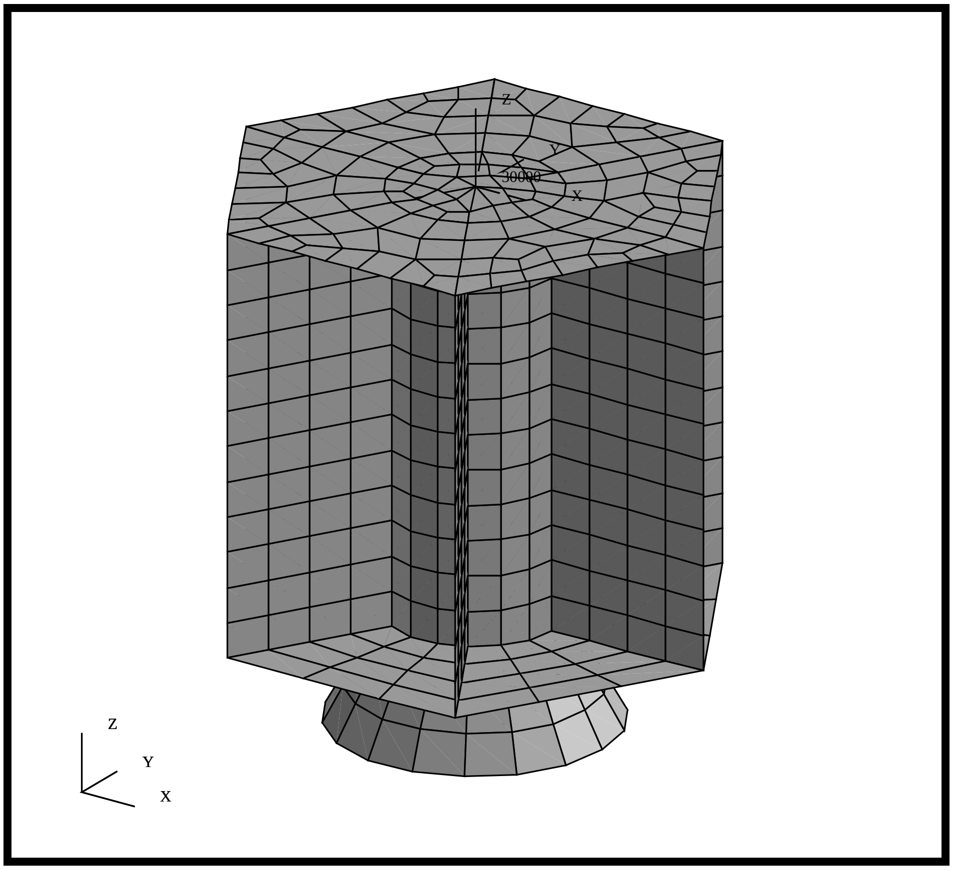
Isometric View 3



Shaded Smooth

Your model should be similar to the display below.

Figure 4.1. All Fem displayed for the Satellite



Aspect Ratio Verification

3. Verify the Quad element's aspect ratio using a threshold value of 5.0.

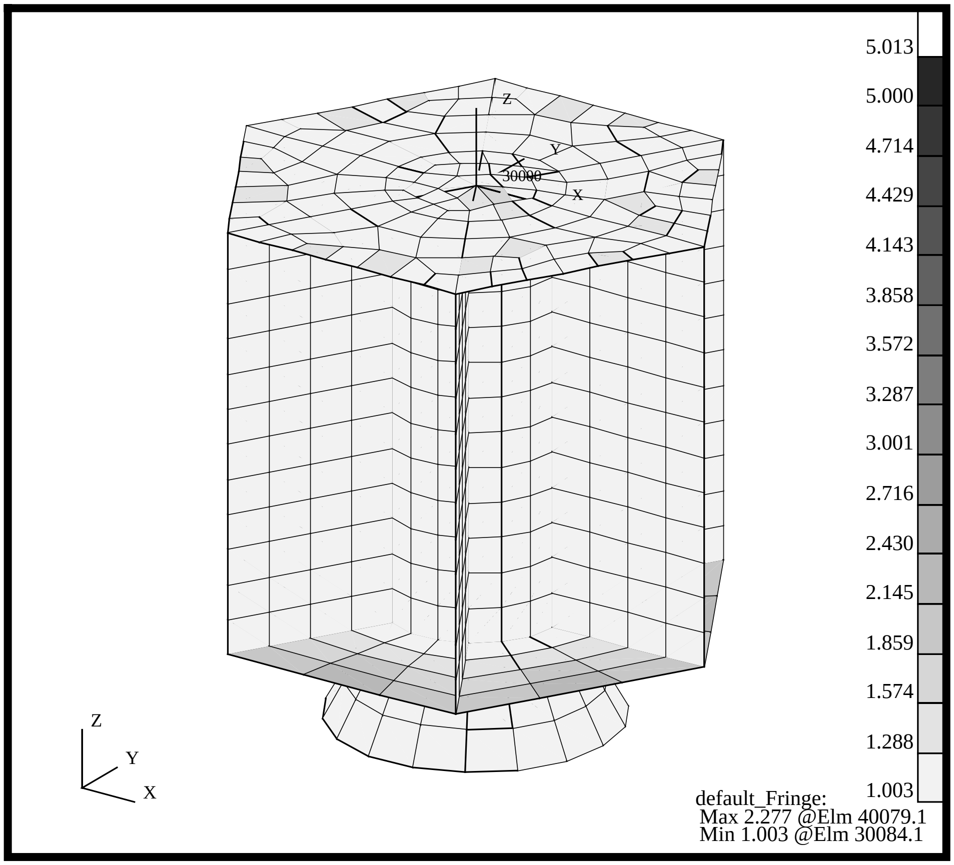
◆ **Finite Elements**

Action:	**Verify**
Object:	**Quad**
Test:	**Aspect**
Aspect Ratio	**5.0**

OK

This will color code the Quad4 elements based on their aspect ratio. Any element with an aspect ratio greater than or equal to your threshold value, 5.0, will be colored red according to the default spectrum.

Figure 4.2. The Quad4 Aspect Ratio Verification Plot.



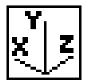
A summary of the minimum and maximum aspect ratio for the elements is located in the lower right hand corner of the viewport.

Reset Graphics

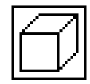
Modify Nodal Analysis CID

4. In this step, you will change the Nodal Analysis Coordinate System to the Launch coordinate system (CID 20,000).

Before we get started, let's re-orient the model in a more convenient orientation to pick the Analysis Coordinate System. Additionally, to better visualize the finite elements and coordinate frame, change the render style using the Toolbar icons.



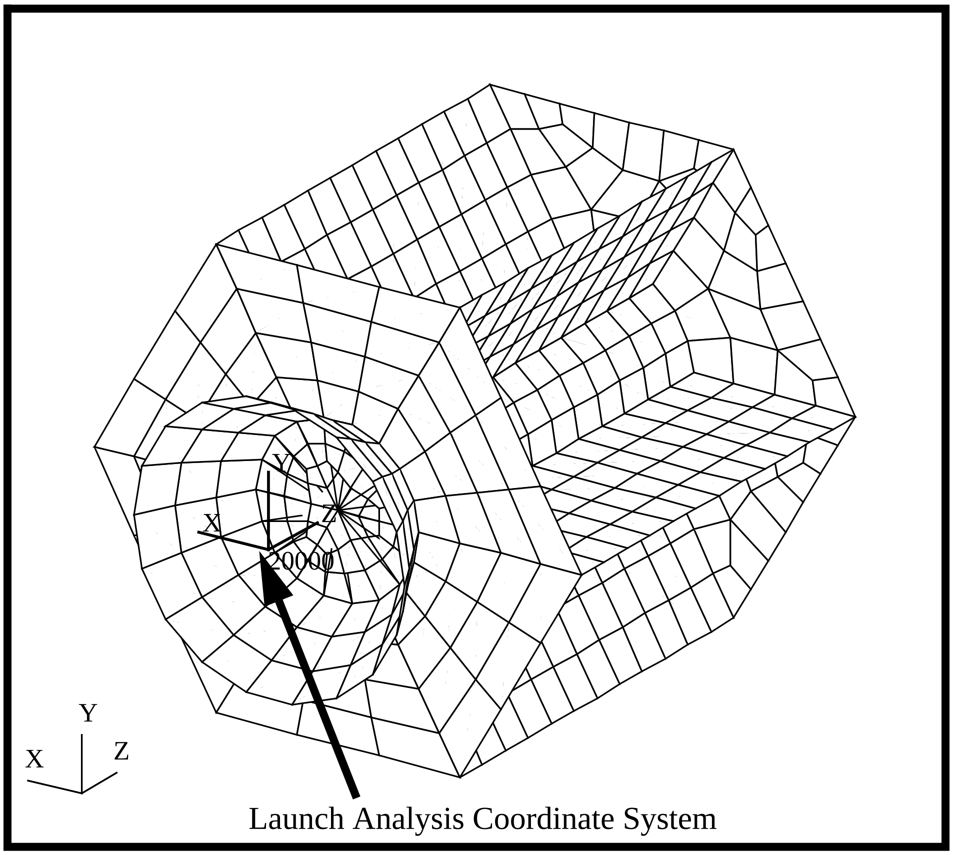
Isometric View 4



Hidden Line

Your model should now look like the following figure

Figure 4.3. Isometric View showing the Launch CID



Now, let's redefine the Analysis Coordinate Frame.

Action:	**Modify**
Object:	**Node**
Method:	**Edit**

Nodal Attributes	**Analysis Coordinate Frame**
Analysis Coordinate Frame	**Coord 20000**
Node List	Select ALL nodes in the current viewport

Apply

We have now completed all the work that we need to do on the model in Finite Elements section. Let's define the materials and any loadings that will be required to model a Launch configuration.

Create Titanium & Aluminum Material Properties

5. Create an Isotropic material, named **titanium**. The material's Elastic Modulus, Poisson's Ratio and Density are 16E6, 0.27 and 0.16, respectively.

Materials

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name	**titanium**

Input Properties...

Constitutive Model:	**Linear Elastic**
Elastic Modulus =	**16E6**
Poisson Ratio =	**0.27**
Density =	**0.16**

Apply

Cancel

You will know the model has been created when the Current Constitutive Model list is updated. Now, let's create a second material, named **aluminum**. The material's Elastic Modulus, Poisson's Ratio and Density are 10.5E6, 0.33 and 0.101, respectively.

Material Name	**aluminum**

Input Properties...

Constitutive Model:	**Linear Elastic**
Elastic Modulus =	**10.5E6**
Poisson Ratio =	**0.33**
Density =	**0.101**

Apply

Again, you will know the model has been created when the Current Constitutive Model list is updated.

Cancel

Define the Launch Constraints

6. Let's create all the Load and Boundary Conditions that will be needed to model a Launch condition for the Satellite. We will start by defining the Launch constraints.

We will define two Loading conditions. The first will be a constraint set and the second will be the inertial loadings. Let's start by posting the groups that need to be defined the displacement conditions, the Upper and Lower Platforms.

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

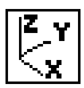
When one group is posted, MSC/PATRAN assumes that group is the current group. However, when more than one group is posted, MSC/PATRAN will require you to define a current group. The current group is used to collect any new entities created. In this step, we will not be defining any new geometry or finite elements. We will select the Lower Platform as the current group.

Current Group Selection

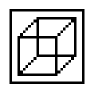
Adapter
All Fem
All Geometry
Central Cylinder
Lower Platform
Navigational Platform
Propulsion Block
Science Platforms
Shear Panels
Upper Platform
default_group

OK

Change the view and render style of the model using the Toolbar icons.



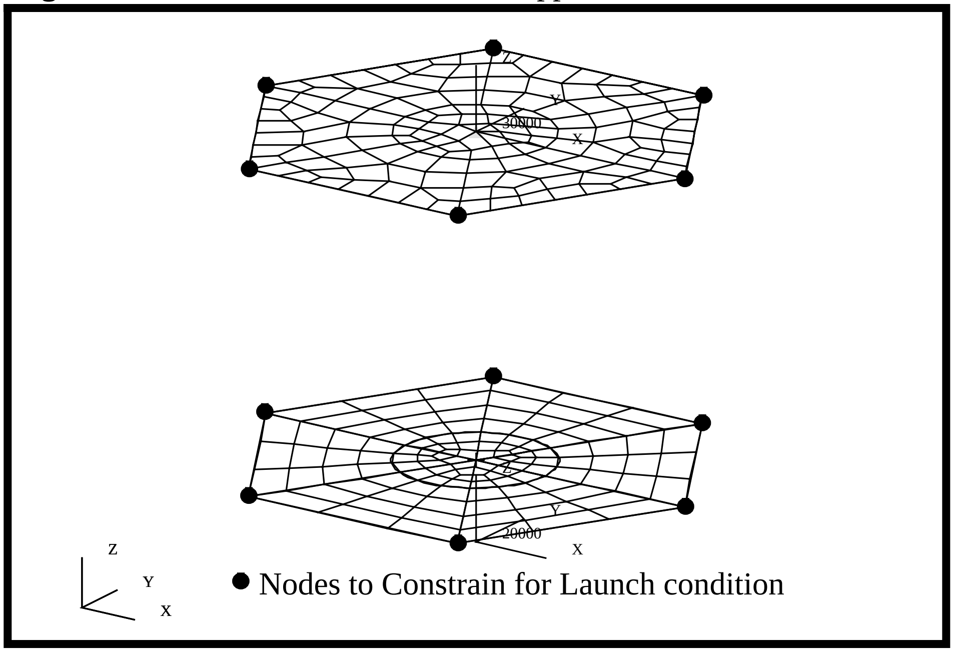
Isometric View 3



Wireframe

Finally, your model should now look like the figure below

Figure 4.4. Constraints for satellite on Upper and Lower Platform



Define the Launch constraints for the Satellite.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Displacement**
Type:	**Nodal**
New Set Name	**Launch constraint**

Input Data...

Translations <T1 T2 T3>	**<0, 0, 0>**
Rotations <R1 R2 R3>	**< >**
Analysis Coordinate Frame	**Coord 20000**

OK

Select Application Region...

Geometry Filter ◆ **FEM**

Select Nodes

Select nodes shown in Figure 4.4

Note: You need to hold the *Shift key* down to pick all the nodes at the same time.

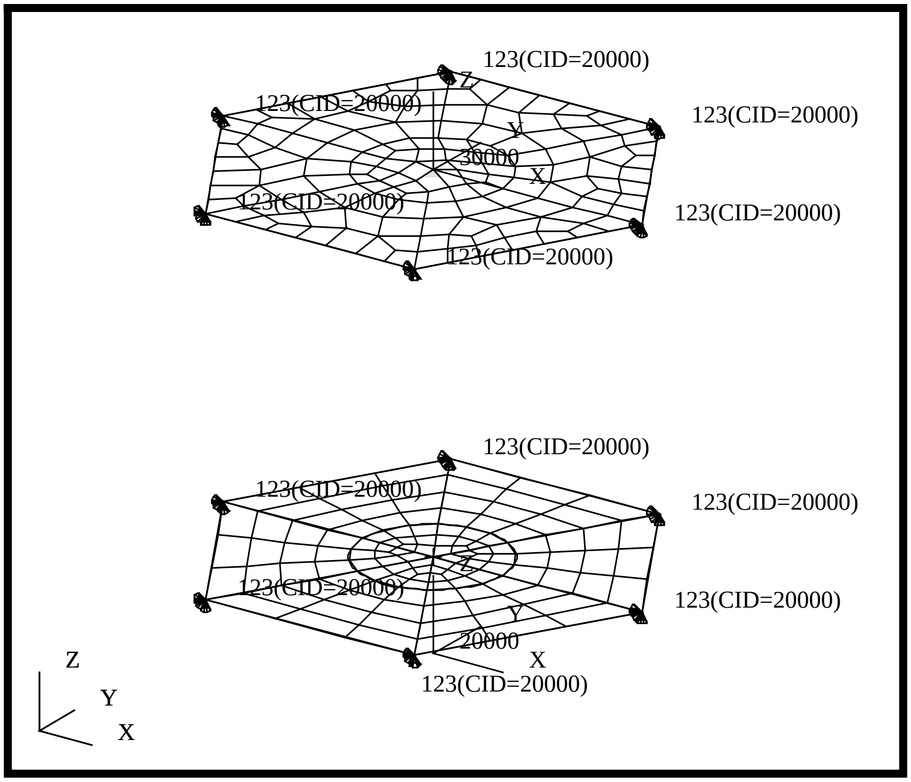
Add

OK

Apply

Your model should now look like the figure shown

Figure 4.5. Launch Constraints applied to the satellite.



Define the Launch Acceleration

7. We will create an inertial loading due to the Launch accelerations. We will define three separate loadings, a unit *G* load in the X, Y and Z directions. In a later step, we will use create subcases that are scaled multiples of these loadings.

We will define 3 inertial loadings as shown in the table below. It should be noted, the gravity constant for this problem is 1 *g* = 386.4.

Load Name	**Trans Accel**	**Analysis Coordinate System**
Launch 1g + X	<386.4, 0, 0>	Coord 20000
Launch 1g + Y	<0, 386.4, 0>	Coord 20000
Launch 1g + Z	<0, 0, 386.4>	Coord 20000

We will start by creating a unit *g* load in the X direction. The inertial loading is applied to the entire model. Since only one possible Application Region (All Finite Elements); therefore, there is no need to define it.

Action:	**Create**
Object:	**Inertial Load**
Type:	**Element Uniform**
New Set Name	**Launch 1g + X**
Input Data...	
Trans Accel <A1 A2 A3>	**<386.4, 0, 0>**
Rot Velocity <w1 w2 w3>	< >
Rot Accel <a1 a2 a3>	< >
Analysis Coordinate Frame	**Coord 20000**
OK	
Apply	

Repeat the above step for the Y and Z directions according to the table above. Make sure to update the New Set Name and Translation Acceleration Vector.

Define Two Load Cases

8. Finally, we will define two Load cases for this model. One Load case that contains only the Launch constraints and to be used for Modal analysis. Another Load case will contains the Launch constraints and a combined set of inertial loadings, and to be used for Static analysis.

We will make use of the capability of scaling the LBC's. We will create the following load combination; where the gravitational vector is defined as <2, 2, -10 >. If we were making several load cases based on accelerations, this procedure would be very handy.

Load Case Name	LBC Scale Factor	Loads & Boundary Conditions
Launch Modal LC	1.0	Launch Constraint
Launch Static LC	1.0 2.0 2.0 -10.0	Launch Constraint Launch 1g + X Launch 1g + Y Launch 1g + Z

We will start by defining the Modal Load case, which will be used to determine the modal frequencies of the spacecraft in a launch configuration.

◆ **Load Cases**

Action:	**Create**
Load Case Name	**Launch Modal**
Load Case Type:	**Static**

Assign/Prioritize Loads/BCs

Selection Multiplier	**1.0**
Select Loads/BCs to Add to Spreadsheet	**Displ_Launch constraint**

The Displacement Launch condition should now have been added to your spreadsheet.

OK

Apply

Let's create the second Load case to model a <2, 2, -10> *G* static loading condition.

Load Case Name	**Launch Static**
Load Case Type:	**Static**

Assign/Prioritize Loads/BCs

The spreadsheet should already have the Launch constraint selected. We will now define the 3 inertial loadings.

Selection Multiplier	**2.0**
Select Loads/BCs to Add to Spreadsheet	**Inert_Launch 1g + X**
Selection Multiplier	**2.0**
Select Loads/BCs to Add to Spreadsheet	**Inert_Launch 1g + Y**
Selection Multiplier	**-10.0**
Select Loads/BCs to Add to Spreadsheet	**Inert_Launch 1g + Z**

The spreadsheet should now show the 3 inertial loading conditions along with the displacement constraint. Additionally, the spreadsheet also indicates the LBC scale factor.

OK

Apply

Close the Database and Quit Patran

9. 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.