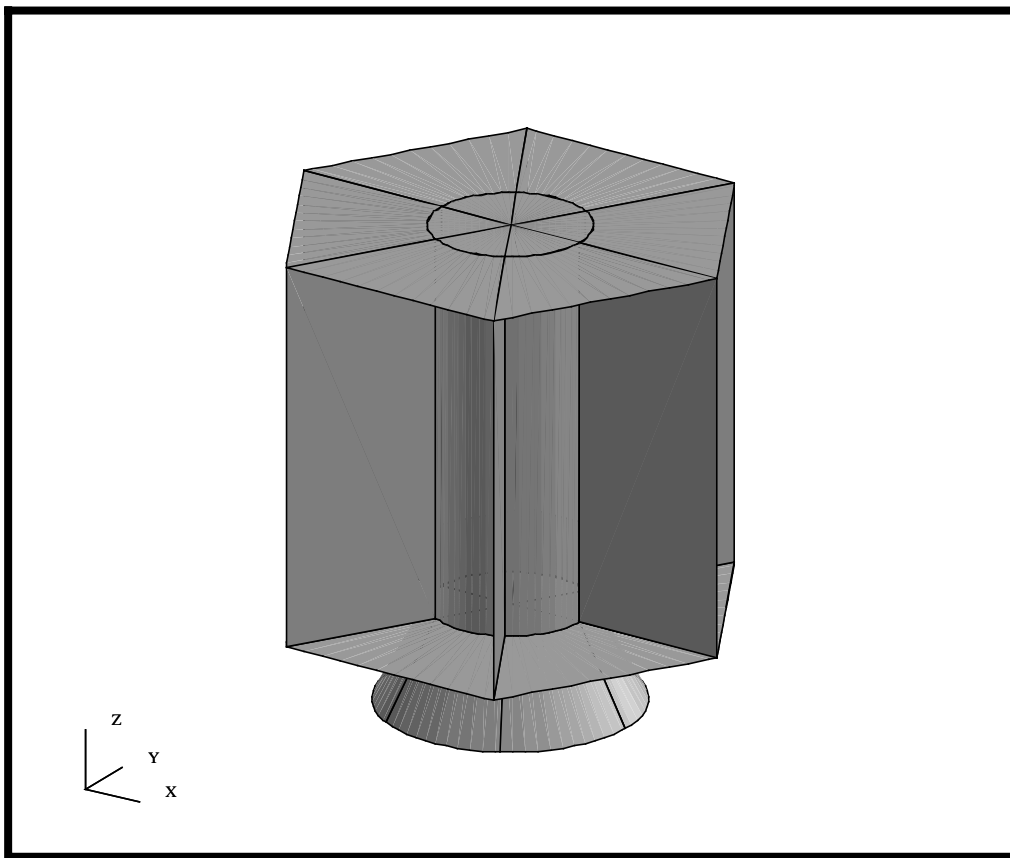


---

## LESSON 6

# *Static and Normal Mode Analysis of a Space Satellite*



### **Objectives:**

- Set up and analyze the Satellite model for a Normal modes and Static analysis.
- Learn to modify the default subcase parameters, solution parameters and output requests.



## Model Description:

In this exercise, we will set up and analyze two analyses for the Space Satellite. One analysis is Linear Static and another is a Normal Modes analysis. PATRAN provides a set of default parameters and output requests for each solution. We will additionally define the following MSC/NASTRAN parameters from PATRAN:

```
param, WTMASS  
param, K6ROT  
param, COUPMASS (MODAL ONLY)
```

We will be making additional output requests and defining other solution parameters, such as the number of modes. We will be requesting two different results formats, an OP2 and XDB. We will be discussing the differences for these in a later exercise. Once the subcases are properly setup and chosen, we will submit the analysis directly from PATRAN.

## Suggested Exercise Steps:

- Start MSC/PATRAN and open the `satelite.db` file.
- Choose a linear static analysis and define a weight mass conversion of 0.00259 and a rotational plate stiffness of 10.
- Select the Launch Static subcase for analysis.
- Submit the Launch Static subcase for analysis and monitor the solution.
- Choose a normal modes analysis and define a weight mass conversion of 0.00259, a coupled mass matrix, and a rotational plate stiffness of 10.
- Modify the Launch Modal subcase to choose the first 6 modes.
- Select the Launch Modal subcase for analysis.
- Submit the Launch Modal subcase for analysis and monitor the solution.
- Close and Exit MSC/PATRAN.

---

## Exercise Procedure:

---

### Open the Satellite Database

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

#### File/Open...

Existing Database Name

satelite.db

OK

---

### Setup the Linear Static Analysis

2. We are now ready for analysis. In this step, you will define the analysis solution type and parameters, a Linear Static analysis. In later steps, we will setup the subcases, write out an Input deck and run the analysis. We will be submitting two analyses, a Static and Normal modes solution.

We will begin by defining the subcase for the static analysis launch condition. For the first analysis, the linear static launch condition, we will request a text format of an *op2* file. Additionally, we will be defining a plate rotation stiffness and a weight mass conversion that is to be applied to all density, non-structural mass and lumped mass (MSC/NASTRAN K6ROT and WTMAS parameters, respectively).

#### ◆ Analysis

Action:

Analyze

Object:

Entire Model

Method:

Full Run

Job Name:

satellite\_static

#### Translation Parameters...

OUTPUT2 Format:

Text

OK

#### Solution Type...

Solution Type:

◆ LINEAR STATIC

#### Solution Parameters...

Plate Rz Stiffness Factor =

10.0

Wt.-Mass Conversion =

0.00259

OK

OK

3. Review the Subcase for the linear static launch condition. We will be using the subcase based on the load cases created earlier.

In MSC/NASTRAN, a subcase provides a tool to associate loads and boundary conditions, output requests and various other parameters depending on the solution type selected. These subcases are essential to perform structural and thermal analysis of structures.

MSC/PATRAN automatically generates one subcase for every Load Case created. For this analysis, we will be using the subcase that was automatically created for the “Launch Static” load case. In addition, each solution type also has an associated “Default Subcase” which corresponds to a default load case.

For this analysis, we will be using the *Launch Static* subcase that was automatically generated. If you wish to access the Subcase information, you can click on the **Subcase Create...** button. You will notice the *Default*, *Launch Modal* and *Launch Static* as Available Subcases. These have automatically been created. If you select any of these subcases, you will notice that each subcase corresponds to a Load Case with the same name. Each automatically generated subcase has a standard set of requested outputs and solution parameters. For a static analysis, the Stress, Constraint Forces and Displacements have already been chosen. These can be inspected by pressing the *Output Requests* button inside the **Subcase Create...** form.

4. We will now select the subcases to analyze.

The subcases that will be analyzed are listed in the **Subcase Select...** form. Selected subcases appear in the *Selected Subcases* list box. To select a subcase, pick any subcase listed in “*Subcases For Solution Sequence: 101*” listbox. To de-select a subcase, pick on it from the Subcases Selected listbox. For this analysis, we will only be analyzing the *Launch Static* load case. Therefore, you will need to de-select the Default subcase and select the Launch Static subcase.

#### Subcase Select...

*Subcases For Solution  
Sequence: 101*

**Launch Static**

(Click on this to select)

*Subcases Selected:*

**Default**

(Click on this to deselect)

**OK**

**Review the  
Static  
Subcase**

**Static  
Analysis  
Setup**

---

## Submit and Monitor the Analysis

5. Submit and monitor the analysis.

To submit the analysis, just press Apply in the main Analysis form.

### Apply

When the job is finished, we will use these results to post-process the analysis.

If you have Analysis Manager installed on the system, you will see a Graphical User Interface (GUI) which shows you how the job is proceeding, errors, CPU and Disk usage. Look for any Errors in the Mon File window. When the job is finished, simply Close the Analysis Manager Job Graph window.

### Close

If you do not have Analysis Manager installed, you can monitor the analysis by opening a shell on the same system/directory location. In UNIX, you can type in the following command:

```
tail -l satellite_static.f06
```

This will display any messages normally output to the *F06* file to the shell while the job is running. These messages will also tell you the progress of the job and whether the analysis was successful or not.

Regardless of which method you used to submit and monitor the analysis, look for any errors that may have occurred. If you have any errors, ask the instructor for assistance. If the job completes normally, proceed onto the next step to create a modal analysis of the Space Satellite.

6. We will now set up and submit a normal modes analysis for the vehicle in the launch (constrained) condition.

We will be creating an analysis deck for a capturing the first 6 modes of the Satellite in a launch (constrained) configuration. We will also be requesting the analysis be written to an *XDB* file as opposed to an *op2* file. Additionally, we will be defining a coupled mass matrix formulation, a plate rotation stiffness term and a weight mass conversion that is to be applied to all density, non-structural mass and lumped mass (MSC/NASTRAN COUPMASS, K6ROT and WTMASS parameters, respectively).

### ◆ Analysis

Action:

Analyze

Object:

Entire Model

---

## Setup a Modal Analysis

*Method:*

*Job Name:*

*Data Output:*

*Solution Type:*

*Mass Calculation:*

*Plate Rz Stiffness Factor =*

*Wt.-Mass Conversion =*

- Define the Subcases for a modal analysis of the Satellite in the launch (constrained) condition. We will be using the subcase based on the load cases created earlier. We will also be requesting the first 6 modes of the structure.

**Define and Select the Modal Subcase**

**Subcase Create...**

*Available Subcases*

Default
<b>Launch Modal</b>
Launch Static

*Subcase Options*

*Number of Desired Roots =*

---

You may get the following message asking if you wish to overwrite the Launch Modal loadcase that was created automatically, if you do, answer Yes.

*Subcase "Launch Modal" already exists. Do you wish to delete the existing subcase and create a new one?*

**Yes**

Select Cancel to close the form and Subcase Select to select the subcase for analysis. Note, you will need to first de-select the Default Subcase.

**Cancel**

**Subcase Select...**

*Subcases Selected:*

**Launch Modal**

**OK**

---

## Submit and Monitor the Analysis

8. Submit and monitor the analysis.

To submit the analysis, just select Apply in the main.

**Apply**

Monitor the analysis using the same approach presented above. Again, if you have any problems, please ask the instructor.

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.

---

## Close the Database and Quit Patran