PRATHAM

IIT BOMBAY STUDENT SATELLITE

Preliminary Design Report

Structures

By

Niranjan Parab

Ramanath Pai

Ankit Chiplunkar

Department of Aerospace Engineering,

Indian Institute of Technology, Bombay

March, 2009

Table of Contents

List of Figures

List of Tables

Chapter 1

Introduction

Pratham is student satellite to be designed and fabricated by students of Indian Institute of Technology Bombay. Pratham will be a nano-satellite with weight 5-10 Kg and dimensions around 270mm x 270mm x 270mm. The purpose of Pratham is to measure the total electron count of ionosphere and perform the tomography of ionosphere.

The requirement that structure must satisfy is that it must house all the components and ensure that it does not fail during the lifetime of the satellite. During the lifetime, the satellite will undergo various types of loading. Structural subsystem should ensure that satellite structure is able to withstand the loading and no component should fail due to structural loading.

The design of satellite structure is dependent on many factors. Some of the most important factors are placement of components and the material properties of components. The design approach that is followed in this satellite is first system engineering team prepares a configuration layout and structures team analyzes it and decides the parameters like material of satellite body and thickness of the material. The analysis is given to system engineering team and there is some flaw in the system, system engineering team prepares a modified configuration layout. This iterative process is followed till a design satisfying both system engineering and structural requirements is obtained. The various parameters that can be changed by structural subsystem are material properties of structure, geometric parameters like thickness of the structure and joining mechanisms.

Finite element software ANSYS is used for structural analysis. A structural model is prepared and meshed to get the finite element model. Various static as well as dynamic analyses are performed on this finite element model to get the response of the structure to various types of loading. ANSYS can be used to perform both static as well as dynamic analyses on the structure. Validation of the results obtained using ANSYS is done by first validation of the finite elements and geometry by comparing results with theoretical results and then by analysis of individual structural element in ANSYS isolated. This approach was giving a good match between the results obtained from finite element analysis and theoretical results as well as between individual element and entire structural model. The results obtained using FEA suggest that satellite will be above the failure stresses and strains when specified loads are applied.

1.1 Overview of report

In chapter 2, requirements and constraints are given. Chapter 3 describes the configuration layout. In chapter 4, design approach is presented and in chapter 5, analysis approach is presented. Qualification and testing plan is given in chapter 6. Conclusion is given in chapter 7.

Chapter 2

Requirements

2.1 System requirements:

1. All components should be properly housed in the satellite structure. Placement of all components should be proper and there should be no interferences due to other components.

2. Satellite structure should be able to withstand the loads during launch. All the components should be safe and working after the launch.

3. Satellite structure should be able to withstand the thermal loads arising in the orbit.

2.2 Subsystem requirements:

2.2.1 Launch Vehicle placement requirements:

1. Launch vehicle interface requires 8 no's M6x1, 9mm long helicoil inserts at 230mm PCD on bottom deck of the satellite.

2. There should not be any interference in the joint from the satellite to the launch vehicle body.

2.2.2 Launch Loading requirements:

During launch, the satellite will be subjected to various kinds of loading. The loading specifications for which the launch vehicle interface is tested is assumed to be the loading data for the satellite during launch as the actual loading data will be available only when launch is fixed. The loading data used in analyses is:

1. Quasi static loading:

   Longitudinal quasi static loads: (+-) 11g

   Lateral quasi static loads: (+-)6g

2. Frequency Response:

(a) Harmonic Loading:

Longitudinal direction:

Table 1: Harmonic loading

Frequency	*Acceptance loading*
5-10 Hz	8mm (DA)
10-100Hz	3.75g

Lateral direction:

Table 2: Harmonic loading

Frequency	*Acceptance loading*
5-10Hz	8mm (DA)
10-100Hz	2.25g

(b) Random vibrations:

Table 3: Random vibrations

	qualification	Acceptance
Frequency	PSD(g^2/Hz)	PSD(g^2/Hz)
20	.002	.001
110	.002	.001
250	.034	.015
1000	.034	.015
2000	.009	.004
g RMS	6.7	4.47

2.2.3 Stiffness requirements:

Satellite should have natural frequency more than 90Hz. This requirement can be translated to the component requirement of having natural frequency more than 175Hz for all the components.

2.2.4 in orbit requirements:

The satellite structure should be able to withstand the cyclic thermal loading in orbit. The orbit will give rise to differential thermal expansion and contraction. This differential expansion and contraction gives rise to thermal stresses. The structure should be able to withstand this loading.

Chapter 3

Configuration layout

The complete satellite internal and external configuration is shown in Figure 3-1 and Figure 3-2.

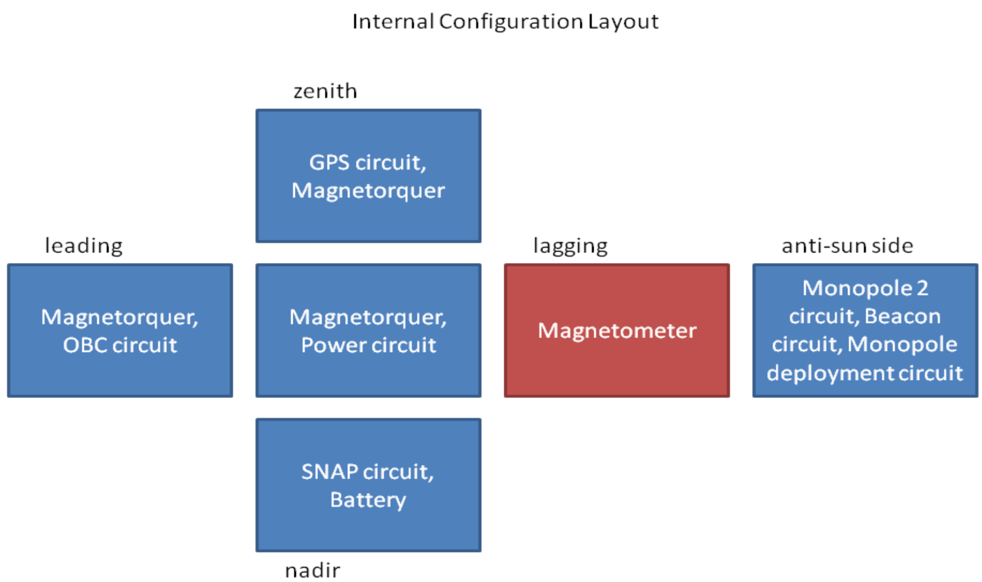


Figure 1: Internal configuration layout

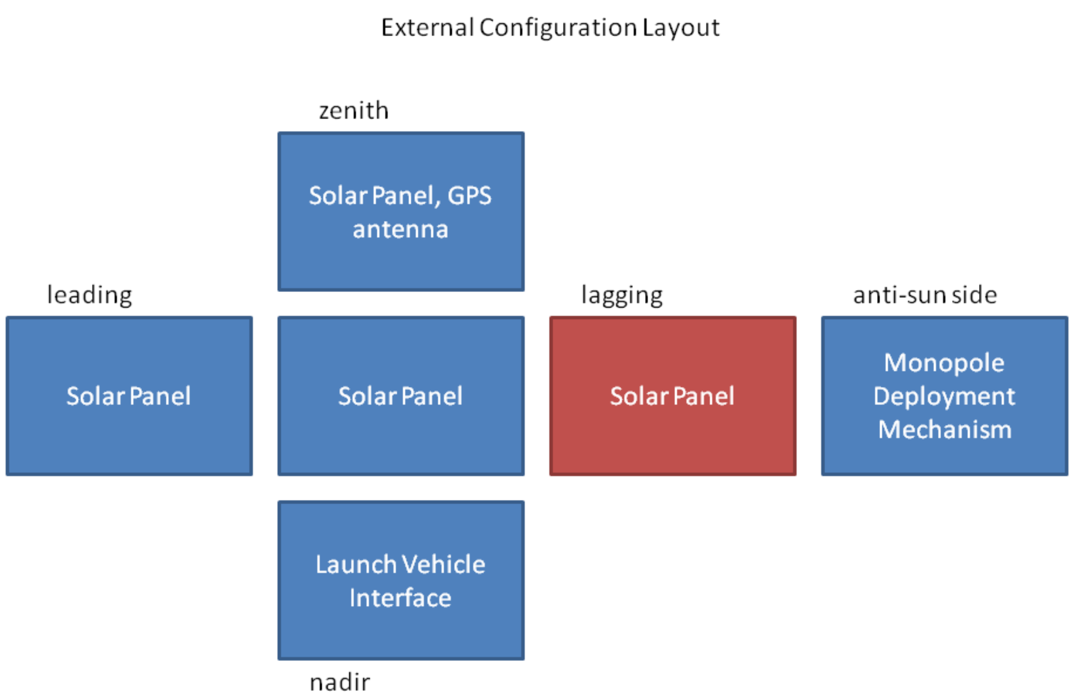


Figure 2: External configuration layout

The satellite structure modeled in CAD software solidworks is shown in figure 3-3 and the inside view is shown in figure 3-4.

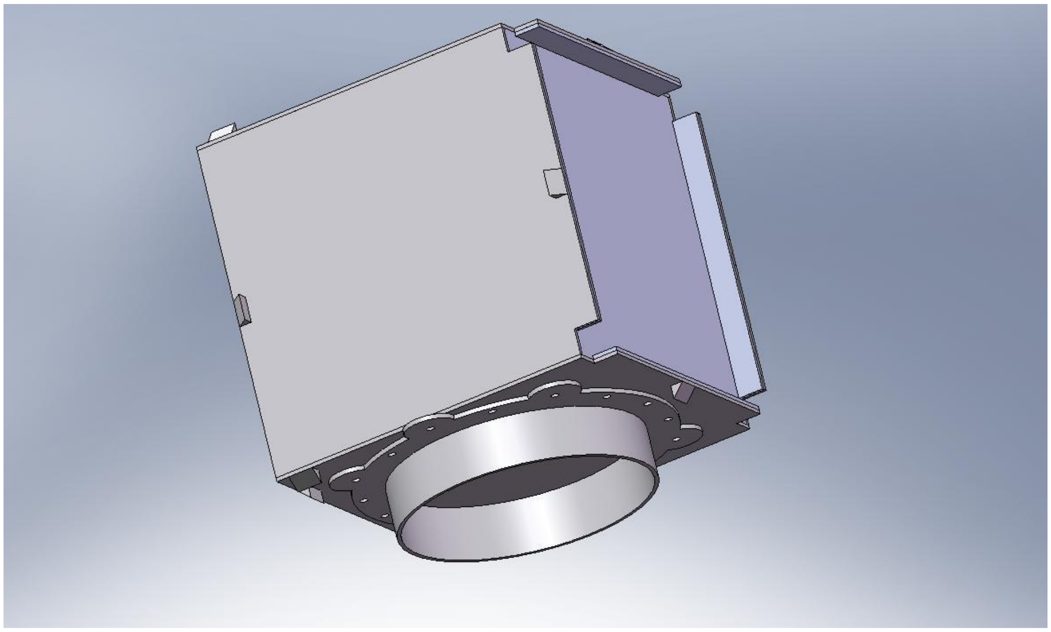


Figure 3: Solidworks model with LVI

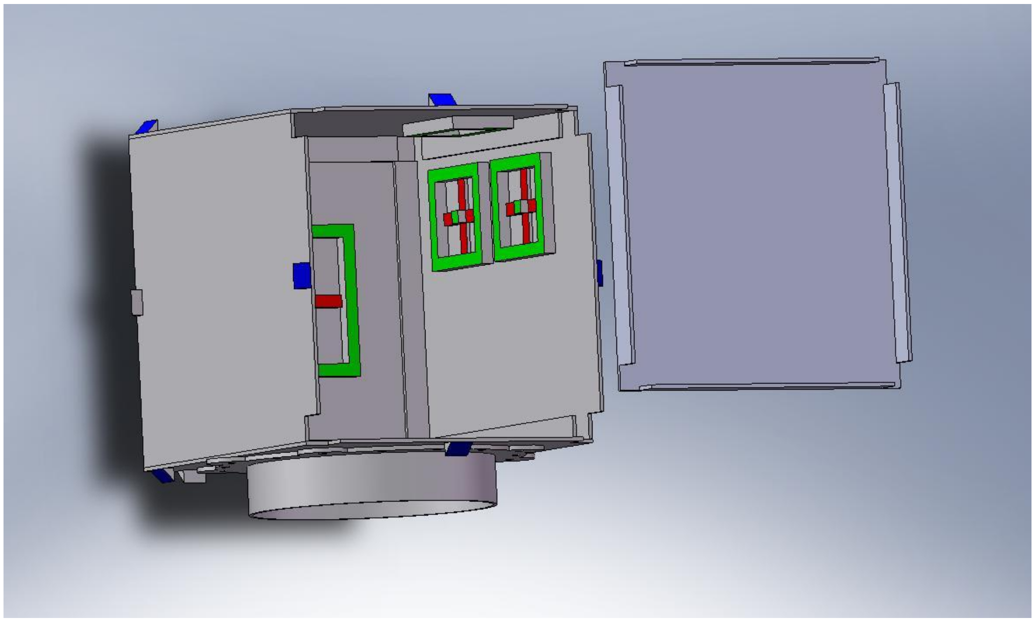


Figure 4: Solidworks model - inside view

Chapter 4

Design

In this chapter, the methodology of designing the structure of the satellite is explained. Various components of structure are presented and mass budget is also given. In the last section, the material properties of various materials used on the satellite are given.

3.1 Design approach

The design approach followed is given in a Figure 1. In the entire design process factor of safety used was 1.5.

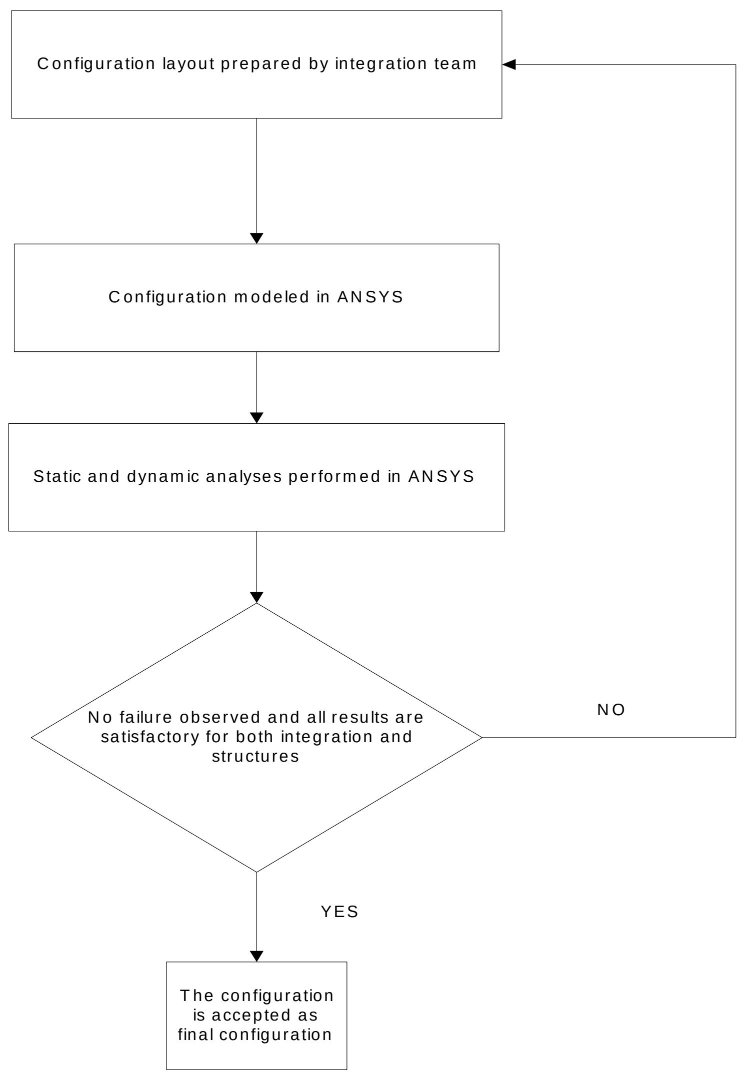


Figure 5: Design approach

3.2 Structural elements

The Satellite essentially consists of six structural members: the panels which make the box of the body. These sides are named Leading-side, Lagging-side, Sun-side, Anti-Sun-side, Zenith and Nadir depending on their position during orbit.

The zenith, Leading, Lagging and Sun-sides have solar panels mounted on them. The solar panels are connected to Aluminium honeycomb which in turn are connected to the main satellite body.

3.2.1 Structural elements:

Zenith: The zenith is the side of the satellite which always faces away from the earth. The Zenith side of PRATHAM incorporates one of the solar panels and the GPS antenna along with GPS circuit and a magnetorquer.

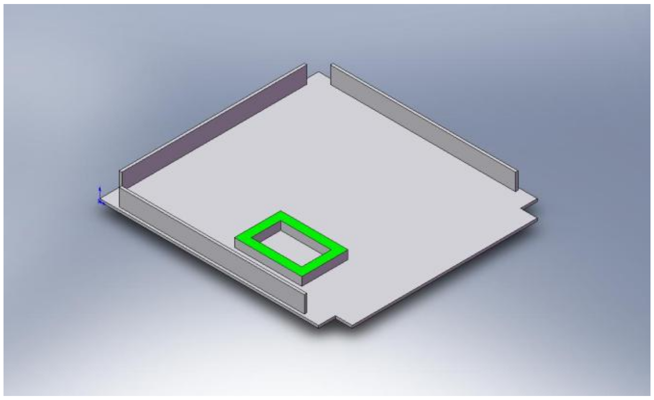


Figure 6: Zenith side

1. Nadir: The Nadir side always faces towards the earth. The Launch Vehicle interface which in our case is the IBL-230 V2 will be attached to Nadir side. It will also have a SNAP circuit and the battery.

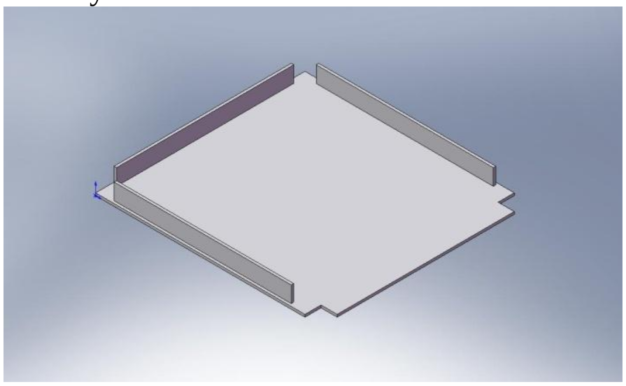


Figure 7: Nadir side

2. Sun-side: As the name suggests, the sun-side faces the sun. Our sun-side has a solar panel mounted on it. From the inside, it has the power circuit and a magnetorquer.

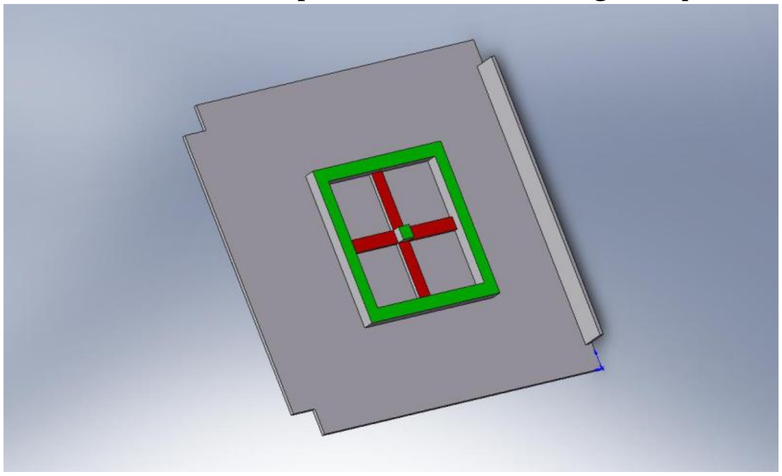


Figure 8: Sun side

3. Anti-Sun side: It is the side opposite to the Sun-side. The monopoles are attached to the anti-sun side. It has the monopole circuit and the beacon circuit.

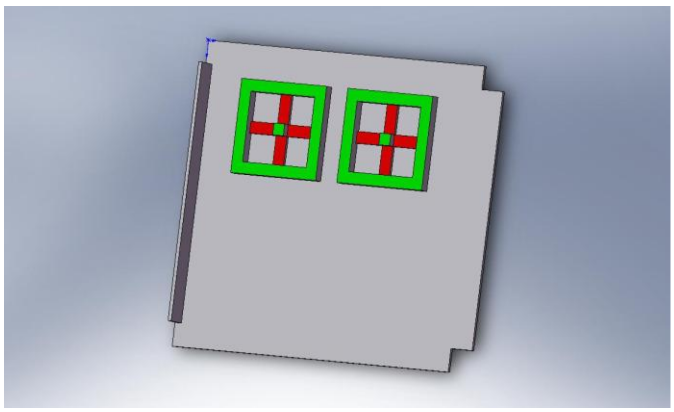


Figure 9: Anti sun side

4. Leading Side: The Leading side is normal to the direction of orbit of the satellite and has a solar panel mounted on it and has a magnetorquer and the OBC circuit.

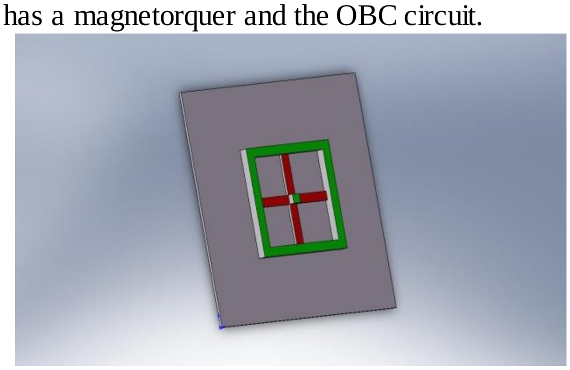


Figure 10: Leading side

5. Lagging side: The Lagging side is opposite the leading side. It contains the magnetometer and a solar panel.

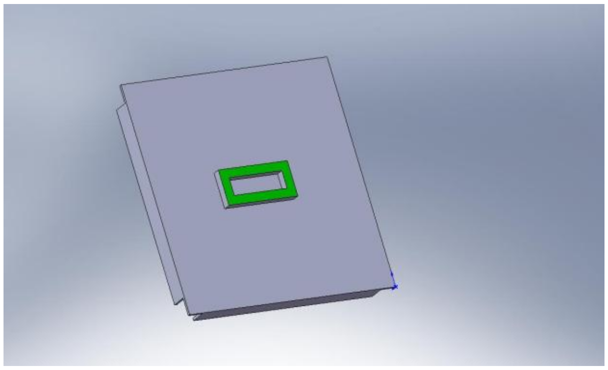


Figure 11: Lagging side

3.3 Mass budget:

The mass budget is presented in table 4-1.

3.4 Material properties:

The material properties used in the analysis of the structure are presented below. These material properties are obtained from various sources including books and websites. These material properties are not of the actual materials which will be used on the satellite.

Al 6061 T6:

Physical properties:
Density 2.70 g/cc

Mechanical Properties:

Table 4: Mechanical properties of Al 6061

Ultimate Tensile Strength	310 MPa
Tensile Yield Strength	276 MPa
Elongation at Break	12.0 %
Modulus of Elasticity	68.9 GPa
Notched Tensile Strength	324 MPa
Ultimate Bearing Strength	607 MPa
Bearing Yield Strength	386 MPa
Poissons Ratio	0.330
Fatigue Strength	96.5 MPa @# of Cycles 5.00e+8
Specimen Fracture Toughness	29.0 MPa-m½
Shear Modulus	26.0 GPa
Shear Strength	207 MPa

Aluminum honeycomb:

Aluminum honeycomb has variable properties depending on the geometry and density of the aluminum sheet from which it is manufactured. The typical values of moduli and strengths are given below for some configurations.

Table 5: Mechanical properties of Aluminium honey comb

PCGA-XR1 3003 Mechanical Properties								
PLASCORE® Honeycomb Designation			Bare Compressive		Plate Shear			
CORE TYPE	CELL SIZE	DENSITY	STRENGTH PSI	MODULUS KSI	STRENGTH PSI		MODULUS KSI	
					"L"	"W"	"L"	"W"
PCGA-XR1	1/4	5.2	620	148	345	215	63	31
PCGA-XR1	3/8	3.6	325	92	210	130	40	20
PCGA-XR1	1/2	2.5	165	40	130	70	25	15
PCGA-XR1	3/4	1.8	110	24	95	55	16	8
PCGA-XR1	1.0	1.4	75	16	55	40	14	7

Other components like mechanical harnesses, joints, screws and nut-bolts are yet to be finalized. Hence their material properties are not included in this report.

Chapter 5

Analysis

5.1 Modeling:

The satellite structure was modelled in ANSYS as shown in Figure 5-1. The joints in the panels were modelled using the "glue" function in ANSYS. In this model, the joint between the harness and panel is assumed to be continuous throughout the area of harness. This is contrary to the actual joint which will be at discreet points where the screws are placed. The antenna deployment mechanism is also not integrated into model.

The PCB modelling is done as only the board and the components on the PCB are not modelled. The rationale behind this is explained in later section.

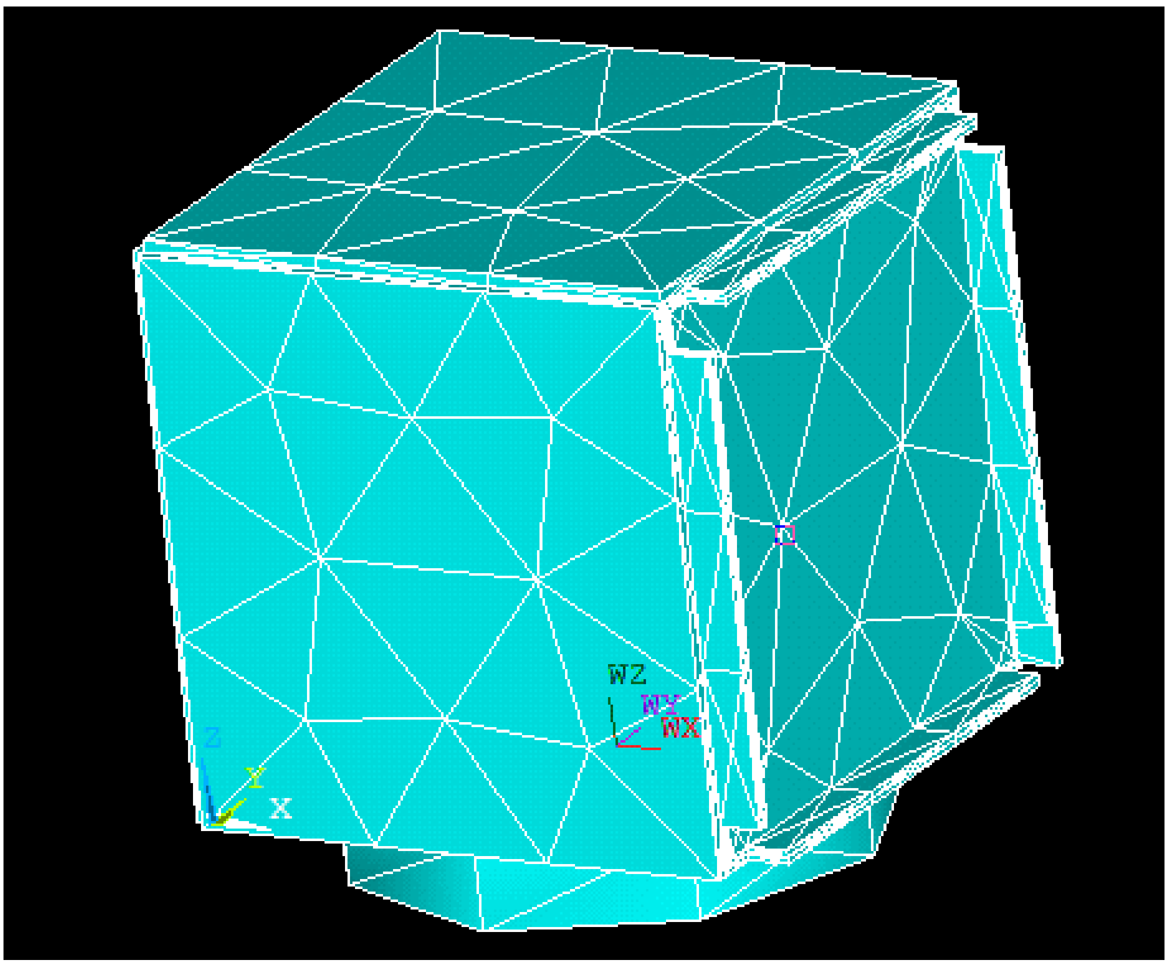


Figure 12: ANSYS model of satellite

The co-ordinate system used in the analysis is the Cartesian co-ordinate system with origin at one of the corner of the satellite. The longitudinal axis used for this analysis is along Y axis as the actual location and orientation of the satellite on the launch vehicle is not known now. The element used in ANSYS is 20 node-SOLID95 because it gives most accurate results for free mesh (see chapter-6).

Boundary conditions used for all analyses are same. The part of launch vehicle interface attached to the satellite is constrained in space. This is the correct assumption as in actual launch; it will be constrained using ball lock mechanism constraining all the degrees of freedom.

5.2 Results:

Results obtained in all the analyses are presented in this section. First the results of modal analysis are presented. Then results for static load cases are presented. In last section, results of frequency response of the structure are presented.

5.2.1 Modal analyses results:

The first 15 natural frequencies are tabulated in table 5-1.

Table 6: First 15 Natural Frequencies

SET	FREQUENCY
1	323.82
2	375.49
3	484.03
4	665.49
5	697.52
6	836.38
7	871.80
8	894.24
9	973.75
10	1245.4
11	1299.6
12	1351.1
13	1428.6
14	1452.4
15	1476.7

First 5 mode shapes are presented in Figure 5-2 to Figure 5-6.

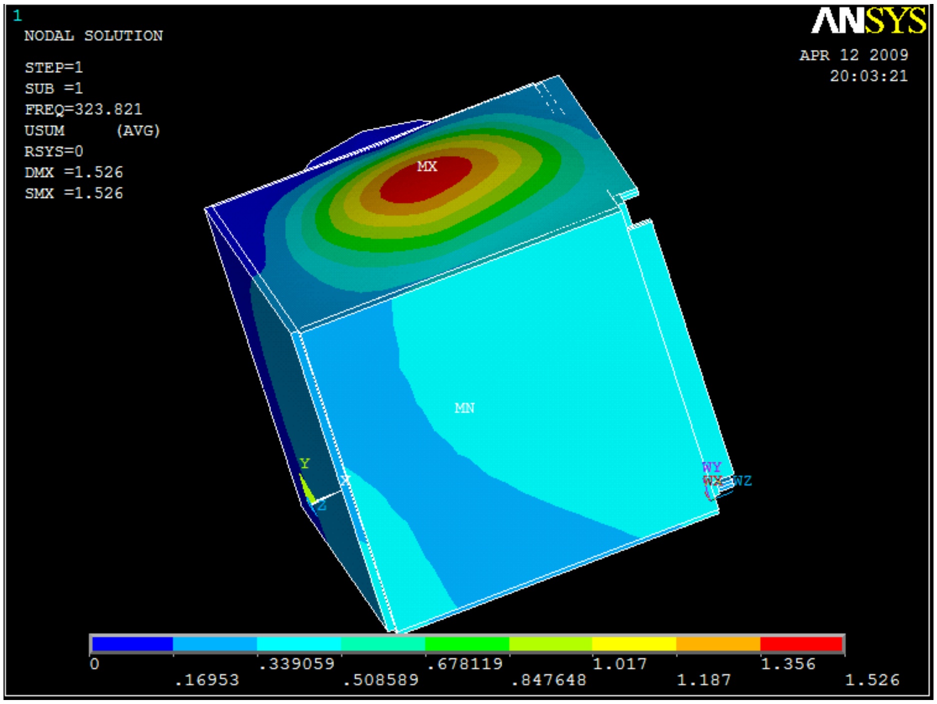


Figure 13: Mode1- Anti-sun side breathing

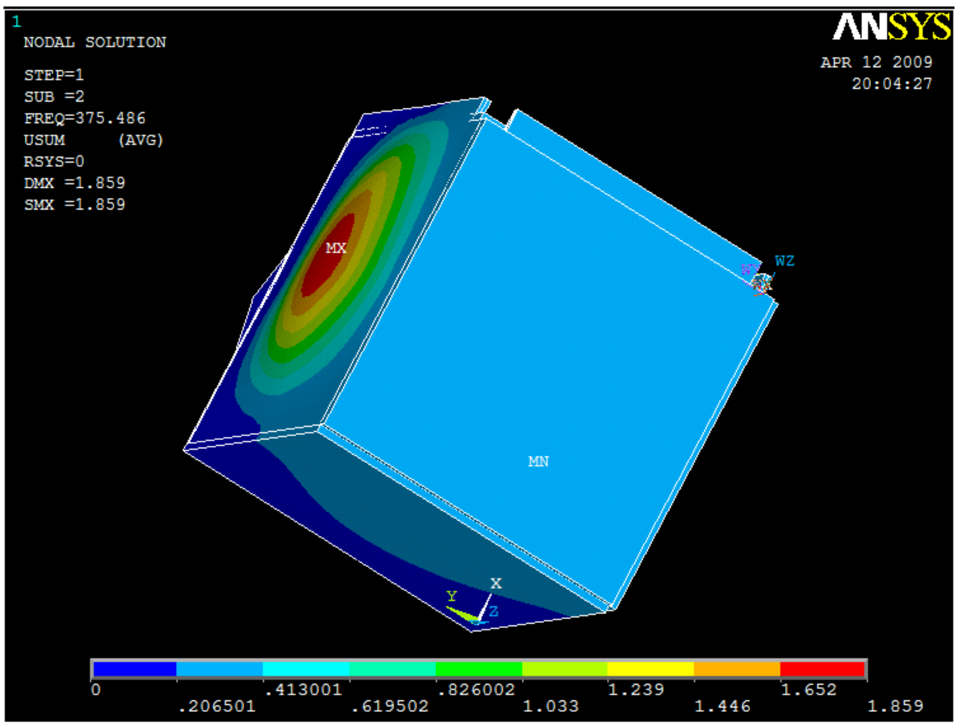


Figure 14: Mode2- anti-sun side breathing

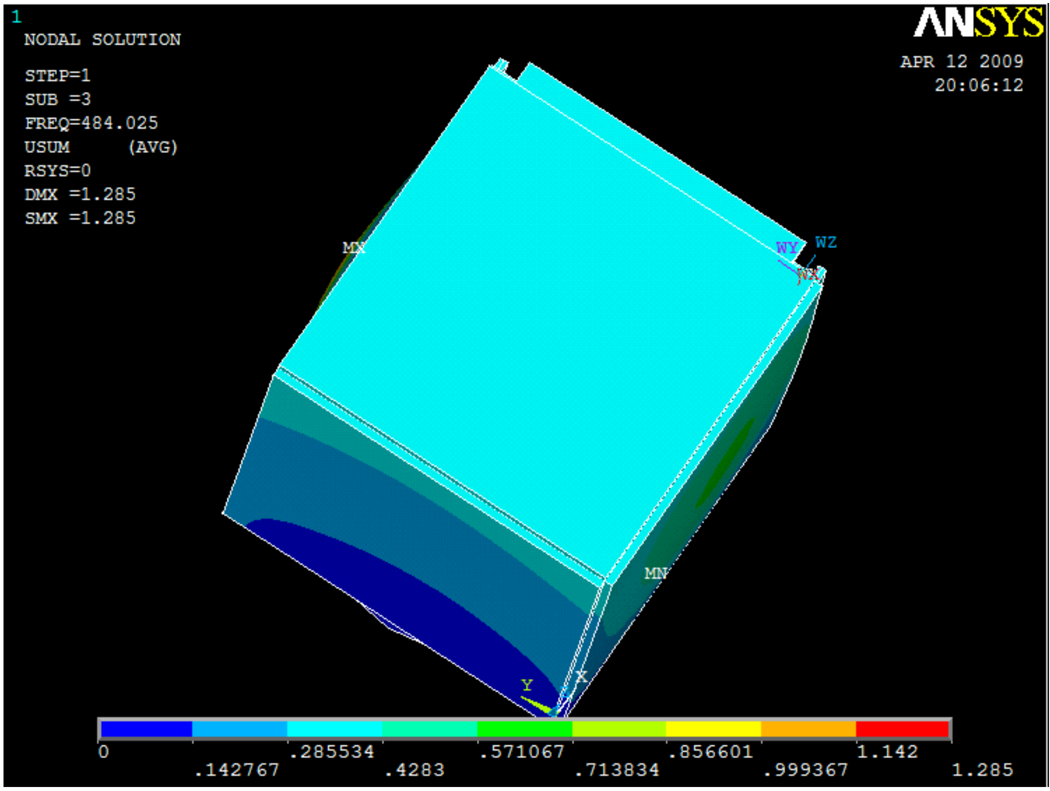


Figure 15: Mode3- Anti-sun side breathing and bending

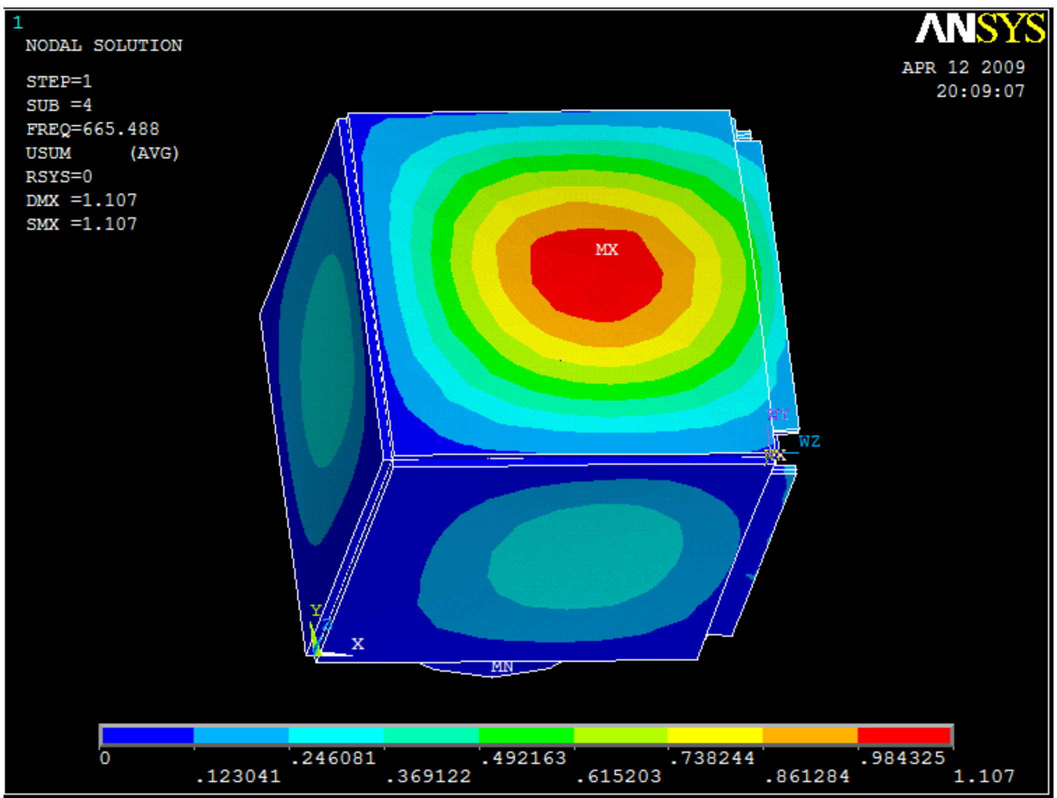


Figure 16: mode4: Zenith side breathing and small amount of deflection on all other sides

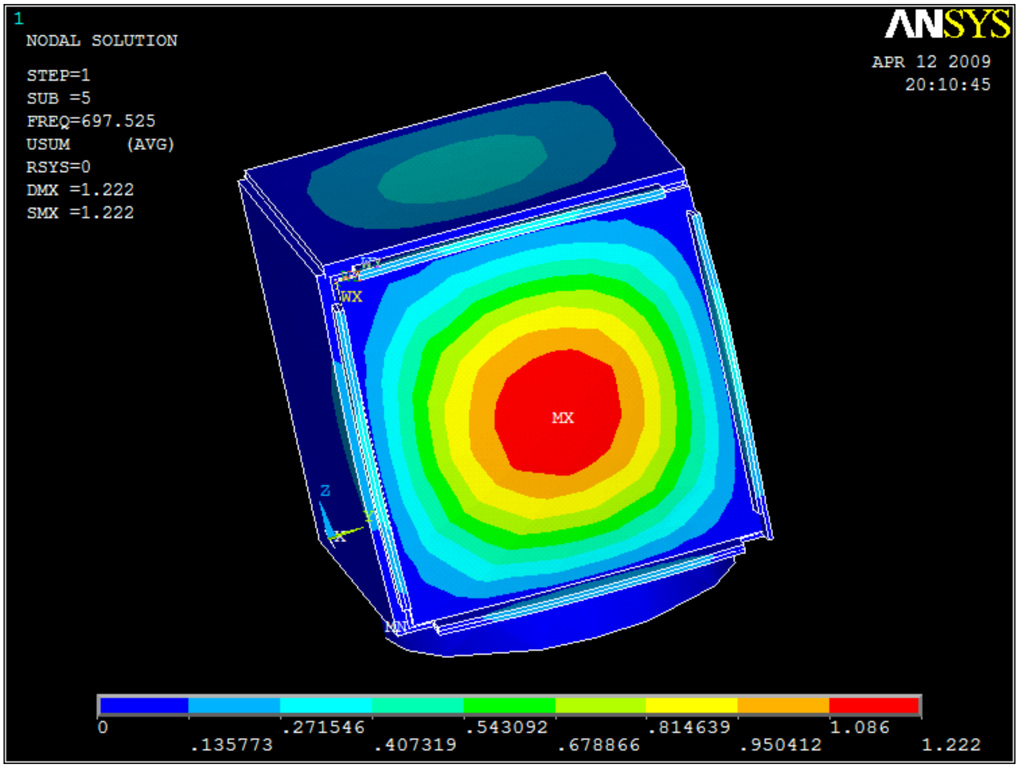


Figure 17: Mode5- Lagging and zenith breathing

5.2.2 Static analysis results:

The satellite is constrained at base of LVI. Loading applied is 9g acceleration in longitudinal direction and 6g acceleration in lateral directions. The plot obtained for the displacement is shown in Figure 5-7.

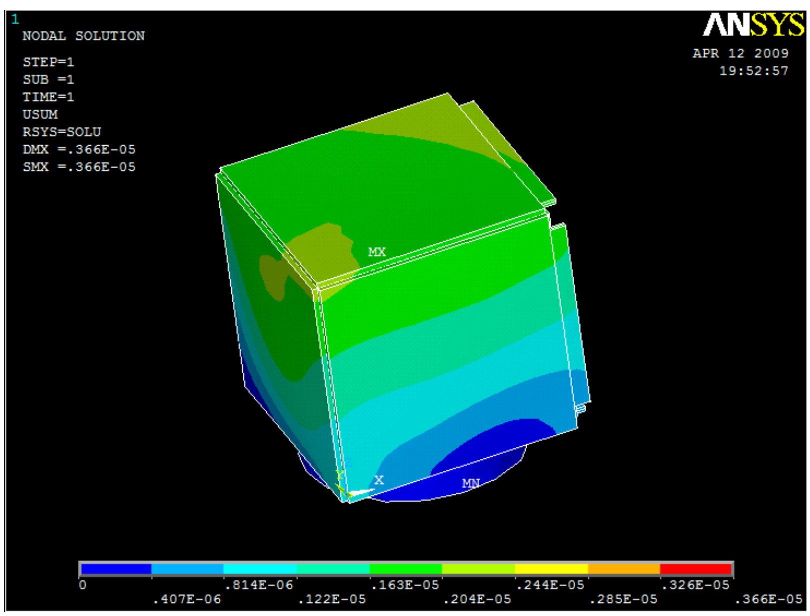


Figure 18: Displacement plot under static loading

The displacement observed for static loading is very small. Von mises stress and strain plots are given in Figure 5-8 and Figure 5-9.

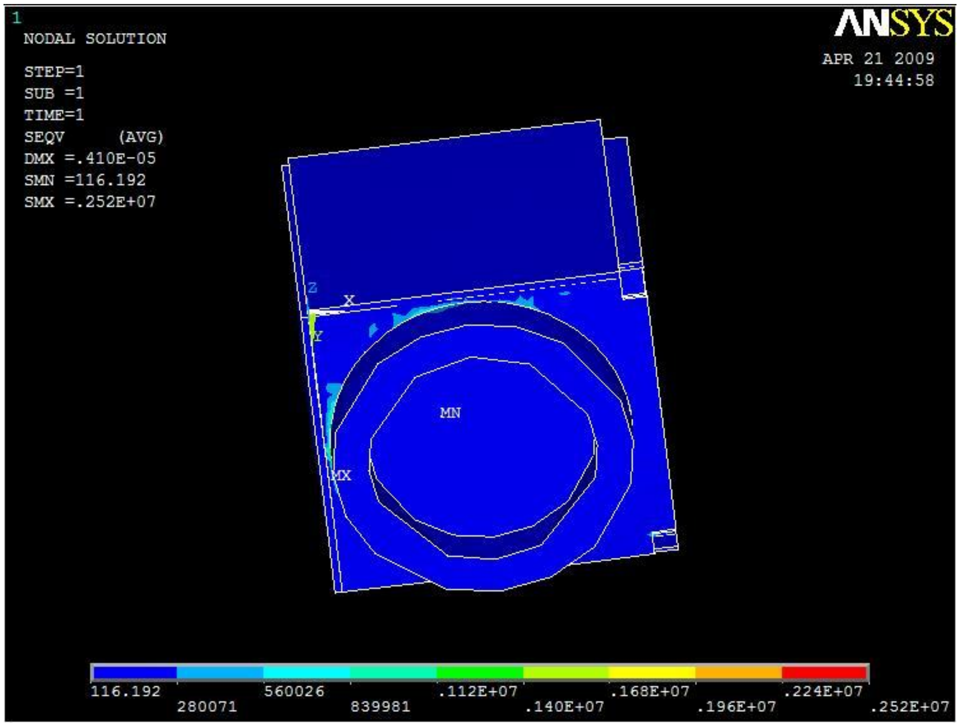


Figure 19: Von mises stress

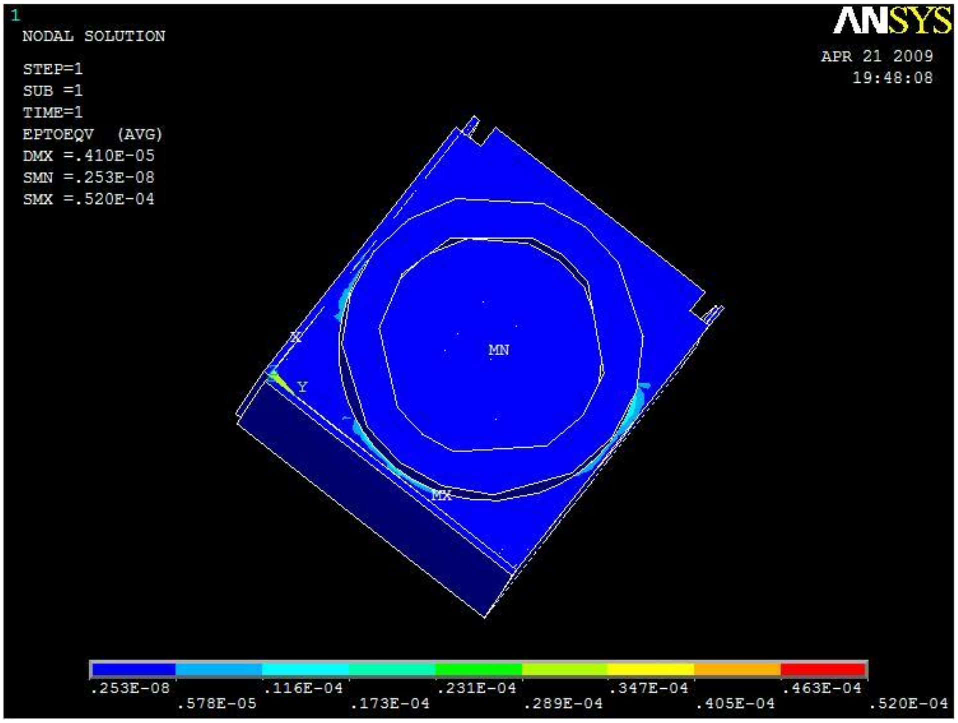


Figure 20: Von mises strain

5.2.3 Center of gravity and inertia matrix:

The location of center of gravity and inertia matrix is obtained in ANSYS. The location of center of gravity from the centroid of the cube is

X = -0.00499m, Y = 0.00353m, Z = 0.07587m

The inertia matrix about center of mass is

$$I = \begin{matrix} 0.3102 & -0.001835 & -0.009890 \\ -0.001835 & 0.3123 & -0.008012 \\ -0.009890 & -0.008012 & 0.2362 \end{matrix}$$

5.2.4 Frequency response:

The model was loaded with 3.75g acceleration in all directions and was analysed for harmonic vibrations. The loading was stepped loading and the material damping was taken to be 1%. The model was analysed with 500 sub-steps. The results obtained are presented in following figures. In each figure first the point at which the plot is obtained, is pointed with a circle. In next plot, the displacement response at that point is plotted as function of frequency.

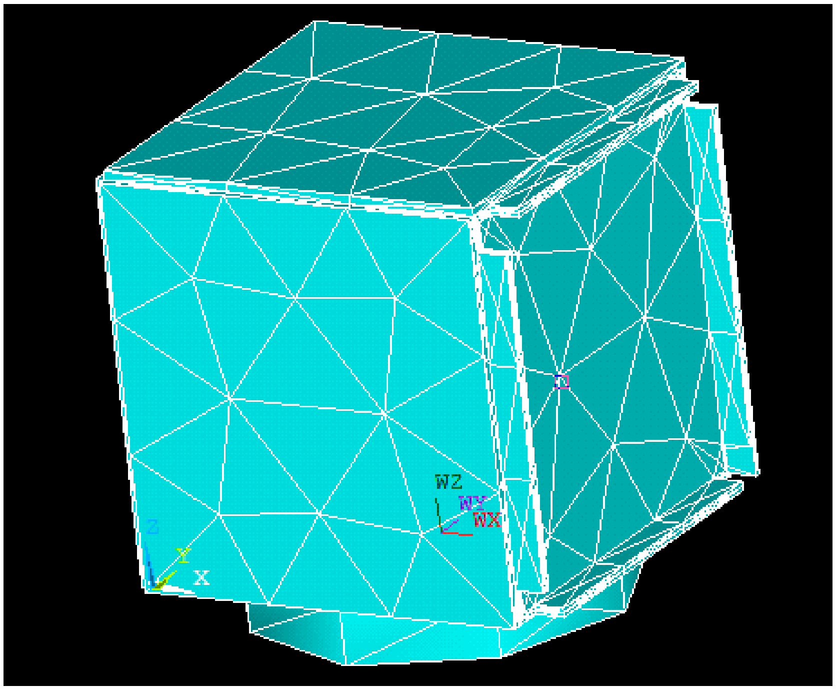


Figure 21: Lagging side point

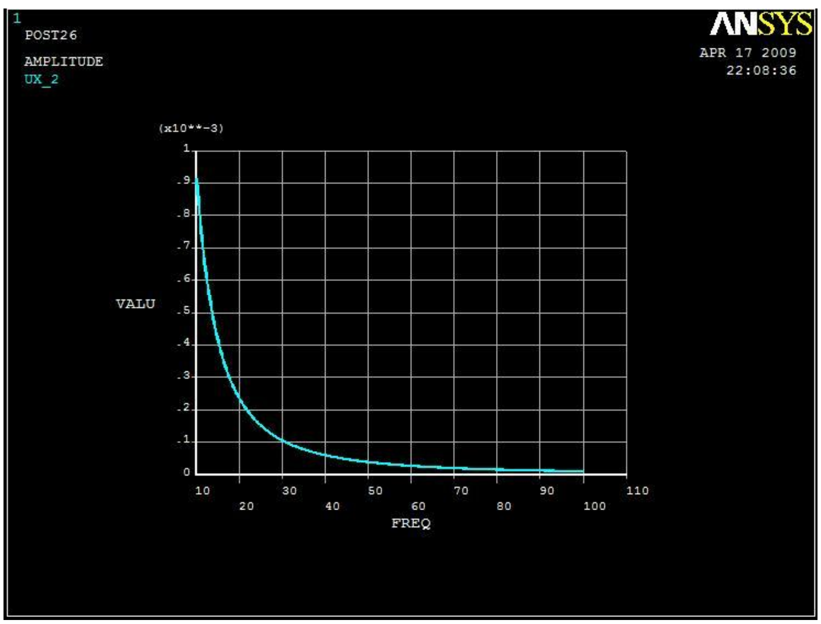


Figure 22: Displacement plot

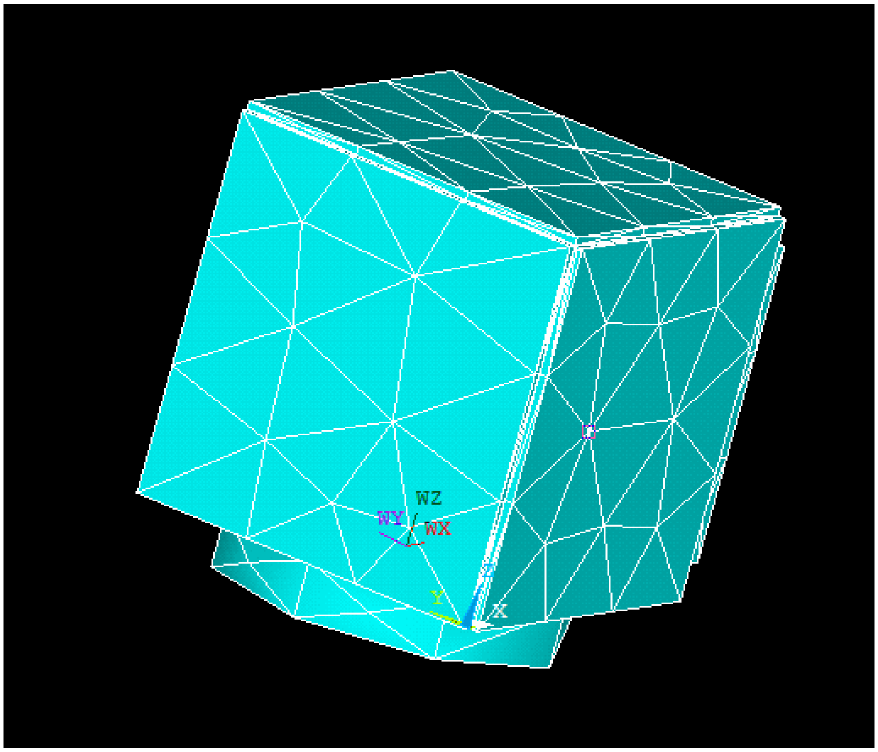


Figure 23: Sun-side

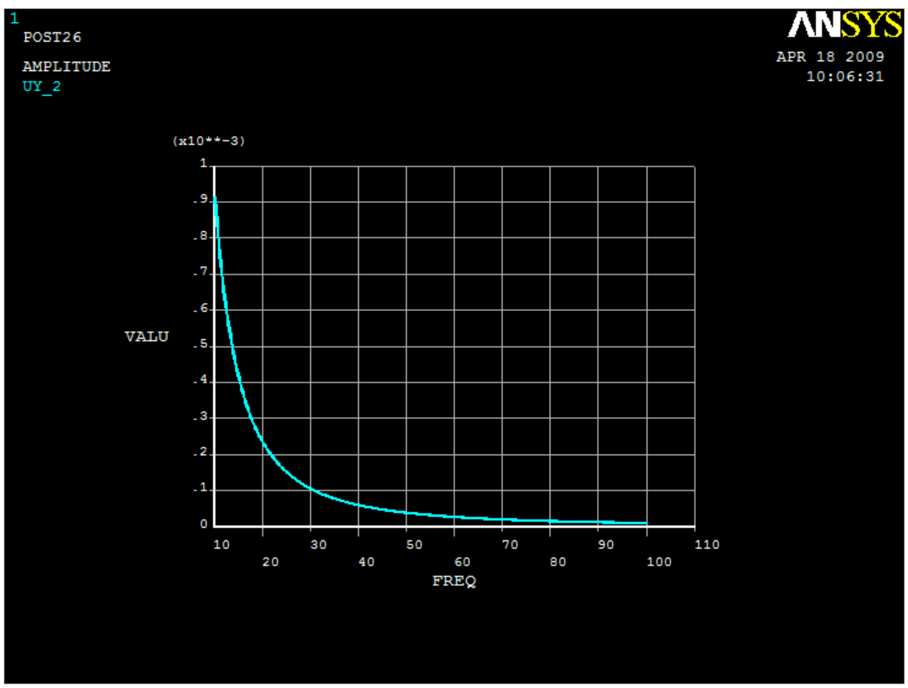


Figure 24: Displacement on sun-side

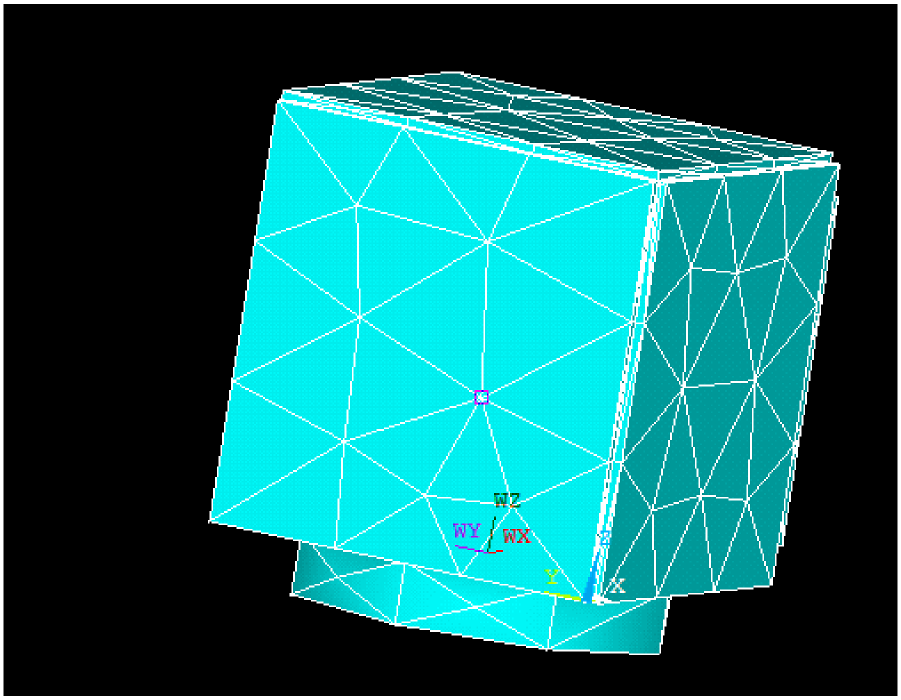


Figure 25: Leading side point

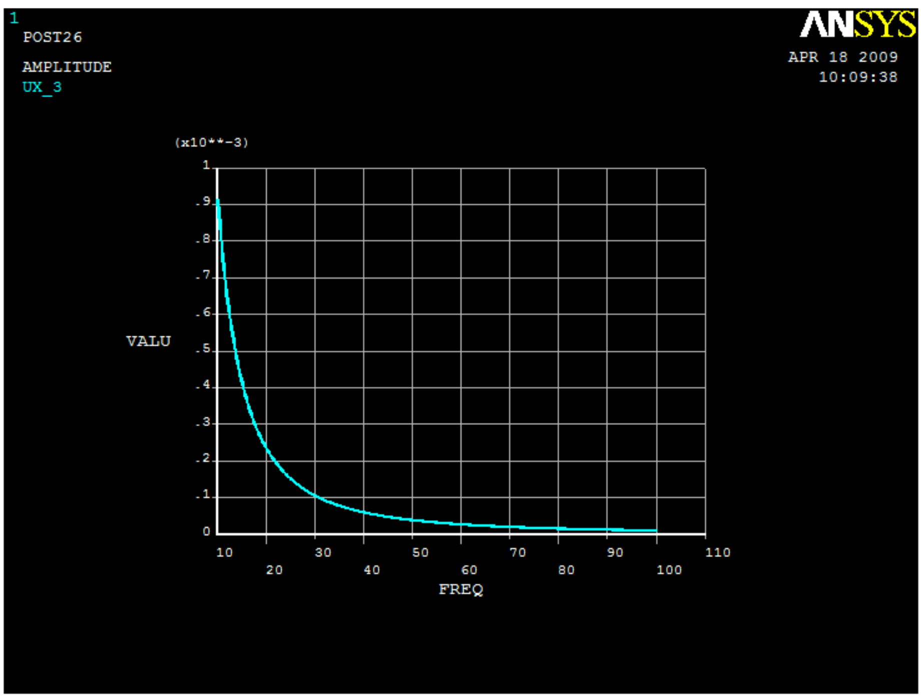


Figure 26: Leading side point displacement plot

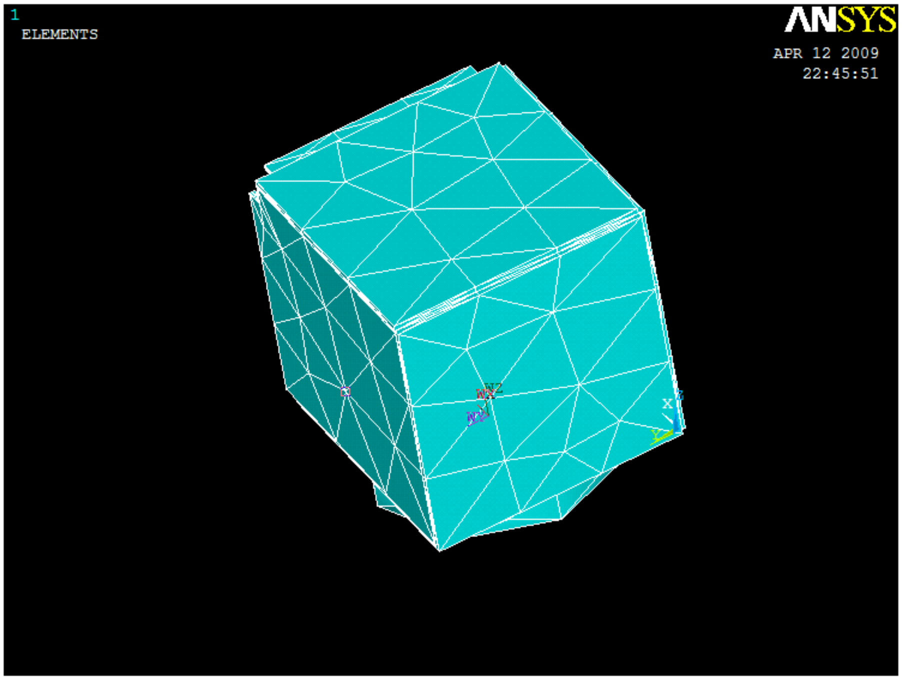
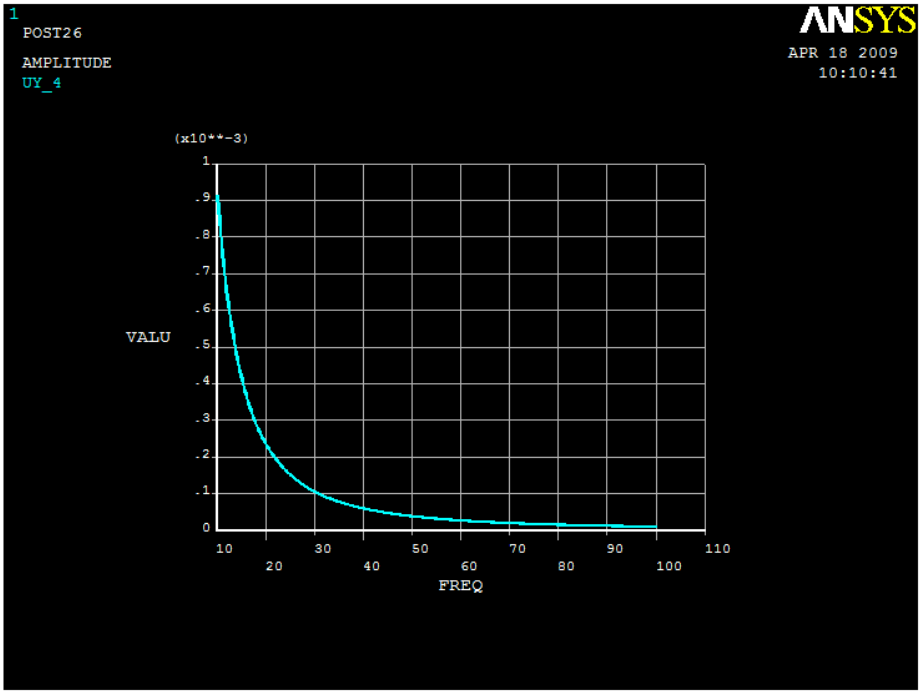


Figure 27: Anti-sun side point



28: Anti-sun side point displacement plot

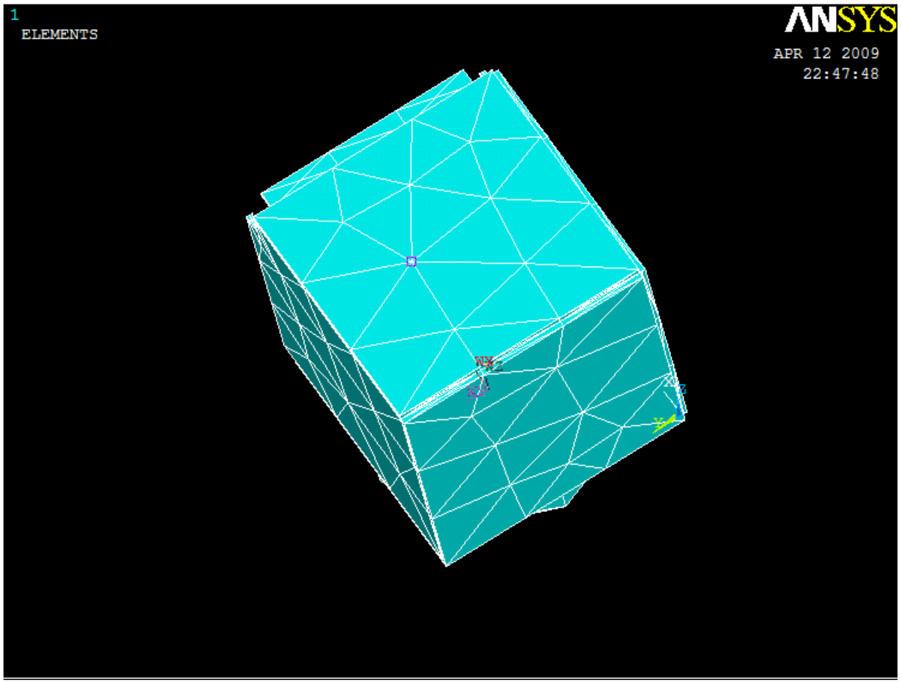


Figure 29: Zenith side point

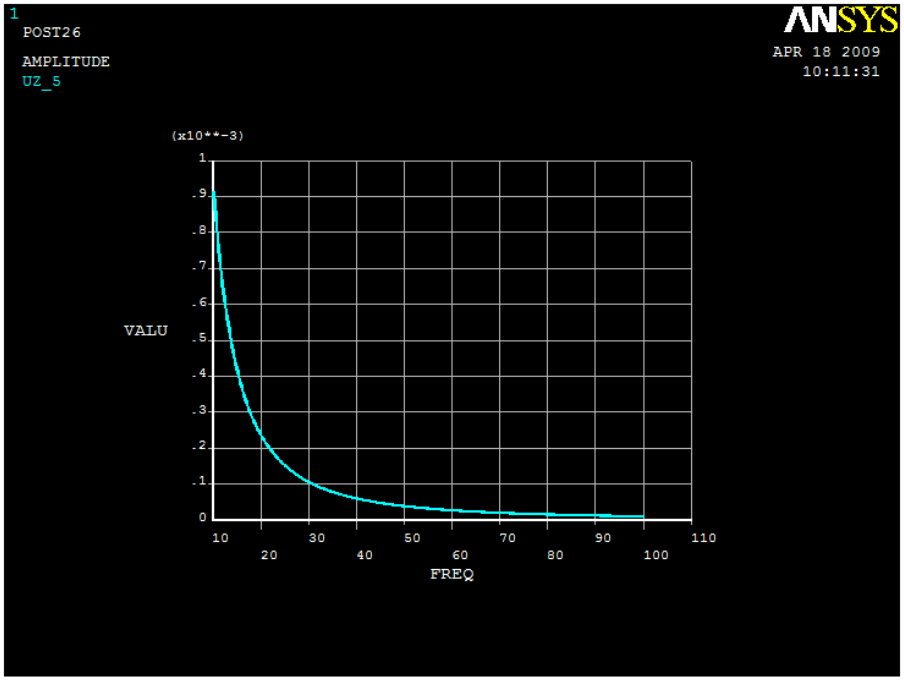


Figure 30: Zenith side point displacement plot

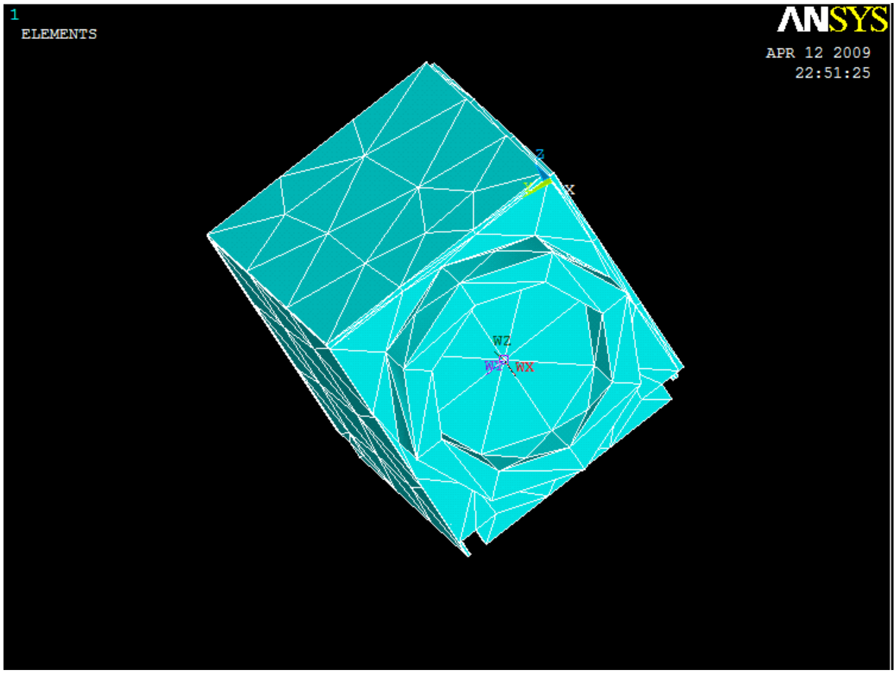


Figure 31: Nadir side point

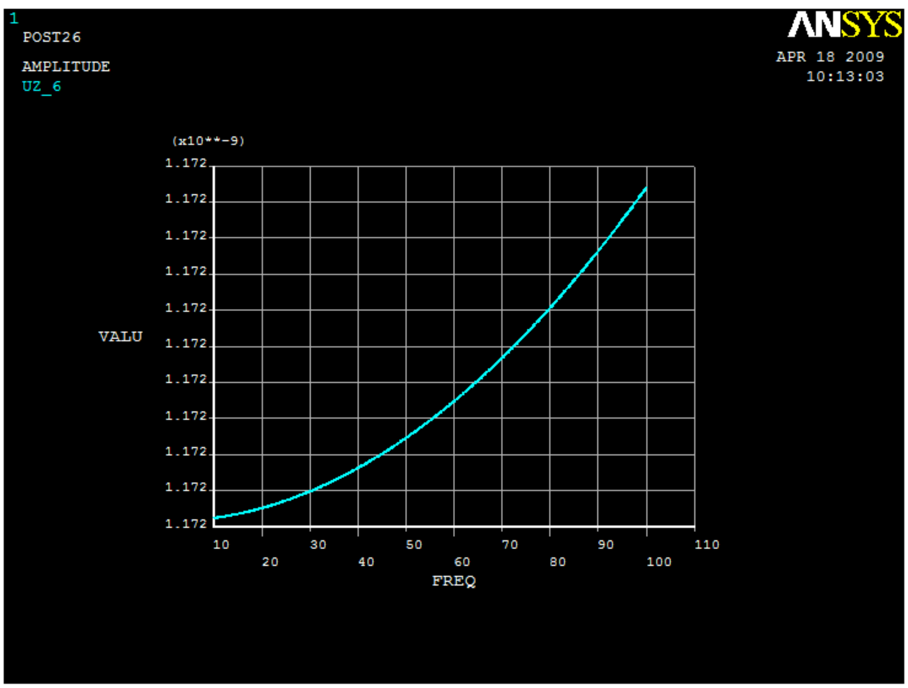


Figure 32: Nadir side point displacement plot

We can observe that in all the plots, the displacement never goes above 1mm. Since all the points chosen are in the middle of the panel, maximum displacement is expected at these points. Since the displacement is very less (less than 8mm specified in the requirements) we can conclude that the structure will be safe.

More detailed harmonic analysis needs to be done with response in terms of stresses and amplification factors. This will give clearer picture of the response of the structure to harmonic loading.

5.2.5 Random vibrations:

The procedure for random vibrations analysis of the satellite structure is not clear. Since it is observed that the plate model is giving accurate results, test case of random vibration analysis is performed on the plate model. Accurate simulations of random vibrations will be done in next stage.

Chapter 6

Validation of simulation results

In this chapter, work done for validation of the simulation results is presented. First the validation of element used is presented and then validation of entire satellite model is done by breaking the model into each component and performing modal analysis on them.

6.1 Validation of elements used in ANSYS:

In ANSYS, we have wide choice for the kind of element we can use. It is important to first validate the element. For validation, results obtained from modal analysis of aluminium plate are compared to the theoretical results for first natural frequency.

The formula used for the theoretical calculation of natural frequency is the formula used in Roark's formulae for stress and strain. The natural frequency of the plate with all sides fixed is given by:

$$f = \frac{k_1}{2\pi}\sqrt{\frac{Dg}{wa^4}}$$

Where f is the natural frequency of the plate, k_1 =36 if length = breadth, w/g is mass per unit area, and D is given by

$$D = \frac{Et^3}{12(1-\gamma^2)}$$

Here E is the modulus of elasticity, t is the thickness and γ is the poisson's ratio. The result obtained using this formula was used to validate the element.

Geometry considered is a plate of sides 230x230x230 mm and of thickness 6mm with material properties of Al6061T6. Three elements were considered. SOLID 45 is 8 node solid element. SOLID95 is 20 node solid element and SHELL63 is a planer shell element. The volume/area is meshed using free meshing option since free meshing will be used for complicated geometry in actual satellite model. The results obtained for first natural frequency are tabulated below.

Table 7: elemental validation for first mode

Theoretical	SOLID45	SOLID95	SHELL63
1001.32	2269.7 Hz	1002.6 Hz	999.58 Hz

Hence we can see that both SOLID95 and SHELL63 give results that are matching with theoretical results. The element used in all analyses is SOLID95 since it is easier to model the satellite as a solid than as plane. SOLID45 gives erroneous results for free mesh hence it is not used.

6.2 Validation of satellite model:

For validation of entire satellite model, the model is broken into individual components and the modal analysis is done on each component by looking at the mode shapes obtained in analysis of entire satellite model.

The geometry considered for validation is two perpendicular plates of thickness 3mm with solar panel backing as aluminium honeycombs of thickness 6mm. The joint between plates is modelled with considering the harness joining them. Harness is modelled as two plates joined together. The side coinciding with the panel is glued to the panel giving it a continuous joint. All the sides except the sides joined together are fixed. The results obtained from both the analyses are compared in table 6-1.

Table 8: Natural frequency comparison for satellite model and validation model

SET	Satellite model frequency	Validation model frequency
1	323.82	347.84
2	375.49	405.40
3	484.03	527.97
4	665.49	632.78
5	697.52	905.37
6	836.38	908.14
7	871.80	916.38
8	894.24	929.89
9	973.75	946.99
10	1245.4	947.43

It can be observed that the initial mode frequencies are matching well. For higher modes, mode shapes become markedly different for the validation model when compared to the actual satellite model. Hence we can see that the difference between two frequencies is comparatively higher for higher modes. However the initial modes are matching well. Hence we can conclude that the entire satellite model is correct and there is no error in the modeling.

6.3 Experimental validation:

The flight model will be tested for following tests before launch. The loading applied during testing is the loading specified in the requirements section.

1. Static test (Strength test)
2. Sinusoidal test
3. Random vibrations test

Satellite will be launched only if it does not fail in all the three tests.

Acknowledgement

We thank Department of Aerospace engineering, IIT Bombay for all the support that they have provided. We would like to thank Prof Mujumdar for his immense help in all kinds of problems. We would also like to thank Mr. Mandar Kulkarni for his help in structures lab. Last we would like to thank entire satellite team for all the support that they have provided in all scenarios.