Design, Modeling and Analysis of Low Earth Orbit Satellite

Hamid Khan, Asif Israr*
Department of Aeronautics and Astronautics, Institute of Space Technology, P O Box 2750,
Islamabad – 44000, Pakistan
* Corresponding Author: Tel: 00 92 51 9075477, Fax: 00 92 51 9273310
E-mail: asif.israr@ist.edu.pk

Abstract

This paper presents design, modeling and analysis of engineering qualified satellite model used for remote sensing. Detailed study is carried out for the design and modeling of the satellite structure focusing on the factors such as the selection of material, optimization of shape and geometry, accommodation of different sub-systems and payload. The center of mass is required to keep within the range of (1–2)cm from its geometric center, and that needs to be calculated theoretically as well as through Pro-E. Once the model is finalized it is required to be analyzed by the use of *Ansys*, a tool for finite element analysis under given loading and boundary conditions. Static, modal and harmonic analyses in *Ansys* are performed at the time of ground testing and launching phase. The finite element analysis results are also validated and compared with the theoretical predictions. These analyses are quite helpful and suggest that the satellite structure doesn't fail and retain its structural integrity during launch environment.

Keywords: Satellite, Design, Modeling, Analysis, Pro-E, *Ansys*, Static, Dynamic, Harmonic.

1. Introduction

Low earth orbit (LEO) satellites are usually used for communication and earth imaging enclosed by a payload of signal processing module only. In designing of LEO satellite, the model is premeditated by an appropriate selection of material based on low weight, high strength to weight ratio and space qualified. In addition, basic geometry of the satellite body is also defined by considering the factors such as heat distribution & heat dissipation, weight of fasteners, accessibility & maintainability, accommodation of sub-systems, center of gravity and manufacturing cost. In most of the literature, aluminum alloy as a material and cubical shape of the structure are employed because of the aforesaid reasons. Swiss cube [1] structure described a better representation for designing and selection of the material for the satellite construction.

The mono-block can be introduced that acts as a back bone for the construction of the basic geometry of the satellite structure as reported in the Swiss cube and SATAX project [2]. It helps to increase the strength of the whole structure, and mainly performs three different tasks; proper aligning of the geometry, holding of all the attitude thrusters, and assembling of sub parts on it. In this way a very strong, robust and extremely light frame structure can be obtained. Misbah [3] performed static and dynamic analysis of small satellite. He maintained a safety factor higher than 5 for the analysis by increasing or decreasing of stringer height and increasing of trays and stringer thickness. The measured first natural frequency was quite below the launcher requirements. He tried to shift the natural frequency to higher value by the increase of trays and stringer thickness but no avail. However, increasing of stringer height to 20mm it considerably effected on frequencies. In addition, it was also observed that without stringer the first natural frequency dropped considerably. In model analysis the first and fifth natural frequency at constant stringer height of 6mm was found to be 41.5Hz and 100.4Hz respectively at a tray thickness of 2mm in longitudinal direction, similarly 41.8Hz and 101.8Hz respectively are observed in the lateral direction. With the increase of tray thickness to 10mm, the first natural frequency increased both in longitudinal and lateral directions. RASAT satellite [4] described the analysis carried out on the main stiffened and honeycomb panel. Normal mode and stress analysis were performed by the use of *Ansys*, a software tool for the finite element analysis. In normal mode analysis the structure was constrained at the place of bolts and the first frequency was found to be 86Hz. In stress analysis static inertial force of 60g was applied in all three directions of the structure and the maximum local stress was found to be 161MPa which was far less than the strength of the material used, hence the structure was safe under these conditions. Fufa et al. [5] investigated the locking processes of satellite and deployment interaction. The modal analysis of satellite solar panels was performed and results were illustrated with extremely good agreement. Libin and Hui [6] carried out the analysis on the telescopic frame using *Ansys*. Solid95 was used an element type for the structural analysis as this element type ignores most of the irregular shape of the structure. Furthermore, the bolts of the holes were ignored as it were relatively very small. The analysis showed that the maximum stress and deformation was greater than the allowable design limit. A centric brace was introduced in the structure which drastically reduced the deformation in the structure. The model analysis was also carried out at different load steps and observed variation in their frequencies. Cihan et al. [7] used *Ansys* to analyze the behaviour of the structure by employing different element types, meshing type of quadrilaterals for shell and hexahedral for volumes. In static analysis the maximum von-mises stress was found to be 5.0956MPa at the intersection points which was less than the yield strength of aluminum and the displacement of 0.004m was observed on the top panel. In model analysis the first and last natural frequency was 633.25Hz and 1948.3Hz respectively, the last frequency was extremely high but still it is within the range of launch frequency. Static analysis of the Adesat was performed in ref. [8]. AdeSat consists of four square rails and six flat plates. The top solar panel was simulated by placing a pressure caused by the load of the solar cells, and the deployable panels were treated similarly with their load acting onto the side plates. In *Ansys*, the PCBs were considered as point masses. For the validation of satellite sub-systems an additional stress analysis were also carried out for the PCBs, hinge, hinge pin and bolts. These analyses were performed using *Ansys* under given frequency range.

During satellite launch, it is subjected to various external loads resulting from vibro-acoustic noise, booster ignition & burn out, propulsion system engine vibration, steady-state booster acceleration and much more. Hence, design and model such a structure that should meet the launcher requirements.

2. Design, Modeling and Analysis of LEO Satellite

In this study, satellite structure is designed and modeled sub-systems to acquire the engineering qualified model. The constructed model is then required to meet all the testing parameters under given loading conditions. Various aspects are considered in designing of the structure such as strength, life, material, shape etc. Aluminium alloy 7075-T6 is used here for building of the basic structure of the system. This material has light in weight and has high stiffness to withstand loading conditions. Although there are many shapes of satellite structure such as cubical, hexagonal, circular etc. that is being currently used in space industry. These shapes can be employed according to application, mission orbit, and internal space requirement. A cubical configuration is selected with external dimension of (1140x1190x1569)mm and weighs 650kg, excluding the solar panels. In order to keep the structure aligns and stable, a mono-block is introduced that provide more strength to the structure. With the addition of mono-block the weight of the structure is increased, however, there must be a compromise in designing; thus strength over weight is more important in this case. Mass distribution is utmost important for modeling of the sub-systems into the inner structure of the satellite. These sub-systems are placed in such a manner to keep the center of gravity (C.G) of the entire structure within (1-2)cm from the geometric center. Otherwise, a catastrophic failure occurs if this safe range is not achieved. For these reasons two heavy sub-systems i.e. batteries and two telescopes are placed above and below the cubic structure walls respectively. The remaining sub-systems are arranged accordingly. 5mm distance is also maintained between all sub-systems to avoid heat dissipation among these sub-systems.

The complete satellite structure is modeled in Pro-E. The C.G of the cubical body is then calculated using Pro-E & some basic formulations, and found that it lies in the specified range as mentioned above. The build satellite structure is then exported in *Ansys* 11. In *Ansys* the structure is assembled and meshed successfully for analyzing the structural body of the system. Static, modal and harmonic analyses are performed to analyze the integrity of the structure under given loading conditions. The main objectives of these analyses are to ensure that the satellite structure survives during launch loads. For these analyses, it is assumed that the satellite is constrained in space with the launch vehicle interface as in actual practice it is constrained by the use of some basic mechanism. Furthermore, these analyses are simplified by assuming that screwing of the sub-systems are omitted, and all extra holes on the structure for various reasons are neglected as they have very small dimensions.

The results obtained from the FE analysis can also be verified by simple formulation for first natural frequency only. There is no such relation available in the literature for the vibration analysis of the satellite structure. For instance, a beam can best describe the present configuration. The theoretical natural frequency of the beam with one end fixed and free on the other end for the first mode is given by:

$$f_1 = \frac{1.875^2}{\;} \sqrt{\frac{EI}{\;}} \qquad (1)$$

where *L* is the length of the beam, *I* is the area moment of inertia, *E* is the modulus elasticity and *m* is the mass of the beam.

Now, Static, modal and harmonic analyses are performed in the subsequent sections followed by conclusions.

2.1 Static Analysis

Static analysis is usually used to estimate the stresses, strains, displacements and forces in the structural components of the system. Hence these analyses are very essential to measure the strength of the satellite structure. Generally, steady loading and response conditions are assumed during the analysis. In this analysis, the weight of the entire satellite and a force of magnitude 9g are acted on the geometric center of the structure while the lower legs of the base are fixed. The maximum deformation and stresses are found at the top sheet and can be seen in Figs. (1-2) respectively.

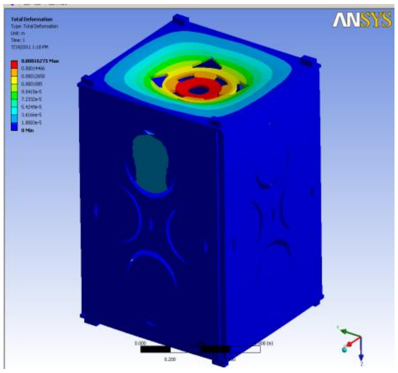


Fig.1: Deformation pattern

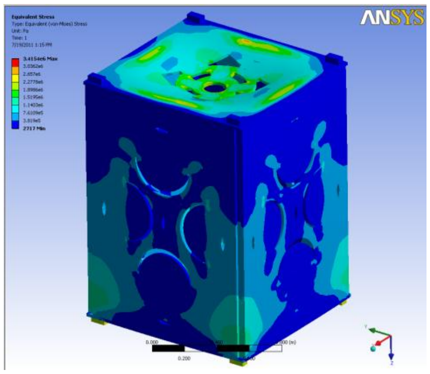


Fig. 2: Stress pattern

The maximum deformation (1.6275x10^{-4}m) is far less as compared to the dimensions of the structure; similarly the maximum equivalent stress is 3.4154x10^{6}Pa which is also lower than the yield strength of aluminum. It means structure can sustain the loading conditions, doesn't fail & maintain its integrity during actual launch after the application of maximum static load.

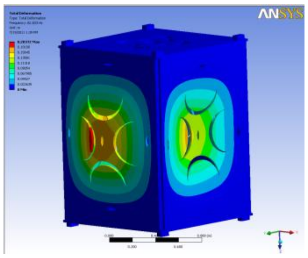
2.2 Modal Analysis

As per global stiffness requirements, the fundamental frequency of the entire system must be ≥ 30Hz. By doing so, the structure will not resonate and will survive during launch without any deformation. In order to get the frequencies and mode shapes of the satellite structure modal analysis need to be carried out for dynamic loading conditions. Solid186 is used as an element type for examining the satellite structure. Table 1 shows ten modal frequencies and deflection values against each frequency value.

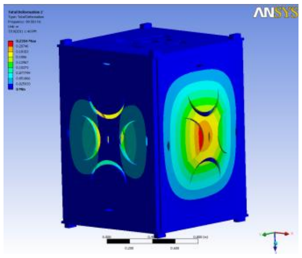
The first modal frequency can also be validated as obtained from modal analysis from the theoretical relation as described in Eq. (1). It can be seen that there is a little difference between the two results, and element type solid186 is the right choice for such analysis. The modal shapes are also illustrated in Fig. 3.

Table. 1 Ten modes of vibration

Modes	Frequency [Hz]	Def. [m]
1	82.033	0.2037
2	89.581	0.2334
3	97.815	0.2296
4	114.07	0.1869
5	137.51	0.1962
6	141.97	0.2507
7	143.13	0.2389
8	146.88	0.2176
9	151.31	0.2524
10	157.43	0.2782



a. Mode 1: Freq.= 82.03Hz and Def.= 0.2037m



b. Mode 2: Freq.= 89.58Hz and Def.= 0.2334m

c. Mode 3: Freq.= 97.82 Hz and Def.= 0.2296m d. Mode 4: Freq.= 114.07Hzand Def.= 0.1869m

e. Mode 5: Freq.= 137.51Hz and Def.= 0.1962m f. Mode 6: Freq.= 141.97Hz and Def.= 0.2507m

g. Mode 7: Freq.= 143.13Hz and Def.= 0.2389m h. Mode 8: Freq.= 146.88Hz and Def.= 0.2176m

i. Mode 9: Freq.= 151.31Hz and Def.= 0.2524m j. Mode 10: Freq.= 157.43Hz and Def.= 0.2782m

Fig. 3 Ten modal shapes for different frequency values

2.3 Harmonic Analysis

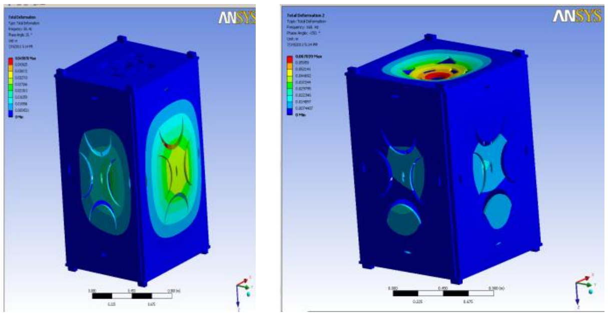
The harmonic analysis is also performed to determine the maximum possible stress on the structure. It is usually done for the vertical alignment of the satellite. The qualification loading levels as shown in Table 2 are used here for analyzing the satellite structure.

Table 2. Qualification loading levels

	Normal to the mounting plane	Parallel to the mounting plane
Freq. [Hz]	Levels	

0-200	12g	8g

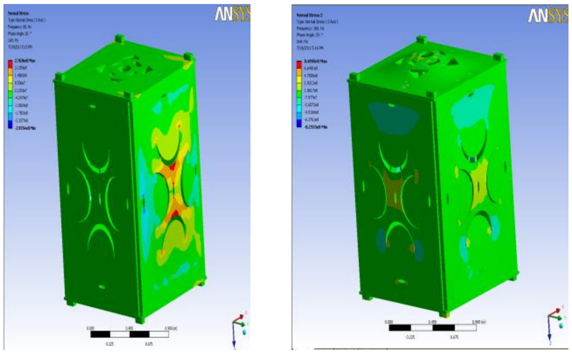
Mode superposition method is applied for the solution in *Ansys* with a damping of 1%. The maximum stresses and deflections are tabulated in Table 3. These can also be demonstrated in the Figs. (4-5). Within the frequency range as mentioned (0-200)Hz, catastrophic failure doesn't observe in the satellite structure.



a. Lateral direction b. Longitudinal direction

Fig. 4: Deformation pattern

Table 3 shows the maximum harmonic stress for the structure is 8.60×10^{8}Pa at the side sheet of the cubic structure in case of longitudinal direction. Similarly, the maximum stresses evaluated by the static analysis are 3.4154×10^{6}Pa at the top sheet. Therefore it doesn't exceed the yield strength of the material used.



a. Lateral direction b. Longitudinal direction

Fig. 5: Stress pattern

Table 3. Harmonic Analysis Results

	Max. normal stress [Pa]	Freq. [Hz]	Max.def. [m]	Phase angle
Longitudinal	8.60×10^{8}	160	6.74×10^{-2}	29^{O}
Lateral	2.76×10^{8}	80	4.91×10^{-2}	20^{O}

3. Conclusions

The satellite structure and its associated sub-systems are designed and modeled in such a manner to keep the center of mass within the range of (1-2)cm from the geometric center of the structure. A simpler model of mono-block is introduced for strengthening of the entire structure. Accessibility of the sub-systems is easier to handle in this case and the structure doesn't exceed the yield strength of the material used.

All these analysis are performed under given loading and boundary conditions. In static analysis the maximum stress and deformation are found to be 1.6275×10^{-4}m & 3.4154MPa respectively which are in the control limits when compared with the yield strength of the material (aluminium alloy 7075-T6 here). In modal analysis mode shapes can easily be seen at the outer surfaces of the structure. A maximum displacement of 0.2782m is observed at the side sheet of the cubic body for a frequency of 157.43Hz. Similarly, harmonic analysis are performed for a frequency range of (0-200)Hz with the application of two forces 8g parallel to the mounting plane and 12g normal to the mounting plane. The amplitude at lateral and longitudinal directions are found to be 6.74×10^{-2}m at a frequency of 160Hz and 4.91×10^{-2}m at frequency of 80Hz respectively which is very small under given loading and boundary conditions.

In summary, the satellite structure is modeled and analyzed for static, modal and harmonic response to ensure that the structure sustains in the harsh launch loads. The obtained estimate values of maximum stresses and deflections are less than the failure values.

4. References

[1] Diana, A. Benjamin, J and Guillaume, R. (2006). S3-A-STRU-1-4 Structure and configuration EPFL Lausanne Switzerland.

[2] Pacheco, E. Conte, R. J. Mendieta, F. Muraoka, R. Luis Medina, J. Arvizu, A. (2002). The SATEX Project: A Mexican effort the development of micro satellite platform for space technologies knowledge and human resources preparation. CICESE Research Center IEEAC paper # 1288.

[3] Sedighi, M. and Mohammadi, B. (2003). On the static and dynamic analysis of small satellite (Mesbah). Mechanical Engineering Department, IROST, Theran, IRAN.

[4] Ontac, S. Dag, A. and Golkler, M.I. (2007). Structural finite element analysis of stiffened and honeycomb panels of RASAT satellite. TUBITAK Space Technology and Research Institute METU Campus, 06531. Ankara, Turkey, and Mechanical Engineering Department Middle East Technical University, 06531, Ankara, Turkey.

[5] Fufa, B. Zhao-Bo, C. and Wensheng, M. (2010). Department of Mechanical Design, School of Mechanical and Electrical Engineering, Harbin Institute of Technology No. 92, West Da-Zhi Street, Harbin, Heilongjiang 150001, China. *Information Technology Journal*, ISSN 1812-5638.

[6] Libin, S. and Hui, F. (2002). Static and modal analysis of telescope frame in satellite. Department of Mechanical Engineering Tsinghua University, and School of Material Science and Engineering University of science and Technology, Beijing.

[7] Cihan, M. Haktany O.O. Akbulut, İ. and Aslan A.R. (2007). Flight dynamic analysis of ITUpSAT1. Istanbul Technical University, Faculty of Aeronautics and Astronautics. Department of Space Engineering 34469, Maslak, Turkey.

[8] Braegen, E. Hayward, D. Hynd, G. and Thomas, A. (2007). The design and build of a small satellite based on Cubesat standards. The University of Adelaide.