STRUCTURAL DESIGN OF THE BRAZILIAN SCIENTIFIC APPLICATIONS SATELLITE 1 (SACI-1)

DeSouza, P.N.; Takahashi, W.K.; Kataoka Filho, M.; Ribeiro, M.S.; Varotto, S.E.C.; Neri, J.A.C.F.
Instituto Nacional de Pesquisas Espaciais
BRAZIL

Abstract

This paper describes the structural design and analysis of the Brazilian Scientific Applications Satellite (SACI-1) that is devised as a multipurpose bus for scientific applications. The structure includes a Mainbody (Platform and Payload), an Adapter, four deployable panels and the holddown and release mechanisms (HDR). Its overall shape resembles a parallelepiped measuring 600 x 400 x 400 mm (Mainbody only) and weighting about 65 kg (Mainbody plus Adapter). The Platform consists on a pack of 9 aluminum ribbed frames of different thickness stacked horizontally and connected by means of 12 stud bolts. The Payload is assembled on the top of the Platform and has six ribbed plates that form a box in which most of the payload is assembled. The separation is accomplished by cutting (or releasing) one central rod that joins the Platform to the Adapter. The analysis of this structure was performed using a FEM model for static, normal modes, frequency response, transient response and random vibration analysis. With these results it was possible: (a) to verify the preliminary design of the structure and define the necessary changes; (b) to determine the first global frequencies of the assembly and; (c) to verify the accelerations on the printed circuit boards.

Introduction

The structure of the SACI-1 satellite follows the small satellite's trend towards a modular design. According to this approach the subsystems are designed as standard modules. The use of a standard lay-out also makes the whole preliminary activity of design faster and easier. Another advantages are compactness, high reliability, and lower production costs.

The lay-out adopted for the SACI-1 favours its spin stabilization and thermal control. Very much care was taken in order to guarantee the electromagnetic compatibility between the payload, that is partially located in the Payload Section of the Mainbody and in the deployable panels, and the electronic equipments assembled in the Mainbody Platform. Field of view requirements of the payload were also satisfied. Fig. 1 is a lateral view of the satellite in which the main parts are identified. Views from the top and bottom sides are presented in Figures 2 and 3.

In the interface between the satellite and the launcher there is a cone adapter (Adapter). This part includes the separation mechanism and the umbilical electrical lines. The satellite is secured by a rod that connects the Mechanical Interface Module to the cone adapter (see Fig. 1). This assembly also includes a cutter or releasing mechanism, a spring (that pushes the satellite) and some dissimilar materials stacked in the interface between the Adapter and the interface module, whose purpose is to reduce the mechanical shock transmitted to the Mainbody during the separation from the launcher.

The structure is designed to withstand, without failure or excessive distortion, the static, dynamic and thermal stresses that occur during launch, deployment and service. Besides that the structure also has to secure the payload and the most sensitive electronic parts against excessive distortions, vibrations, temperature changes, and undesirable radiations.

The mechanical design using standard modules as structural building blocks is the reason of the major challenge of the structural analysis of this satellite. This is because it is not possible to separate the primary structure from the equipments, as in a "conventional" design. This means that there is no structural "cushioning" to lower the shock and vibration levels that the electronic parts have to endure during the launching. This fact is a burden to the structural model, which must provide reliable information about the behaviour of the printed circuit boards assembled inside the horizontal modules.

Another demand to the model lies on the fact that it embraces not only what is considered the Structure Subsystem (Mainbody and Adapter), but also the deployable panels and their mechanisms. The panels and the mechanisms have been kept in separate subsystems in order to simplify the documentation necessary to their procurement (the structure is being manufactured in-house but not the panels and mechanisms). This strategy, although increasing the size of the model, avoids the frequent exchange of data between several partial models.

Structure Description

SACI-1 overall shape resembles a parallelepiped measuring 600 x 400 x 400 mm (Mainbody) and weighting about 65 kg (Mainbody and Adapter). The Platform consists on a pack of 9 aluminum ribbed frames of different thickness stacked horizontally and connected by means of 12 stud bolts. The Payload is assembled on the top of the Platform and has six ribbed plates that form a box in which most of the payload is assembled.

The standard module mentioned above has a squared shape with a 375 mm side. The strength requirements are satisfied by an inner frame. The printed circuit boards are assembled on this frame (see Fig. 4). Other modules are designed according to some special needs of the subsystem. Figures 5, 6, and 7 present three examples. The first one

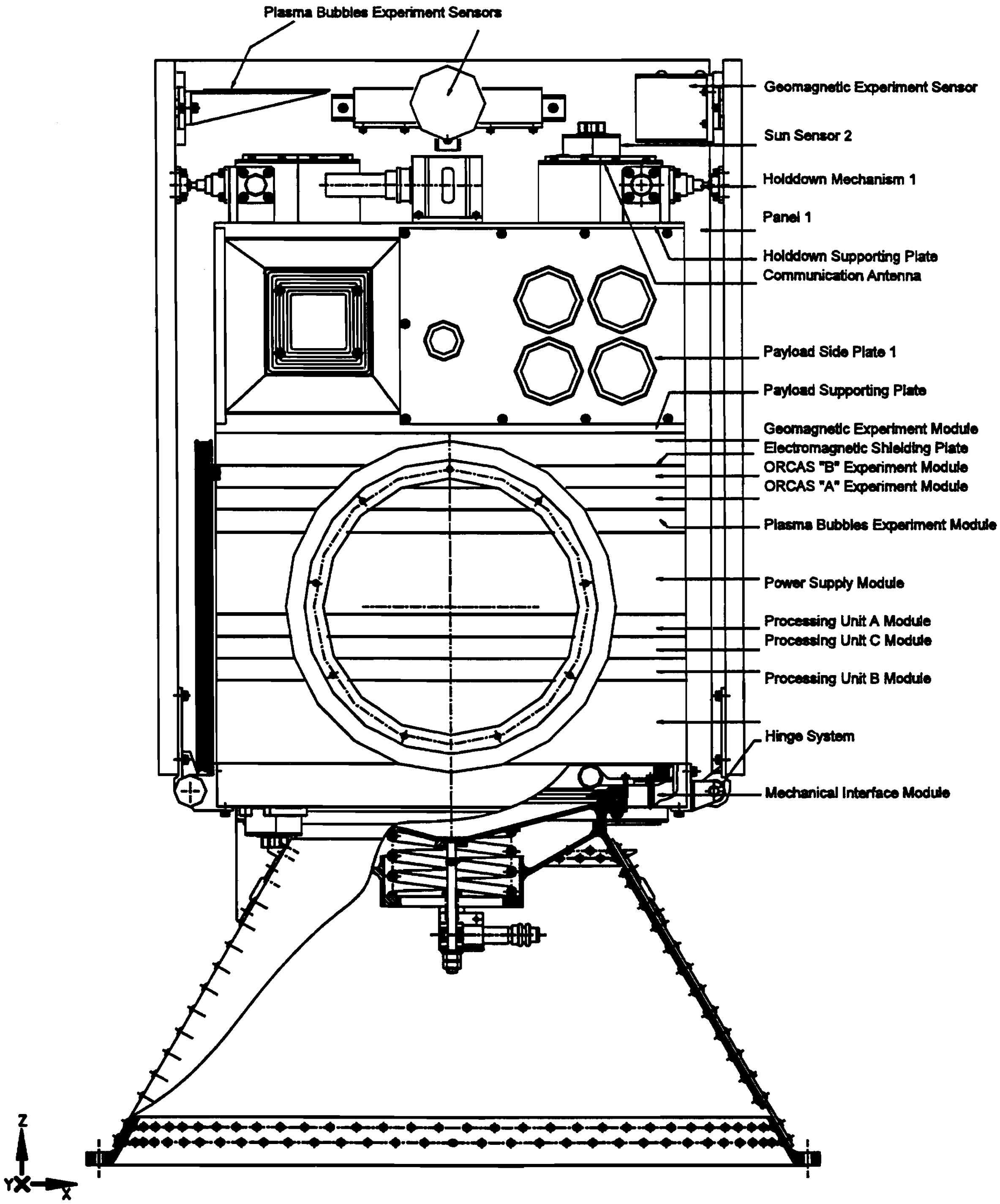

Fig. 1: SACI-1 side view.

(Fig. 5) is the Communication Module that holds several small boxes containing parts like transmitters and decoders. The second one (Fig. 6) is the Batteries Module that contains the batteries and the power supply circuitry. The last one (Fig. 7) is the Mechanical Interface Module. This Module holds a Nutation Damper, a Torque Coil, a Solar Sensor, two Antennas and provides the interface to the Adapter.

The satellite has also four deployable panels measuring 440 x 570 mm that support the solar arrays and four payload sensors (there is one sensor in each deployable panel back side tip). Each panel has a pair of hinges and a set of HDR mechanisms (1 holddown and 2 snubbers). All parts mentioned above are shown in Figures 2 and 3.

The junction of the modules is accomplished by 12 stud bolts ¼" in diameter. Four pins located in the corners of each module guarantee their alignment and prevent relative lateral movements.

All parts of the Platform and Payload are made from 6061 T 651 aluminum alloy plate or rod machined to their final shape.

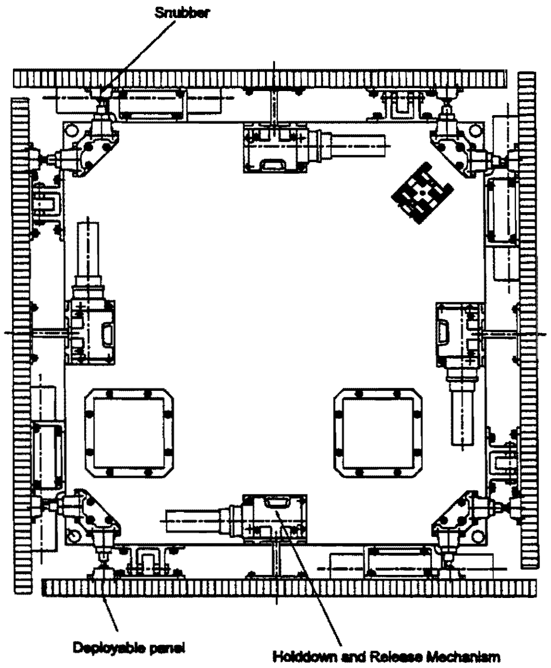

Fig. 2: SACI-1 top side showing the deployable panels and the holddown and release mechanisms.

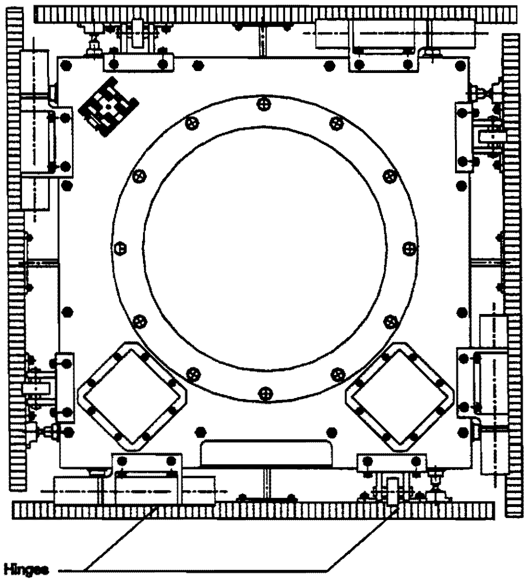

Fig. 3: SACI-1 bottom side showing the hinge assembly and interface to the Adapter.

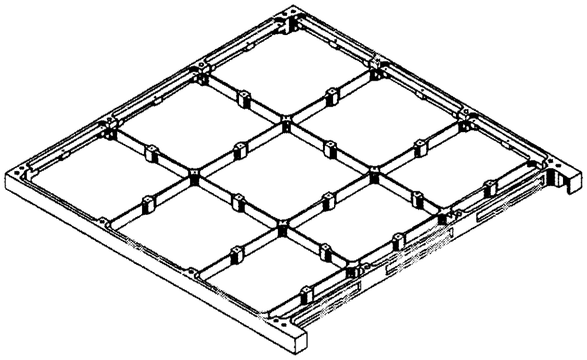

Fig. 4: Standard Module.

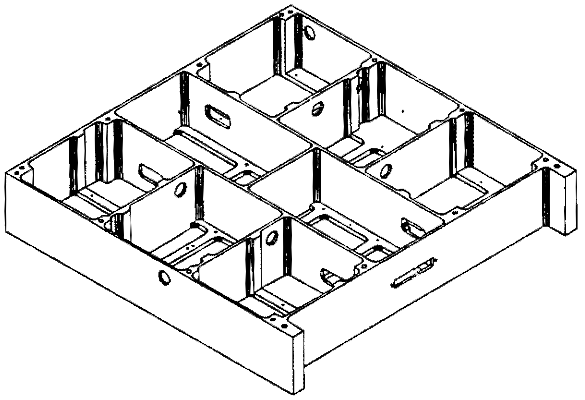

Fig. 5: Communication Module.

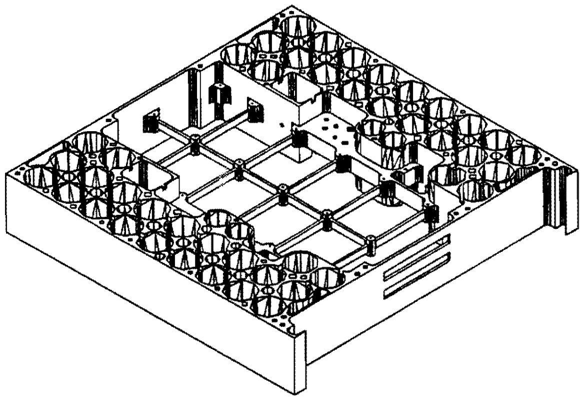

Fig. 6: Batteries Module.

Performance Requirements

The Structure, as an integral part of the spacecraft, should be capable of supporting the spacecraft total mass of 65 Kg and, in order to guarantee the controllability of the launcher, the satellite fundamental frequencies, considering the Adapter in the frequency determination, should be:

- Minimum transverse: 20 Hz;
- Minimum longitudinal: 35 Hz;
- Minimum z axis torsion: 25 Hz.

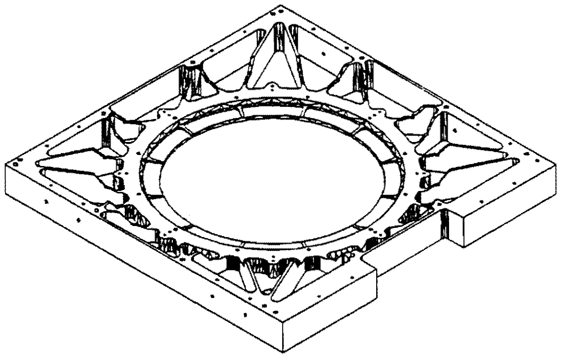

Fig. 7: Mechanical Interface Module.

The structure should maintain all its specified dimensions, alignments and tolerances during the mission lifetime after being submitted to all handling, testing, transportation, launching and orbital loads.

Dynamic analysis should determine responses to the qualification sinusoidal and random vibration levels and define notching frequencies and levels to prevent excessive loads in dynamic tests. In these response analysis, a modal damping of 10% should be used for the primary structure modes.

Loads

The testing loads provided by the launching agency for the structural dimensionning are, as it is usually done, classified in three basic cases: Quasi-Static, Sinusoidal and Random. Tables 1, 2 and 3 present the levels for all cases.

Table 1: Loads for the Quasi-Static Analysis.

Case	**Acceleration (g)**		
	X	**Y**	**Z**
1	2.00	0.00	-11.24
2	0.00	2.00	-11.24
3	3.14	0.00	-4.66
4	0.00	3.14	-4.66
5	2.80	0.00	-4.94
6	0.00	2.80	-4.94
7	2.00	0.00	5.82
8	0.00	2.00	5.82

Table 2: Sinusoidal Vibration Test Levels.

Frequency (Hz)	**Acceleration or Displacement (peak-to-peak)**	
	Qualification	**Acceptance**
5 - 8	9.4 mm	6.2 mm
8 - 100	1.2 *g*	0.8 *g*

Table 3: Random Vibration Test Levels.

Frequency (Hz)	**Spectrum Density**	
	Qualification (8.8 grms)	**Acceptance (6.22 grms)**
20 - 100	+ 3 dB/oct	+ 3 dB/oct
100 - 600	0.08 g^2/Hz	0.04 g^2/Hz
600 - 2000	- 6 dB/oct	- 6 dB/oct

Factors of Safety

The following design load factors of safety should be applied to qualification loads to obtain the structural design yield and the design ultimate loads:

- Yield: 1.20;
- Ultimate load: 1.35.

For special parts, whose structural properties distributions are not completely known, additional safety load factors should be employed and the final value should not be less than 1.33 for yield and 2.00 for ultimate loads.

The structure should be designed to withstand the quasi-static qualification loads multiplied by the design safety factors and to the dynamic loads resulting from the response to qualification sinusoidal and random inputs.

Margins of Safety

Safety margins should be calculated as per formula:

$$\mathrm{MS} = \mathrm{Minim.}_{i=1,m}\left(\frac{\text{critical stress}}{\text{qual. stress} \times n}\right)_i - 1$$

where "m" is the number of combined failure modes, "qual. stress" stands for the stress induced by the environmental qualification loads, "n" is the corresponding ultimate safety factor and "critical stress" is the maximum allowable stress, above which a failure is expected.

Structural Analysis

Structural Mathematical Model

A detailed three-dimensional finite element model of the satellite was developed using MSC/XL[1] graphical pre and post-processor. The model has 5944 elements and 5974 grids. The type of the elements and their quantities are as follows:

• ELAS2 (scalar spring element);	12
• QUAD (quadrilateral plate element);	4144
• BAR (bar element);	1696
• TRIA3 (triangular plate element);	92

This model was processed by the MSC/NASTRAN[2] finite element program version 68.2 in a HP 9000/730 workstation. The purpose of the model is to predict displacements, loads, stresses, deformations, natural frequencies of the structure and accelerations of the printed circuit boards. Figure 8 shows the full finite element model.

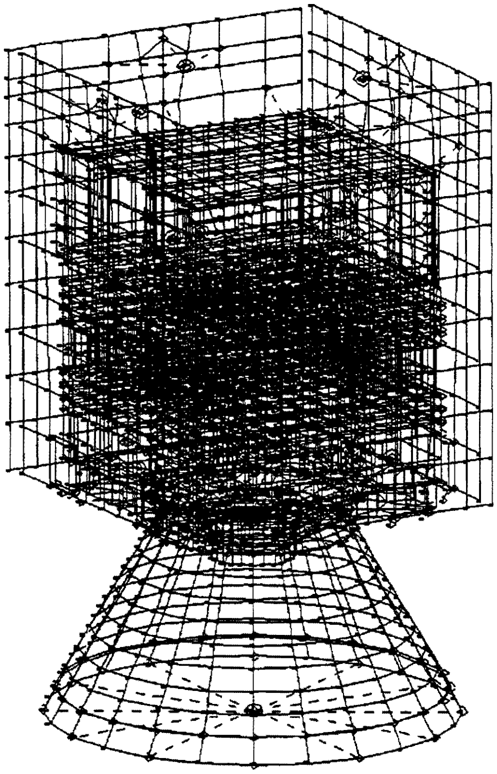

Fig. 8: Full SACI-1 FEM model.

The structure was numerically verified throughout static, modal, sinusoidal, and random analysis, as described below.

Quasi-Static Analysis

From the data provided by the launching agency, eight quasi-static load cases were considered by the analysis (Table 1). Figures 9 to 12 show the level of Von Mises stresses in the structure under the loads of Cases 1, 3, 5 and 7. Case 1 causes the highest stress levels (110 MPa, not visible in Fig. 9) in a bar element of the Communication Module. Considering an allowable yield stress of 200 MPa for the aluminum, this gives a minimum margin of safety equal to 0.81.

Modal Analysis

In order to verify if the structure fulfils the stiffness constraints, a natural frequency analysis was performed to determine the first three natural frequencies. Figures 13 to 15 show these vibration modes and their natural frequencies. The global modes are two lateral and one longitudinal and all frequencies are above the specified lower bound.

Sinusoidal Analysis

This analysis was conducted to determine accelerations, displacements and stresses on the structure submitted to the sinusoidal spectrum described in Table 2. In this analysis the modal frequency response method was used. This method uses the mode shapes of the structure to reduce the size of the system of equations to be solved uncoupling the equations of motion and consequently making the numerical solution more efficient. The frequency range of sinusoidal response analysis was spanned up to the frequency of the higher normal elastic mode already calculated by the modal analysis. The damping was applied to each mode separately in order to maintain the equations of motion uncoupled. In order to show the kind of results that can be obtained by this method, the next figures present some results calculated to the Mechanical Interface Module (MIM):

- Figure 16 presents the level of stresses in the bar elements;
- Figure 17 exhibits the level of stresses in the plate elements;
- Figures 18 to 20 display the acceleration in the three directions of the grids;
- Figure 21 presents the accelerations observed in the Grid 1186 of the MIM.

All these results are obtained when the structure is excited in the X direction.

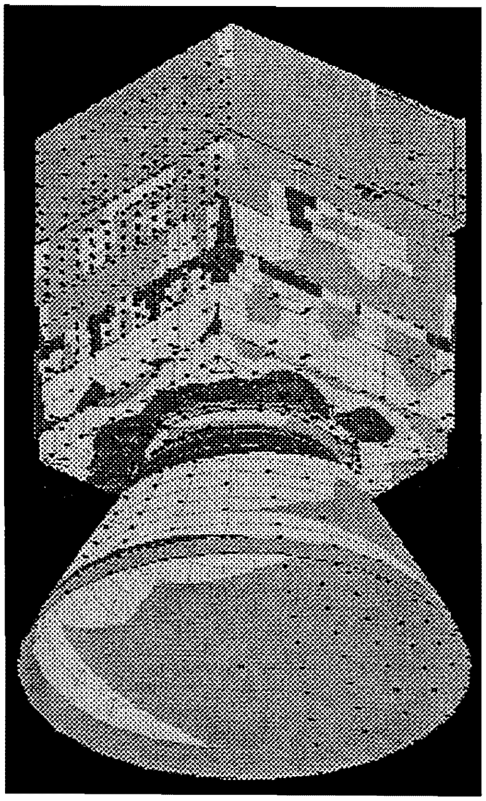

Fig. 9: Case 1. Stresses from 0.03 (green) to 26 (red) MPa.

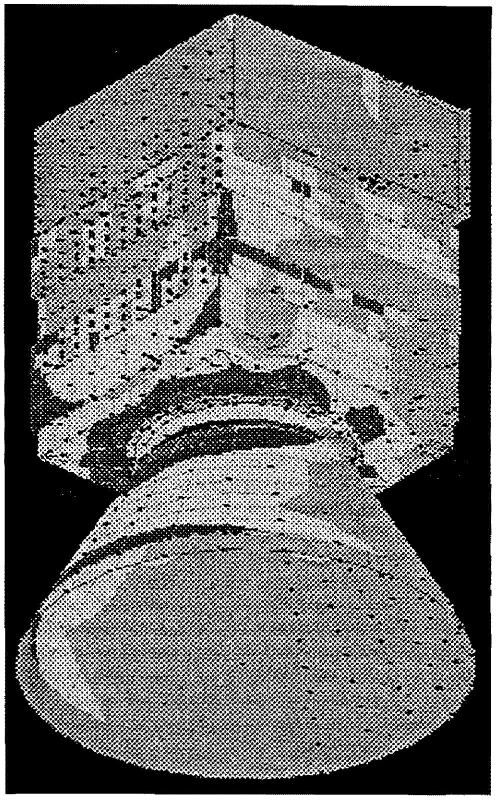

Fig. 11: Case 5. Stresses from 0.02 (green) to 17 (red) MPa.

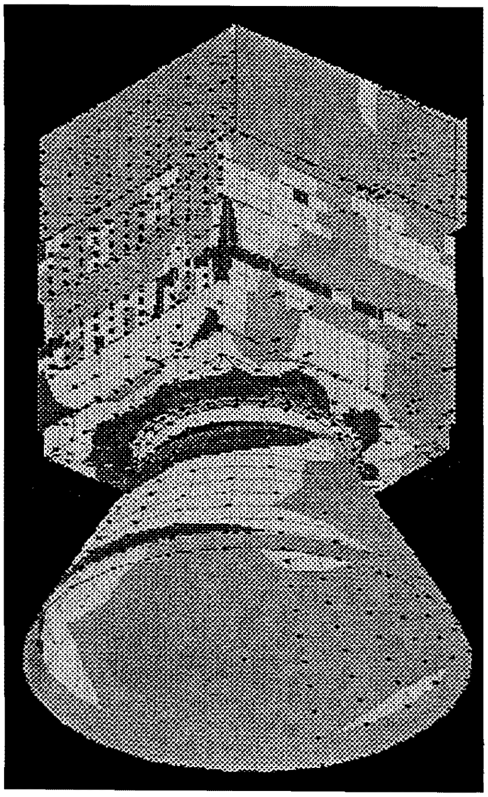

Fig. 10: Case 3. Stresses from 0.01 (green) to 18 (red) MPa.

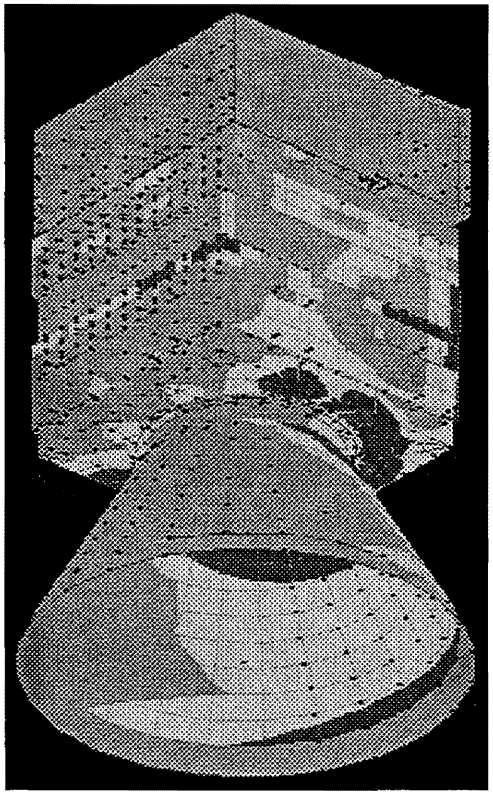

Fig. 12: Case 7. Stresses from 0.01 (green) to 17 (red) MPa.

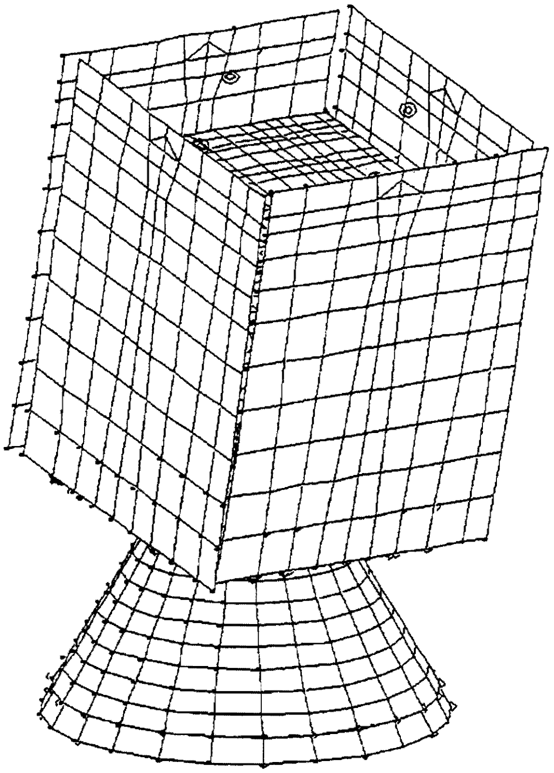

Fig. 13: Mode 1 natural frequency (37 Hz, lateral).

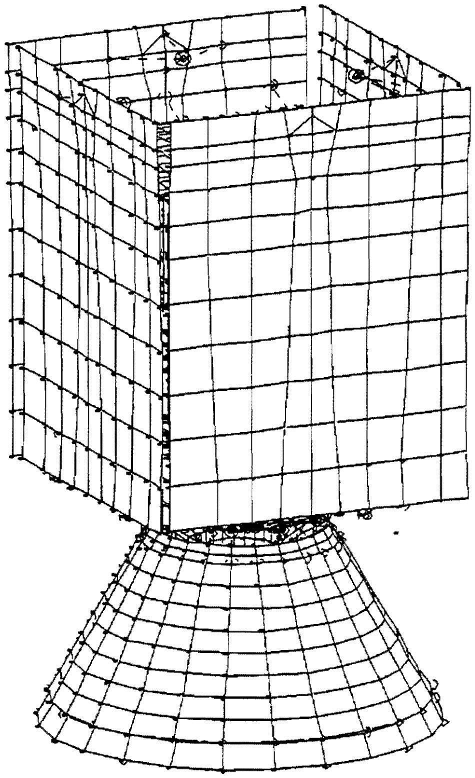

Fig. 15: Mode 3 natural frequency (59.7 Hz, longitudinal).

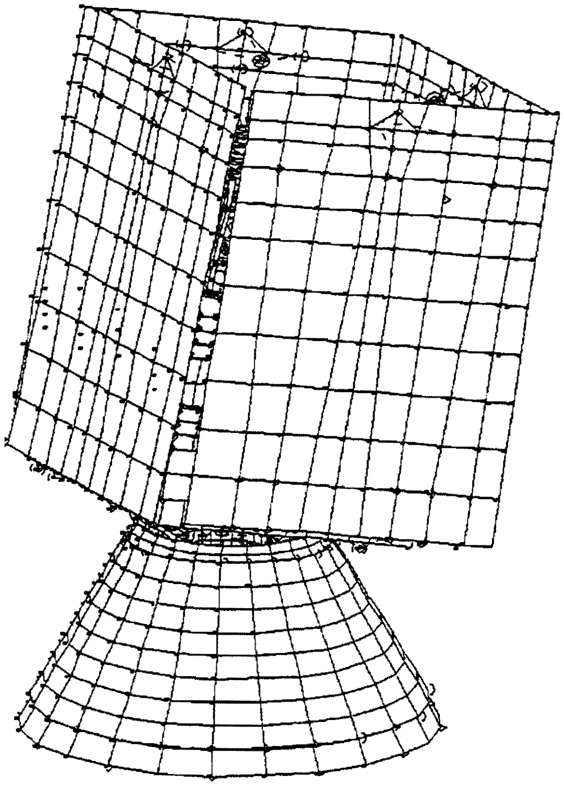

Fig. 14: Mode 2 natural frequency, (38.1 Hz, lateral).

Random Analysis

This analysis was conducted to determine the accelerations of the printed circuit boards (PCB's) as much as displacements and stresses on the structure subject to the random excitations described in Table 3. The random analysis was performed as a post-processing step after frequency response analysis. The frequency analysis is used to generate the transfer function and obtain the power spectral density (PSD) response from the PSD input. As output of this analysis the program gives the response PSD, autocorrelation functions, number of zero crossings with positive slope per unit time, and RMS (root-mean-square) values of response acceleration.

This analysis comprises modes up to 800 Hz, which includes more than 95% of the effective mass. The random response analysis was performed for the frequency range from 0 Hz to 2000 Hz. The variation of the acceleration along the structure due to the random excitation is presented in Figures 22 to 24. They are related to the Mechanical Interface Module, Processing Unit B Module (the PCU-B Module is similar to the Standard Module shown in Fig. 4) and Geomagnetic Experiment Module (Geomag Module). Fig. 1 shows their actual position in the satellite.

The MIM is the closest to the base, consequently it presents the lowest levels of acceleration. The same accelerations grow with the distance from the base, achieving the highest values, among the three modules, in the Geomag Module.

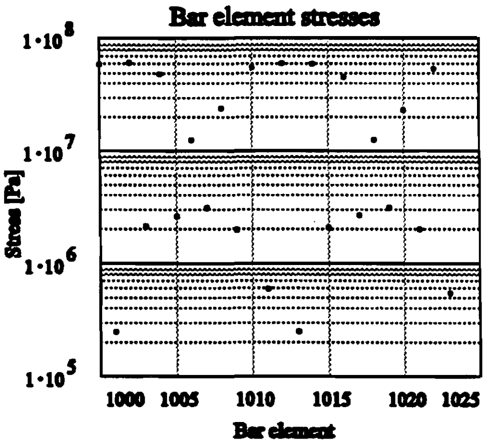

Fig. 16: MIM bar element stresses.

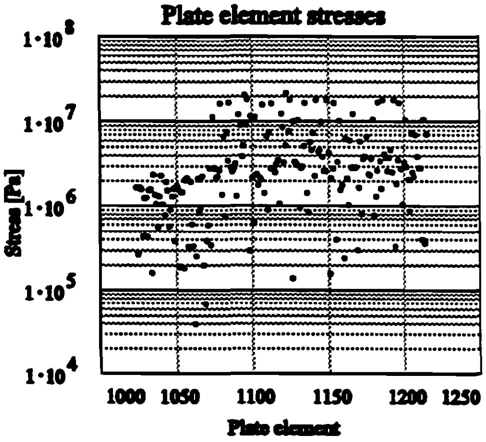

Fig. 17: MIM plate element stresses.

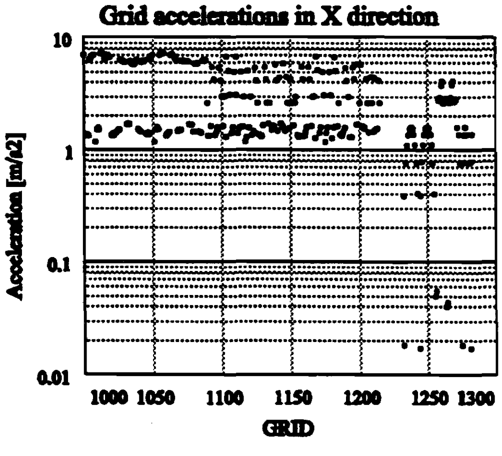

Fig. 18: MIM grid accelerations in the X direction.

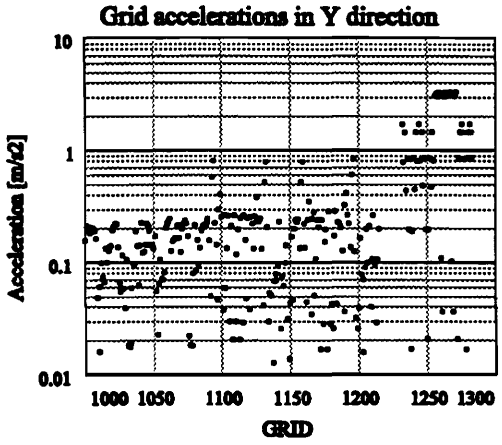

Fig. 19: MIM grid accelerations in the Y direction.

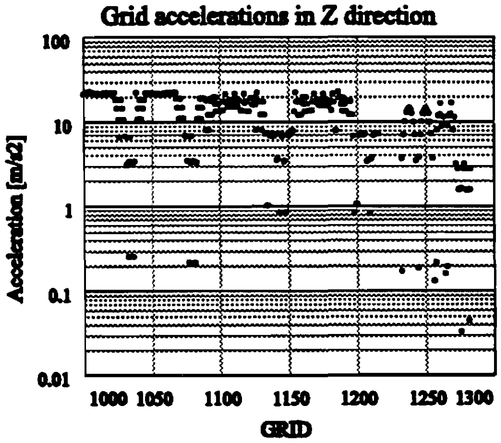

Fig. 20: MIM grid accelerations in the Z direction.

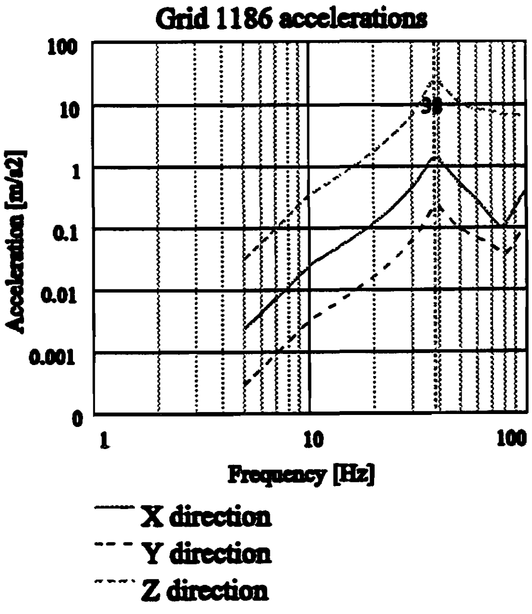

Fig. 21: Grid 1186 accelerations.

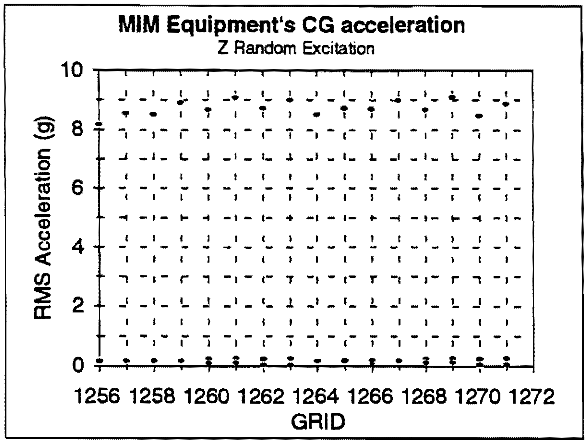

Fig. 22: Random accelerations of the MIM equipment's centre of mass. The highest values are in the Z direction.

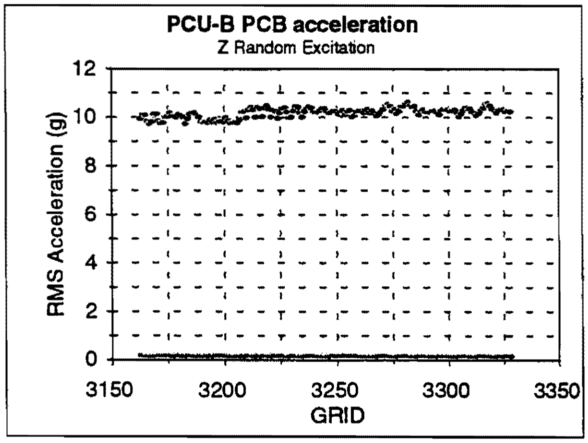

Fig. 23: Random acceleration of the PCU-B module PCB. The highest values are in the Z direction.

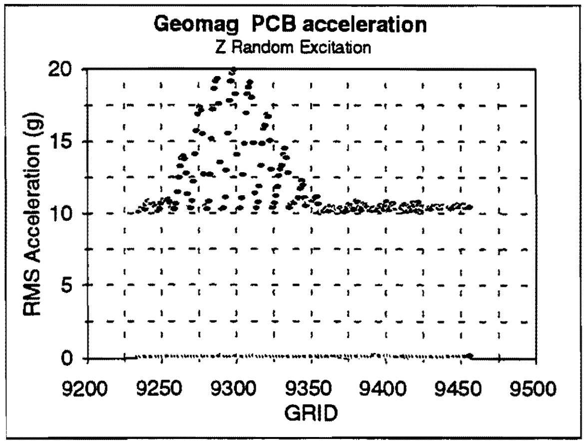

Fig. 24: Random accelerations of the Geomag module PCB. The highest values are in the Z direction.

Conclusion

This paper presented a brief description of the structural design and analysis of a small scientific satellite (SACI-1) with the goal of providing an account of the mechanical design of the structure and the kind of analysis required for its qualification. The analysis results have shown that the design is feasible and, due to the level of detail of the model, no great discrepancies are expected between these results and the test ones.

References

[1] The Macneal-Schwendler Corporation, *MSC/XL User's Manual*, Version 3, 1991.

[2] The Macneal-Schwendler Corporation, *MSC/NASTRAN Reference Manual*. Version 68, 1994.