ANALYSIS AND VALIDATION OF THE STRUCTURE OF PRATHAM, INDIAN INSTITUTE OF TECHNOLOGY BOMBAY'S FIRST STUDENT SATELLITE

Ankit Chiplunkar*, Ramanath Pai**, Anirudh Subramanyam***

* Department of Aerospace Engineering, Indian Institute of Technology Bombay, Powai, Mumbai 400076. (E-mail: ankitchiplunkar@gmail.com)
**Department of Mechanical Engineering, Indian Institute of Technology Bombay, Powai, Mumbai 400076. (E-mail: ramanathanator@gmail.com)
***Department of Chemical Engineering, Indian Institute of Technology Bombay, Powai, Mumbai 400076. (E-mail: anirudh.381@gmail.com)

ABSTRACT
‘Pratham’, is a nano-satellite built by the students of IIT Bombay and is slated for launch by the Indian Space Research Organization (ISRO) in the third quarter of 2011. This paper discusses the work done by the Structures Sub-system of Pratham. The objective of the sub-system is to ensure the robustness of the satellite structure so that it survives launch loads. A finite element model of the satellite structure has been made and representative launch loads have been applied. Various static and dynamic analyses have been performed on the satellite structure to obtain the response. Finite Element Analyses of the printed circuit boards (PCBs) onboard the satellite have also been performed. The FEA results have been validated in 2 stages: the geometry was validated by comparing with theoretical results while the element types were validated by comparing with analyses of isolated individual structural elements. The results suggest that the satellite will maintains its structural integrity during launch and that no component of the satellite will fail during launch.

KEYWORDS:
Finite element analysis; Launch load vibrations; Microsatellite; Validation.

INTRODUCTION
‘Pratham’ is a microsatellite with a payload to measure the total electron count of the ionosphere and perform the tomography of ionosphere. During launch, Pratham is expected to withstand static loads of about 9 times the gravitational force and dynamic loads leading to vibrations. Under such harsh conditions, it is imperative that the structure of a satellite withstand these loads with little or no deformation so that the internal circuitry and actuators are not damaged and the satellite can execute the complete life cycle it was designed for. This paper discusses the various analyses that were carried out to ensure that the structure of the satellite satisfies the above requirements. The design approach has been briefly explained. A CAD model of the satellite is prepared and it is then meshed to obtain the Finite Element Model. FE models of the components onboard, including that of a typical Printed Circuit Board (PCB) are made. Simulation conditions, including simulation of launch loads and physical constraints have been discussed. Validation of the analysis is done by comparing with theoretical results for the same geometry and validation of the elements used for analysis is done by comparing with results that would be obtained for some standard geometry.
REQUIREMENTS AND CONSTRAINTS

Pratham is expected to satisfy the following requirements:

1. **Launch Vehicle Placement**: The satellite is launched into Low Earth Orbit by the Polar Satellite Launch Vehicle. The launch vehicle interface to be used is the IBL230V2, to be provided by VSSC.
   - Launch vehicle interface requires 8 M6x1, 9mm long helicoil inserts at 230mm PCD on bottom deck of the satellite.
   - There may be no interference in the joint from the satellite to the launch vehicle body.

2. **Launch Loads**: The satellite is carried to its orbit by a launch vehicle in a flight lasting about 17 minutes. The high levels of acceleration, vibrations and shocks experienced by the vehicle during this period are transmitted to the payloads attached to the flight decks of the vehicle and impose strict requirements on the overall structure. The loading specification for which the launch vehicle interface is tested is assumed to be the loading data for the satellite during launch and has been specified in the IBL230V2 documentation.

   - **Static Loads**: Lateral loads are considered to act simultaneously with longitudinal loads. Earth’s gravity is also included in the levels given below. All loads apply at the centre of gravity of the satellite as body forces. The longitudinal and lateral axes are defined later.

<table>
<thead>
<tr>
<th>Direction</th>
<th>Loading</th>
</tr>
</thead>
<tbody>
<tr>
<td>Longitudinal</td>
<td>± 11g</td>
</tr>
<tr>
<td>Lateral</td>
<td>± 6g</td>
</tr>
</tbody>
</table>

   **Table 1**: Static loading levels

   - **Harmonic Loads**: The qualification loading levels are used for analysis of the satellite structure.

<table>
<thead>
<tr>
<th>Frequency Range (Hz)</th>
<th>Qualification Level</th>
<th>Acceptance Level</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Longitudinal axis</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5-10</td>
<td>10mm (DA) 3.75g</td>
<td>8mm (DA) 2.5g</td>
</tr>
<tr>
<td>10-100</td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Lateral axis</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5-8</td>
<td>10mm (DA) 2.25g</td>
<td>8mm (DA) 1.5g</td>
</tr>
<tr>
<td>8-100</td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Sweep rate</strong></td>
<td>2 oct/min</td>
<td>4 oct/min</td>
</tr>
</tbody>
</table>

   **Table 2**: Sine sweep test levels
- **Random Vibration Analysis**: Random vibration levels represent a more real world environment than others and are specified to check if the electronics can survive launch load conditions. As in Harmonic Analysis, qualification levels are used for Random Vibration Analysis.

<table>
<thead>
<tr>
<th>Frequency (Hz)</th>
<th>Qualification PSD ( \frac{g^2}{Hz} )</th>
<th>Acceptance PSD ( \frac{g^2}{Hz} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>0.002</td>
<td>0.001</td>
</tr>
<tr>
<td>110</td>
<td>0.002</td>
<td>0.001</td>
</tr>
<tr>
<td>250</td>
<td>0.034</td>
<td>0.015</td>
</tr>
<tr>
<td>1000</td>
<td>0.034</td>
<td>0.015</td>
</tr>
<tr>
<td>2000</td>
<td>0.009</td>
<td>0.004</td>
</tr>
<tr>
<td>g RMS</td>
<td>6.7</td>
<td>4.47</td>
</tr>
<tr>
<td>Duration</td>
<td>2 min/axis</td>
<td>1 min/axis</td>
</tr>
</tbody>
</table>

**Table 3**: Random vibration levels

3. **Stiffness**: No component onboard the satellite is free to vibrate at a natural frequency below 90Hz, i.e., the global fundamental frequency must be greater than 90Hz. This requirement is applicable to all extended structures as well as the structure as a whole.

**DESIGN AND CONFIGURATIONAL LAYOUT**

**Design Approach**
The satellite structure weighs 9.57kg and it is a 254mm x 254mm x 254mm cube, excluding the monopole antennae.

**Full Configuration Layout**
The satellite essentially consists of six structural members: the panels which make the box of the body.

- **Zenith Side**: The Zenith side always faces away from the earth. The Zenith side incorporates a solar panel and the GPS antenna along with GPS circuit and a magnetorqu器.
- **Nadir Side**: The Nadir side always faces the earth. The Launch Vehicle interface, IBL-230 V2, will be attached to Nadir side. It also houses the battery and the battery box.
- **Sun Side**: The Sun side faces the sun and has a solar panel mounted on it. It contains the power circuit and a magnetorquer inside.
- **Antisun Side**: The side opposite to the Sun side, 3 monopole antennae are attached to it. It also has the monopole circuit and the beacon circuit.
- **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 magnetorqu器 and the OBC circuit.
- **Lagging Side**: Opposite to the Leading side, the lagging side contains the magnetometer and a solar panel.
ANALYSIS

Modelling
The entire satellite structure was modelled in Solid Works 2009; the joints were modelled using its “mate” function. As the joints are assumed to be at discrete points where the screws are placed, this model provides a highly accurate representation of the structure. The components on the PCBs were not modelled since their masses are insignificant and have negligible effect on vibrations of the PCB. To obtain board-level responses, a dummy board was assembled on the sun-sensor board located on the zenith. It includes standard components like connectors, ICs, capacitors, etc. They are not representative of the actual board.

Figure 1: CAD model of the dummy PCB showing co-ordinate axes (made in SolidWorks 2009)

Simulations and Results
Simulations were performed on:
- **Qualification Model (QM):** Consisting of all components onboard the satellite, except the fasteners. The joints were appropriately mated to accurately represent the final model.
- **Circuit Board:** A dummy board was simulated to launch loads flight qualification.

Presented below are simulations and results of the QM and the PCB: All simulations were performed in ANSYS Mutliphysics and ANSYS Workbench (v11).

**Qualification Model:**
The Cartesian co-ordinate system is used with origin at the base of the Nadir of the satellite. The longitudinal axis used for this analysis is along Y axis (axis joining Nadir and Zenith).

Figure 2: CAD model of satellite showing co-ordinate axes (made in SolidWorks 2009)
Boundary conditions: The part of launch vehicle interface attached to the satellite is constrained in space. This is a correct assumption as in actual launch it will be constrained using ball lock mechanism constraining all the degrees of freedom.

1. **Static Analysis of QM:**
   - Loads applied: As specified in Table 1.
   - Results:
     - Maximum von Mises stress (occurs on a washer): 20.51MPa (4.6% of failure stress)
     - Maximum total deformation (at the tip of the middle antenna): 0.214mm

<table>
<thead>
<tr>
<th>Mode</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frequency</td>
<td>152.4</td>
<td>154.45</td>
<td>155.21</td>
<td>155.46</td>
<td>155.90</td>
<td>156.14</td>
<td>168.04</td>
<td>168.41</td>
<td>170.47</td>
<td>195.41</td>
</tr>
</tbody>
</table>

Table 6: Modal frequencies of the satellite (in Hz)
• The first 6 modes are antenna transverse modes. The first panel mode is the 7th mode of the structure and is a breathing mode. The first side mode (Zenith) is the 20th mode (=416.48 Hz) of the satellite. Of the first 25 modes extracted (25th free-free mode = 572.93 Hz), no PCB modes were observed.
• The fundamental frequency of the structure is > 90 Hz as demanded by global stiffness requirements. We can deduce that the structure will not resonate with the Launch Vehicle Interface and will survive the launch without any deformation. Moreover, the absence of PCB modes is reassuring for the survival of onboard PCBs.

3. Harmonic Analysis:
• Additional constraints applied: The structure was assembled on a block having a weight 100 times that of the satellite. This was done to make the satellite’s structure flexible and not rigid.
• Loads applied: 2 analyses were performed. In the first case, a pressure equivalent to an acceleration of 3.75g in the Y direction was applied to the base of the block. In the second case, a pressure equivalent to an acceleration of 2.25g was applied in the X direction. (As specified in Table 2)
• Frequency Range: 10Hz-200Hz
• Solution Method: Mode superposition
• Damping: 1%
• Results: Data for the middle antenna are given below. Graphs were also obtained.

<table>
<thead>
<tr>
<th></th>
<th>Maximum normal stress along y axis (σ_y)</th>
<th>% of yield stress</th>
<th>Maximum y-deformation</th>
<th>Frequency(Hz)</th>
<th>Phase angle (maximum stress)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Longitudinal</td>
<td>15.9MPa</td>
<td>5.76</td>
<td>3.5mm</td>
<td>152.5</td>
<td>89.31°</td>
</tr>
<tr>
<td>Lateral</td>
<td>7.9MPa</td>
<td>2.86</td>
<td>0.17mm</td>
<td>154.4</td>
<td>106.21°</td>
</tr>
</tbody>
</table>

Table 7: Results of Harmonic Analysis

Figure 3: σ_y and displacement in y direction versus frequency respectively (longitudinal loading)
Figure 5: $\sigma_y$ and displacement in y direction versus frequency respectively (lateral loading)

- A peak is observed in each graph at a frequency near the first mode of the structure. It is thus consistent with the modal analysis. Stresses are far less than the failure values; deformation in lateral mode is less. Deformation in longitudinal mode is slightly higher, but that is acceptable as it is the displacement of the antenna tip.
- Within the frequency range (10 – 100 Hz) specified by the Interface Control Document of the LVI, the satellite does not undergo failure in any conceivable mode.

4. Random Vibration Analysis:
- Loads applied: PSD base excitation as specified in Table 3 was applied along the x-axis.
- Number of modes used for analysis: All (25)
- Damping: 1%
- Probability: 68.3%
- Results:

| Maximum von Mises stress (occurs on a washer) | 24.97MPa (5.6% of failure stress) |
| Deformation along x-axis (relative to base motion) | 0.17mm |
| Deformation along y-axis (relative to base motion) | 0.06mm |
| Deformation along z-axis (relative to base motion) | 0.06mm |
| Maximum $\sigma_x$ | 27.63MPa |

Table 8: Random vibration analysis

- The maximum stress is obtained on a washer and is far less than the yield strength. Also, the total deformation is far too less for any contact between 2 surfaces to take place.
- It can be said with confidence that the structure will not fail during launch.
Harmonic Response of Circuit Board:
With the help of the harmonic response described previously, the displacement and acceleration responses of a node on the Zenith side’s sun-sensor board were calculated.

Figure 6: Acceleration response of node on the sun sensor board

Certain standard components were assembled on this board and the board was given the acceleration response so obtained. It was then analyzed for its stress response. The co-ordinate system used was carried over from the simulation of the QM.

- Loads applied: An acceleration of $(5.1923 \times 10^{-2}, 1.0563, 0.29655) \text{ m/s}^2$ was given to the board. This corresponds to the peak value of acceleration obtained in the acceleration response described previously. It was obtained at a frequency of 194.3 Hz, and the y component of the acceleration peak was obtained at a phase angle of $171.01^\circ$.
- Frequency Range: 10Hz-200Hz
- Solution Method: Mode superposition
- Damping: 1%
- Results:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value (MPa)</th>
<th>Component</th>
<th>Frequency (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum von Mises stress</td>
<td>4.77 (2.3% of failure stress)</td>
<td>Connector</td>
<td>194.3</td>
</tr>
<tr>
<td>Maximum $\sigma_y$</td>
<td>1.34</td>
<td>Bolt on connector</td>
<td>198.1</td>
</tr>
</tbody>
</table>

Table 9: Harmonic analysis results of PCB

- Conclusion: Stresses are very small compared to yield stress. PCB breakage due to launch loading is highly unlikely.
VALIDATION

**Validation of elements used in ANSYS**

It is important to first validate the element used for meshing. The results obtained from modal analysis of aluminium plate are compared to the theoretical results for first natural frequency. The theoretical natural frequency of the plate with all sides fixed is given by:

\[
f = \frac{k_1}{2\pi} \sqrt{\frac{Dg}{wa^2}}
\]

Where \(f\) is the natural frequency of the plate, \(k_1=35\) for a square plate, \(w/g\) is the mass per unit area and \(D\) is given by:

\[
D = \frac{E\tau^3}{12(1-\nu^2)}
\]

Here \(E\) is the modulus of elasticity, \(\tau\) is the thickness and \(\nu\) is the Poisson’s ratio. The result obtained using this formula was used to validate the element.

The geometry considered is a plate of sides 230x230mm\(^2\) and of thickness 6mm with material properties of Al6061T6. Six elements were considered. SOLID45 is an 8 node solid element. SOLID95 is 20 a node solid element, SHELL63 is a planar shell element, SOLID186 is a 20 mode structural solid hexagon and SOLID187 is a 10 node structural solid tetrahedron. The results obtained for first natural frequency are shown in Table 10.

<table>
<thead>
<tr>
<th>Element</th>
<th>Frequency (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theoretical</td>
<td>1001.32</td>
</tr>
<tr>
<td>SOLID45</td>
<td>2269.7</td>
</tr>
<tr>
<td>SOLID95</td>
<td>1002.6</td>
</tr>
<tr>
<td>SHELL63</td>
<td>999.58</td>
</tr>
<tr>
<td>SOLID186</td>
<td>999.6</td>
</tr>
<tr>
<td>SOLID187</td>
<td>995.7</td>
</tr>
</tbody>
</table>

**Table 9: Results of element validation**

Hence we can see that SOLID95, SHELL63, SOLID186 and SOLID187 give results that match theoretical results. SOLID186 and SOLID187 are used for further analyses since they are directly loaded in Workbench.

**Experimental Validation**

An experiment was carried out to validate simulation methodology and simulation model. The structure in figure 7 was fabricated and natural frequencies were calculated experimentally through a rap test. The same structure was modelled and simulated in ANSYS. The two results obtained are compared in table 11.
The first mode cannot be observed experimentally due to inaccurate positioning of the accelerometer. We may conclude that the overall satellite model is correct and there is little or no error in the analysis.

Validation of the satellite model

In this geometry, since the antenna is major contributing factor in stiffness characteristics of the model. The modal frequencies of the QM are compared to those obtained from antennae alone. The monopole holder was fixed and first two resonant frequencies were obtained. Since the first 6 modes of the structure are antenna modes, the 2 modes extracted from this model serve as a good representative of the first 2 QM modes. From a structural point of view, the monopole antenna attached to the satellite is similar to a cantilever beam. The formula for the natural frequency of a cantilever beam is given by:

\[ f = \left(1.875\right)^2 \sqrt{\frac{E l g}{w L^4}} \]

Where \( f \) is the natural frequency of the cantilever beam, \( E \) is the modulus of elasticity, \( l \) is the bending moment of inertia, \( L \) is the length of the cantilever beam, \( g \) is the gravitational constant, \( w \) is the weight of the cantilever beam. The result obtained using this formula is used to validate the element.

<table>
<thead>
<tr>
<th>Mode</th>
<th>QM frequency (Hz)</th>
<th>Antenna model frequency (Hz)</th>
<th>Theoretical</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>152.46</td>
<td>164.26</td>
<td>166.85</td>
</tr>
<tr>
<td>2</td>
<td>154.45</td>
<td>164.56</td>
<td>-</td>
</tr>
</tbody>
</table>

Table 11: Comparison of QM and antenna model

The close matches in the modal frequencies indicate that the entire satellite model is correct.
CONCLUSIONS

The aim of the project was to ensure robustness of the satellite structure to harsh launch loads. The structure was modelled in detail and analysed for modal, harmonic and random response. The results obtained estimate the maximum stresses and deflections to be less than 10% of failure values. The analyses and the element were validated using theory and experimentation and were found to be in agreement.

ACKNOWLEDGEMENTS

The authors wish to thank the Department of Aerospace Engineering, IIT Bombay for all the support they have provided.

The authors would particularly like to thank Prof Mujumdar, Prof. Sudhakar and Prof. Arya and also Mr. Mandar Kulkarni for his help in the structures lab.

Lastly, the authors thank the entire satellite team, especially the former members of the Structures team: Mr. Niranjan Parab, Ms. Haripriya and Mr. Ayanangshu Dey.

REFERENCES

List of references