Structural Analysis of Small Satellites

Spectral Color Ocean (SPOC)

Parker Ensing*[1], David L. Cotten^[1,2]

*pae10848@uga.edu, ^dcotte1@uga.edu, [1]Small Satellite Research Laboratory, University of Georgia,

[2]Center for Geospatial Research, University of Georgia,

Overview

The success or failure of satellites often depends on their abilities to complete designated tasks remotely without considerable intervention from people. Despite this, satellites need to survive their journey to prove their capabilities at all. For this reason, structural modeling, by means of finite element analysis, plays a crucial role in determining the survivability of satellites during launch and liftoff. At the University of Georgia's Small Satellite Research Lab, structural analysis was conducted on working models of the Spectral Color Ocean Color (SPOC) satellite as well as the Multiview Onboard Computational Imager (MOCI) satellite. While SPOC's funding comes from NASA and MOCI's funding comes the United States Air Force, both satellites share similarities regarding structural components, namely the 3-U CubeSat frame. Elements of each satellite such as the frame, fasteners, and structural ribs provide critical support against dangers like vibrations and inertial loading. Using the finite element analysis package ANSYS Workbench, these components, among others, were analyzed and simulated in a variety of different tests: inertial loading, modal vibration, random vibration, and response frequency. Using the data retrieved from these simulations, materials and locations for each part were chosen and adjusted throughout the entirety of the design process.

Structural Analysis Process

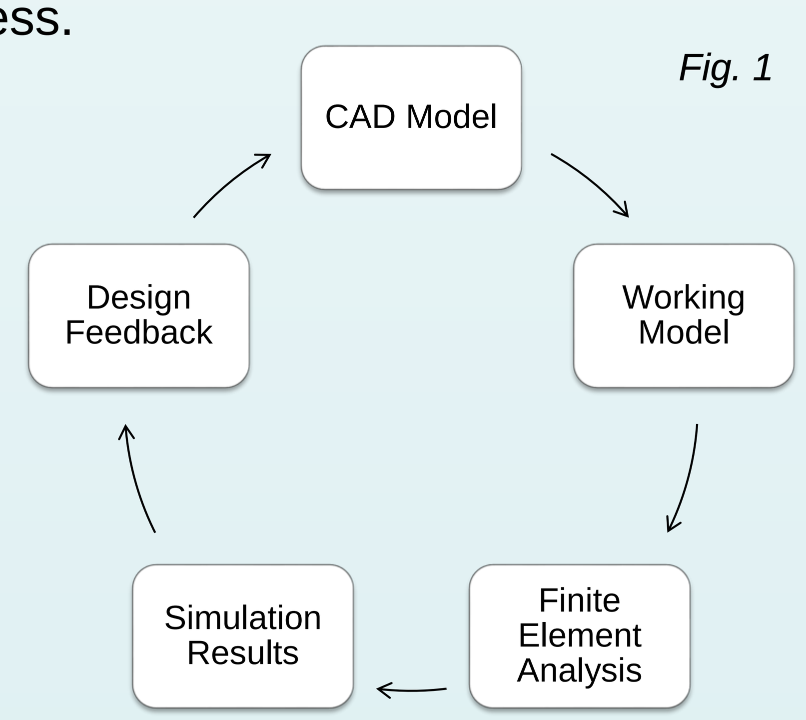

Fig. 1

Based on the classic engineering design process, which is cyclical in nature, the structural analysis process employed to simulate launch conditions follows the cyclical and iterative process featured in Figure 1.

The process begins with a 3-D model of the satellite. These models are designed using feedback from data collected by all project teams (i.e. mechanical team, electrical team, lab operations, etc.). As the project develops, the computer-aided design (CAD) model is constantly improved upon and modified to fit current needs and constraints. When structural analysis is needed, the current CAD model is converted to a working model. The working model is heavily simplified and removes excessive detail from the CAD model in order to reduce simulation times and processing requirements. This conversion process involves replacing detailed mock-ups of parts with basic geometry. For example, a model of a printed circuit board (PCB) with necessary electrical components is reduced down to a simple rectangular block that matches the dimensions of the board but lacks the specific geometric details and components. The only time parts are not simplified in the conversion process is when specific elements of a component are predicted to have a significant impact on the outcome of the analysis, such as cutouts and holes in the satellite frame.

After the CAD model has been converted into a working model suitable for analysis, finite element analysis can begin. For the SPOC and MOCI analysis, ANSYS Workbench was used to mesh and simulate the forces on the satellites. The analysis is broken into four different simulations: modal vibration, random vibration, response spectral, and inertial loading. Figure 2 shows the workflow between these simulations.

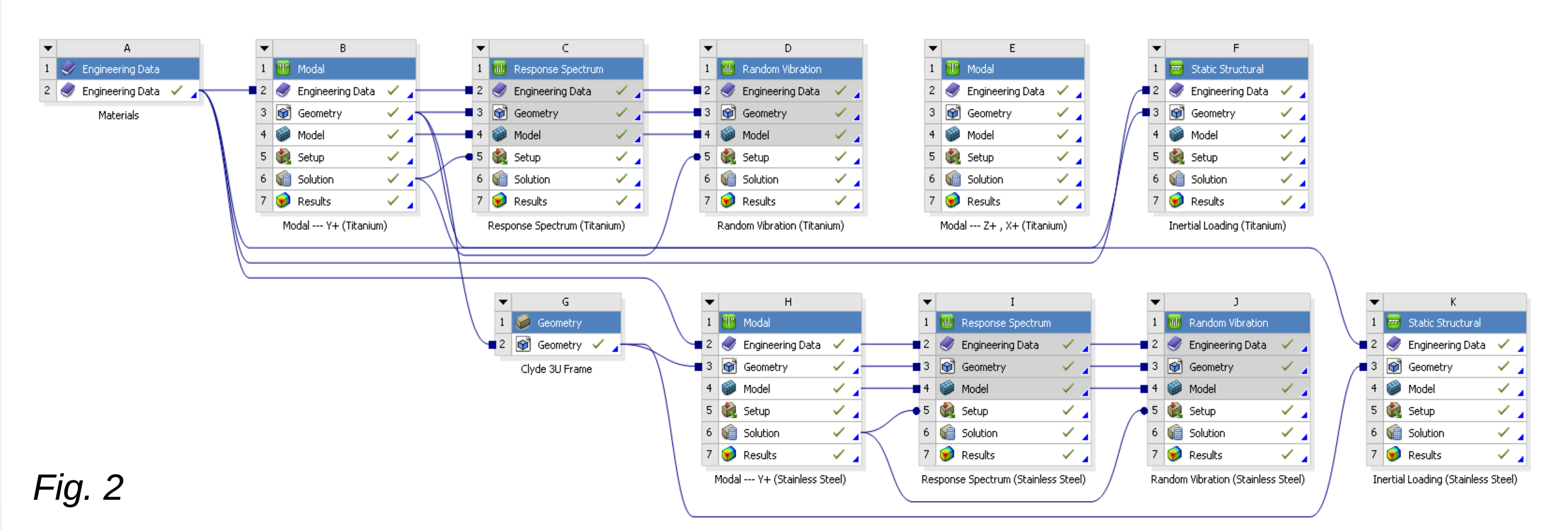
Fig. 2

Engineering data, namely component material information, is entered into the program. The working model is then meshed by ANSYS Workbench. The meshing process converts the working model into a finite element model which can be used for simulations. For each iteration of simulations, modal vibration tests were run first due to the dependency of the random vibration and response spectrum simulations on the results of the modal test. Inertial loading is tested for separately.

Upon completion, ANSYS outputs data spreadsheets for each simulation that are then compared to expected values and constraints. For the modal vibrations simulations, we tested for the first three modes of vibration for the structure. ANSYS will calculate the frequencies at which these modes occur and the predicted deformation at each one. Using this information, the frequency response simulation determines the maximum total deformation across a range of frequencies. In parallel to the response spectrum test, the random vibration uses the modal vibration data to determine the probable response based on excitation and dynamic reactions of the structure at its natural frequencies. Finally, the inertial loading simulation uses predetermined external forces to determine the deformation of the satellite during launch. All of these simulations provide information about maximum and minimum deformations as well as the components that experience max and min deformations. This information is compared to material properties for each component to determine the factors of safety (FoS) for the structure.

Data collected from each analysis is then compared to constraints and design needs indicated by NASA, the USAF, and the lab leadership. Changes are then made to the CAD model of each satellite, and the process repeats. Figure 3 shows the progression of working models used in analysis based on feedback from the cycle.

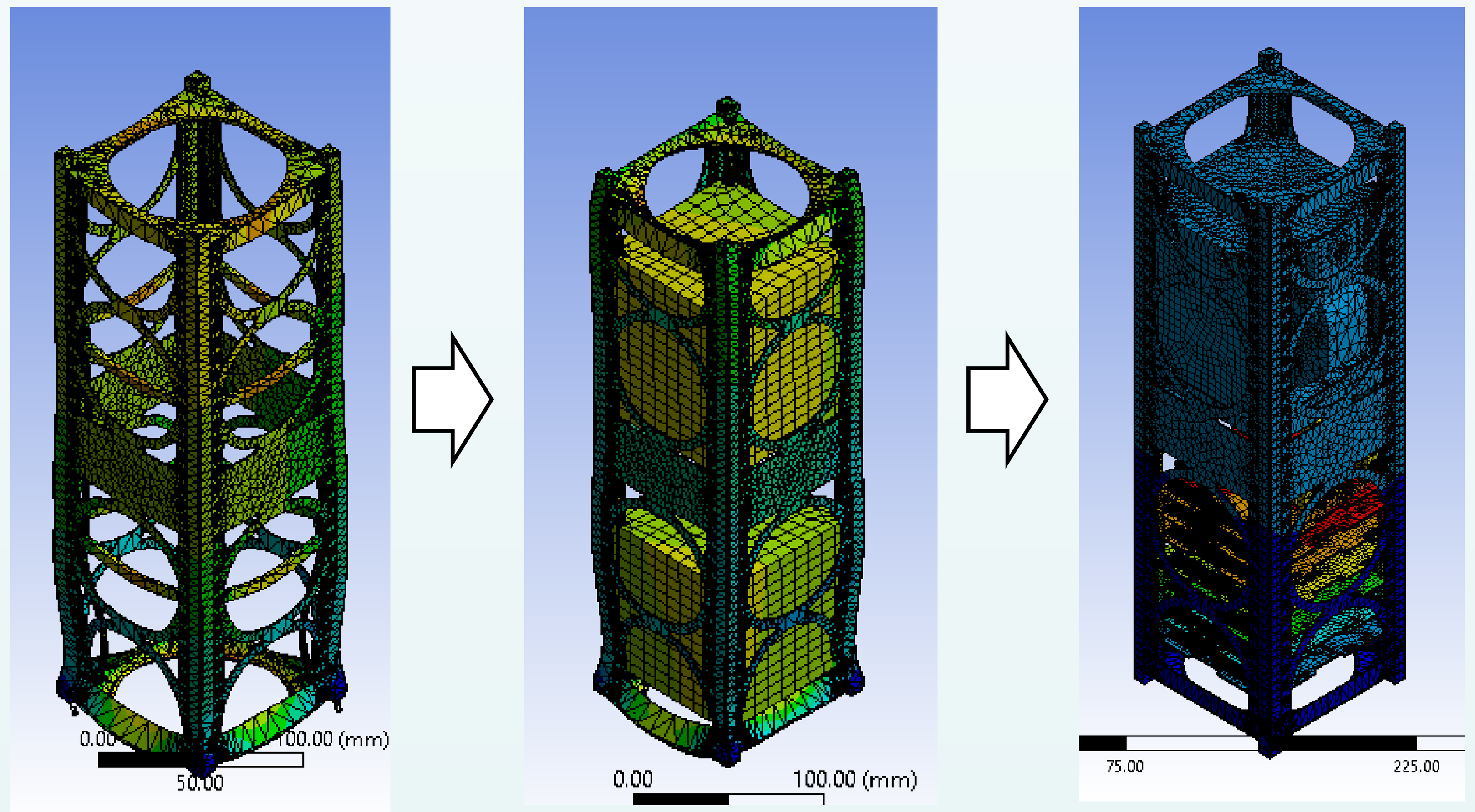
Fig. 3

The image on the right in Figure 3 represents a finite element analysis model for solely the structural frame used in both SPOC and MOCI. The center image is a mass-block model with two large rectangular cubes standing in for individual components. The left-most image shows a finite element model with many individual components represented despite being individually simplified.

Acquisition and Application of Results

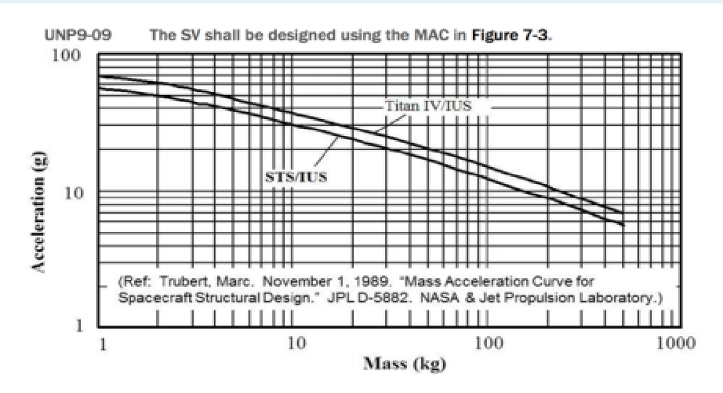

Fig. 4

Frequency	ASD Level	Slope
20	.026	Start
20-50	+6 dB/oct	+
50-800	0.16	Flat
800-2000	-6 dB/oct	-
2000	.026	Finish

Fig. 5

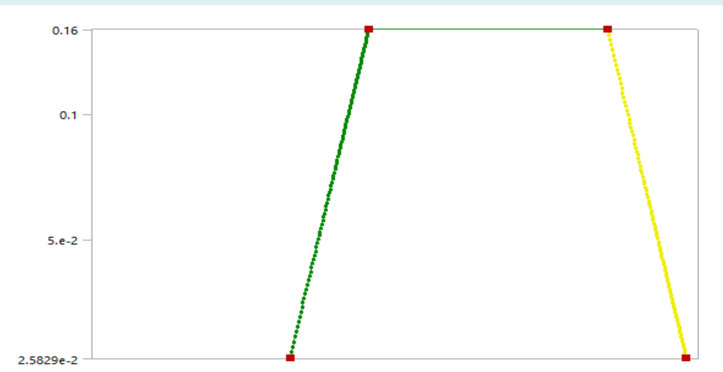
Fig. 6

To achieve accurate results from structural analysis, data inputs must also be accurate. Inertial loading simulations require acceleration information that is discerned from MATLAB code based on the Mass Acceleration Curve in Figure 4.

Random vibration tests needed an Acceleration Spectral Density curve. Figures 5 and 6 detail information on the curve used in the analysis of SPOC and MOCI. This curve data was provided by Clyde Space, the designer of the structural frame used in both satellites.

For the design to be valid, the first natural frequency (first mode) needs to be above 100 Hz. In addition, the FoS needs to be at least 2.0 to meet Air Force standards. Results were compared to these values when evaluating the results of analysis.

For more information about CubeSats, the Small Satellite Research Laboratory (SSRL), our missions, visit www.smallsat.uga.edu.