Structural Verification of the Rigidizable Inflatable
Get-Away-Special Experiment

Anna Gunn-Golkin, Sarah Helms, Richard Cobb
Air Force Institute of Technology, United States Air Force
2950 Hobson Way, Wright-Patterson AFB, OH, 45433-7765; (937) 785-3636 x4559
Anna.Gunn-Golkin@afit.edu, Sarah.Helms@hill.af.mil, Richard.Cobb@afit.edu

ABSTRACT: The Air Force Institute of Technology has developed the Rigidizable Inflatable Get-Away-Special Experiment (RIGEX) in order to advance development of inflatable, rigidizable space structures. RIGEX will test the deployment and structural characteristics of three thermoplastic composite tubes in the space environment. RIGEX is designed to fly in the Canister for All Payload Ejections container within the Space Shuttle Orbiter’s payload bay. This paper summarizes the science and motivation behind RIGEX’s inflatable rigidizable structure research. It then details the design and analysis of the RIGEX flight structure, which will house the inflatable rigidizable experiment. The paper details the development of a RIGEX finite element model. Results from this model are used to validate structural integrity of the experiment’s design and as a step towards meeting NASA natural frequency and load limit requirements. A dynamic modal analysis showed the first natural mode of the structure to be well above the 50 Hz minimum requirement. Then a static analysis was completed to show that the loads on all bolts were within factor of safety limits at maximum expected loads. The positive results from these analyses allow for the continued development and construction of the RIGEX experiment, scheduled for launch in 2007.

INTRODUCTION

Many of the latest space technology concepts involve the use of large space structures such as solar arrays, solar sails, large aperture antennas, and sunshields. Unfortunately, tight launch constraints on payload mass and volume preclude many of these concepts from ever getting off the ground. Mechanically deployed systems that can be packed into a stowed configuration for launch provide one solution to this problem. However, they often result in undesirably complex mechanisms. Alternatively, employing inflatable, rigidizable structures can reduce payload mass and volume and provide necessary packing flexibility without overcomplicating the system. However, lack of flight testing and deployment observation in space has been a large limiting factor for the application of such structures on operational payloads.

The Air Force Institute of Technology (AFIT) has developed the Rigidizable Inflatable Get-Away-Special Experiment (RIGEX) in order to advance development of inflatable, rigidizable space structures. This small experiment, designed to fly in the Space Shuttle Orbiter payload bay inside the Canister for all Payload Ejections (CAPE), will test the deployment and structural characteristics of three thermoplastic composite tubes in the space environment. Once on orbit, an oven will heat one tube, which becomes malleable above 125 degrees Celsius (°C). The tube will then be inflated, cooled (hardened), vented and excited using piezoelectric patches. This, along with a tri-axial accelerometer mounted at the free end of the cantilevered tube, will produce modal characterization data. The process will then be repeated, independently, for the other two thermoplastic composite tubes. Digital cameras will provide photographic documentation of the deployment process. The main objective of this research is to compare spaceflight results with deployment data gathered in the AFIT lab. This correlation will contribute towards validating on-orbit reliability and ground testing techniques of such inflatable rigidizable structures for application on future operational programs.

Payload Motivation

Space exploration has always been an expensive endeavor with many restrictions and limitations. Such limits are often dependant on the size of the launch vehicle fairing and the mass the launch vehicle can lift to the prescribed orbit. Inflatable structures have the potential to reduce spacecraft mass, physical dimensions, and in doing so, construction and launch costs. Over the last several decades, inflatable structure concepts have been developed and tested, producing enough data to show their potential to provide a low cost, low weight alternative to conventional space hardware, with high mechanical packing efficiency and deployment reliability. The term inflatable structure...
indicates that a condensed configuration will be launched into space and then deployed by pressurization to its full intended form. This pressure must remain within the structure in order to keep it in a rigid, structurally stiff state. As documented by the Jet Propulsion Laboratory (JPL) at the California Institute of Technology, small leaks caused by material imperfections or damage by micro-meteoroids are unavoidable. These leaks make sufficient back-up inflation gas a necessity for long term success. This addition can be very costly in terms of volume, weight, and expense due to added or enlarged pressure system components, which can negate and even overrun any initial prospect of savings from the use of inflatable technology. Therefore, with the growing maturity of inflatable space structure technology, space rigidization is of great interest. Rigidization of an inflatable structure is a process whereby, following deployment by inflation, the structure is physically rigidized to the point where it will maintain its intended shape without reliance on pressurization.

Due to their potential benefits inflatable, rigidizable structures are very intriguing for a variety of space applications. These structures, most with relatively high strength and stiffness, can provide “enhancements in the performance characteristics of many space deployable systems such as large antennas, solar arrays, and sunshields” due to small volume and mass as compared to conventional constructions. Inflatable, rigidizables can also be applied to large aperture sensorcraft, deployable booms, solar sails, and countless other large ultra-lightweight technologies yet to come into fruition.

While this innovative technology sounds very practical, the actual value of inflatable, rigidizable structures must be substantiated by research and successful on-orbit testing before use on operational satellites. While multiple inflatable structure experiments have been proven in space, and an aluminum laminate inflatable, rigidizable material has been flown in space as a structural component, all other inflatable, rigidizable structure technologies have only been tested and deployed on the ground, in thermal vacuum chambers or on air tables. Spaceflight heritage of a proposed technology is a significant risk mitigation method. In order to make an inflatable, rigidizable material marketable for satellite application, steps to prove its functional capability and reliability must be made. RIGEX was designed to address the development of inflatable, rigidizable structure technology by progressing three main developmental issues: concept maturity, technology database, and the capability for analytical performance simulation.

**RIGEX Overview**

The Rigidizable Inflatable Get-Away-Special Experiment is a preliminary step in employing large-scale inflatable, rigidizable structures in space applications. RIGEX is a Space Shuttle Orbiter Payload Bay container experiment that was originally slated to mount inside of a NASA Get-Away-Special (GAS) canister. NASA’s plan to discontinue use of the GAS canister on future Space Shuttle flights led to modification of RIGEX in 2004. The experiment is now revised to mount inside the Canister for all Payload Ejections (CAPE) container as shown in Figure 1.

![Figure 1. CAPE / RIGEX Configuration](image)

CAPE was developed by Muniz Engineering, Inc. in conjunction with the Department of Defense (DoD) Space Test Program (STP) in response to the need for “a single ejection platform capable of ejecting payloads with requirements that are not compatible with current NASA developed ejection systems.” CAPE makes use of the previous GAS Beam mounting plate in order to attach the whole payload assembly onto the orbiter bay sidewall. The CAPE ejection capability will not be exploited for RIGEX.

The goal of RIGEX is to take three 20 inch long inflatable, rigidizable tubes through their full deployment process and then test their modal characteristics on orbit. The tubes were designed and manufactured by L’Garde, Inc. located in Tustin, California and are characterized by a glass transition temperature ($T_g$). Below the specified $T_g$ the tubes are structurally stiff, but above the $T_g$, tube material becomes malleable. Hence, this rigidization method is also known as Sub-$T_g$. The tubes used in RIGEX have a $T_g$ of 125°C and are made of a proprietary composite material consisting of Kevlar fibers with a
polyurethane-based resin. A layer of Kapton tape has been applied to both the inside and outside of the composite tube construction. Two piezoelectric patches are mounted opposite each other at the base of each tube to serve as an input vibration source for modal characterization.

Each of the three tubes will be fixed to the RIGEX main structure at one end, while the opposing end will be left free to form a cantilevered configuration. The tubes have been folded into a stowed configuration by L’Garde using a z-fold design. Comparison of the stowed configuration to the deployed configuration is shown in Figure 2. The stowed configuration tube will be mounted to the RIGEX main structure inside a small oven. This oven will provide enough heat to transition the tube beyond its $T_g$ preparing the tube for inflation.

The whole deployment process is then repeated (heating, inflation, cooling, venting, and excitation) for the next tube, until all three tubes have been deployed. As each tube is deployed, the RIGEX subsystems will collect data on pressurization and temperature levels as well as a series of digital photographs for further documentation. RIGEX must then be returned to AFIT for analysis of the collected data because there will be no telemetry sent from the experiment while in orbit. All of the data gathered during experiment execution is stored internally on the PC-104 computer. Once returned, this data will be compared to similar modal characterization analysis data obtained in the laboratory. Ground test data currently exists for a set of Sub-Tg tubes of the same make and configuration as those used for flight. Comparison of space versus ground test data will aid in determining the accuracy of laboratory simulation of structural performance. These analyses will provide a complete evaluation of the performance and reliability of the L’Garde Sub-Tg tubes.

The RIGEX detailed design is shown in Figure 3. An outer shroud will encapsulate the entire experiment (shown as transparent in Figure 3). This shroud will protect the CAPE from any RIGEX components that might come loose if the Orbiter should experience an over-g. There are four bays in the RIGEX structure, separated by four ribs. Three of these bays contain identical hardware: an inflatable rigidizable tube, an oven, a camera, a pin puller to release the oven’s latching mechanism, and appropriate instrumentation. The fourth bay contains the PC-104 flight computer and the power distribution plate, which routes power and command inputs throughout the experiment and provides feedback on the experiment’s progress to the astronauts. The rectangular area in between the four bays houses the three pressure vessels and various pressure system components which will be used to inflate the experiment tubes once on orbit.

There is a five step sequence of events for each Sub-Tg tube during the execution of RIGEX. First, a tube is heated to over 125°C within its oven. Then, the latch over the oven door is released and the tube is pressurized with nitrogen gas causing inflation. After inflation, the tube remains pressurized until its material temperature drops below the $T_g$ and the tube stiffens. This is the rigidization step. The nitrogen is then vented out of the tube. Finally, the tube is excited by the two piezoelectric patches while a triaxial accelerometer mounted at the cantilevered end collects data to be used for modal characterization.

**Figure 2. Inflatable, Rigidizable Sub-Tg Tubes**

![Stowed Tube before Inflation](image1.png)  ![Deployed Tube](image2.png)
Analysis of the structural design of the RIGEX payload was completed to fulfill NASA requirements for launch. This involved first developing a RIGEX structural model. Finite element modeling techniques were used to build the model in NX Nastran for Finite Element Modeling and Post-Processing (FEMAP) Version 9.0 software. Then, the RIGEX finite element model (FEM) was subjected to eigenvalue analysis to solve for its first three natural frequencies. Finally, a set of static analyses were executed in order to assess the maximum internal loads seen by the structural bolts. All three steps (developing a RIGEX FEM, modal analysis, and bolt analysis via static loading) were completed to validate integrity of the RIGEX structural design before construction could begin. The following sections detail the processes taken and results obtained from each set of analysis.

Development of the RIGEX structural model provides an analytical means for strength of material verification of the RIGEX design. Structural analysis, in the form of analytical calculations paired with physical hardware testing, is mandated by NASA and STP to ensure that RIGEX is structurally compatible with the National Space Transportation System (NSTS). Additionally, this paired analysis proves that the combined CAPE/RIGEX payload will meet all of its mission objectives when subjected to NSTS flight loading conditions.

The RIGEX FEM, or structural model, is used to show that the structure will exceed the minimum natural frequency parameter as determined in the CAPE Hardware Users Guide. The model also provides for the analytical portion of the structural strength verification. Structural strength is assessed by applying 64 unique possible load limit combinations to the model and ensuring that the loads transferred through fasteners do not exceed their allowable limits plus a factor of safety. Static analysis is also used to ensure that internal stresses do not exceed limits for structural materials and fasteners. The internal stresses, as produced by the FEM, are also analyzed to insure they do not exceed material factors of safety as required by NASA.

Methodology

The RIGEX FEM is designed to create a virtual model that will accurately represent the static and dynamic behavior of the true flight structure. The FEM is a three-dimensional deformable model of the primary RIGEX structure. In its simplest form, RIGEX is an assembly of aluminum plates combined into a complex cylindrical geometry with supporting ribs. The Finite Element Analysis (FEA) method is used to break down complex geometries into small elements which can then be assigned a simple spatial variation. FEA is an approximation tool which provides “piecewise interpolation of a field quantity” that can then be reassembled in order to draw big picture conclusions.

With FEA, small groups of elements with simple applied loads and boundary conditions can be solved by hand. However, the computational complexity quickly escalates with added elements, complex geometries, compound loads, and refined boundary conditions. Therefore, FEA software programs are used. FEMAP is a commercial finite element modeling and post-processing software analysis program that allows development of stress, temperature and dynamic performance analysis. For the RIGEX structural model, NX Nastran for FEMAP was utilized to construct a representative model, dynamically solve for natural frequencies and mode shapes, and statically solve for internal loads.

Natural frequencies are the frequency values at which a structure, when subjected to sinusoidal excitation, is...
inclined to react. The mode shapes associated with each natural frequency define the deformed appearance of the structure as it is reacting, or vibrating.\(^{10}\) Computing a structure’s natural frequencies and mode shapes involves a process called normal modes analysis, otherwise known as eigenvalue analysis. In this method, eigenvalues signify the natural frequencies and eigenvectors characterize the mode shapes. FEMAP solves for the undamped free vibrations of a structure using the following equation:

\[
[[K]] - \lambda_i [[M]] \cdot \{\phi_i\} = 0
\]  

(1)

Where \([K]\) is the structure’s stiffness matrix and \([M]\) is the structure’s mass matrix. These matrices are determined by the geometry and properties applied in the structural model. The other two variables, \(\lambda_i\) and \(\phi_i\), are eigenvalues and corresponding eigenvectors that are computed in the FEMAP software. From the resulting eigenvalues, the natural frequency values can be computed using the relationship \(f = \sqrt{\lambda_i}\) where \(f\) is the frequency in radians per second. While there are many different methods which can be employed for eigenvalue analysis, the Lanczos method was chosen to solve for the RIGEX FEM natural frequencies and mode shapes. This method provides robust results with a relatively small amount of required memory and fast calculation times.\(^{10}\) “Normal modes analysis forms the foundation for a thorough understanding of the dynamic characteristics of the structure.”\(^{10}\) This is due to the fact that eigenvalue analysis results are useful for a variety of applications and, most importantly, provide a straightforward baseline assessment of model confidence. If the natural frequencies and mode shapes presented in a FEM correlate well with physical test data, the FEM can then be used for other, more complicated analyses with a high confidence of obtaining realistic results.

Building a high fidelity model with appropriate element, meshing, and constraint choices is key in obtaining meaningful results. Therefore, a method of testing various models and their correlations to a physical specimen similar to the RIGEX structure was carried out. The modeling method that most closely resembles the test specimen results will be used to build the RIGEX FEM. This FEA approach validation will be accomplished using an available RIGEX engineering model structure as the test specimen and a set of preliminary FEMs as the computer models. By correlating a preliminary FEM with lab test data, the FEA modeling method is validated.

The step-by-step process for developing a finite element model starts with creating the geometry. In FEMAP this process involves employing an extensive set of options, such as shape, line, extrusion and solid tools, to draw the intended structure. Then, once each shape is drawn correctly, it is labeled as a boundary surface. Once each closed shape is identified as a boundary surface, the set of surfaces can be transformed from geometry into a set of finite elements. This transition is made by meshing each surface with a set of nodes and elements. Each element consists of a connected group of nodes with assigned material properties. Material property values are then applied to each newly meshed surface. These property values are standard for the various construction materials. Once it is ensured the properties are accurate and the mesh is correctly aligned, analysis can begin. Many different analysis sets can be defined and saved in FEMAP, each with their own selected load and constraint sets. A load set defines the amplitude, direction, and location of loads being applied to the structure. These include point loads, body loads, and radial accelerations. A constraint set tells the solver what type of boundary conditions to apply to the selected nodes. Nodes can be constrained in any combination of their six degrees of freedom. Once an analysis set is defined, the FEMAP software passes along all model and analysis set information to the NX Nastran solver. The solver then sends the computed results of a successful run back into the FEMAP interface for viewing. NX Nastran also saves a data output sheet for each analysis run which can be investigated numerically at a later time. FEMAP can provide many options for visual assessment, including deformation and contours based on stress, strain, translation, rotation, and forces calculated for each node. Finally, the results can be interpreted for structural analysis purposes and model analysis accuracy.

For the RIGEX FEM normal mode analysis, a constraint set of fixed nodes at the CAPE Mounting Plate/RIGEX bolt pattern interface was used. Applied loads sets are not used for normal modes analysis; therefore, any stress or strain values developed from the analysis are strictly intended for differential, not quantitative, identification. The structural design goal is for the integrated CAPE/RIGEX payload to have a first natural frequency above 50 Hz.\(^{6}\) The root of this requirement comes from the NSTS 21000-IDD-SML document which states that all sidewall mounted payloads with a natural frequency less than 35 Hz must complete coupled loads analysis for all stages of flight.\(^{11}\) However, if the payload has a first natural frequency of greater than 35 Hz, with respect to the adapter interface, a table of limit load factors from
NSTS 21000-IDD-SML can be used rather than having to complete coupled loads analysis. Therefore, STP requires that all of its payloads have a first natural frequency greater than 50 Hz in order to meet the 35 Hz cutoff for the integrated structure.

**RIGEX FEM Classification and Assumptions**

The first step in developing a structural model is to understand the nature of the problem, otherwise known as problem classification. Background information on the specific problem delineates how to model, discretize, and analyze a structure. In addition, any simplifying assumptions made during the modeling process must be recorded for future reference. Finite element analysis is a simulation tool, not reality. Therefore, a mathematical model with generalized equations and assumed conditions will give meaningful results only when paired with all the background information used to develop the model.

The RIGEX structure is designed for spaceflight aboard the Space Shuttle Orbiter. The primary structure is built from aluminum plates of various shapes and thicknesses formed into a generally cylindrical form. While there are many complex subsystems within the structure, a detailed analysis of these items is not necessary because they are space qualified separately. However, the mass they add to the structure does need to be taken into account. They are included as point masses, which generally reduce structural stiffness and increase applied loading. Nonlinearities from material properties are not allowed in the analysis, as yielding would be considered structural failure in this context. Natural frequency data and internal load values are the two primary results sought from the RIGEX FEM analysis.

As RIGEX is a complex structure, simplifying assumptions are made in order to create a model that is computationally feasible. The following items will used as baseline assumptions for the development of the RIGEX FEM.

1. The RIGEX primary structure is constructed out of 6061-T651 plate aluminum. In FEMAP this material was assumed to be isotropic and homogeneous. In reality, there will be material imperfections present in any metal, but prediction of such imperfections is impractical. The properties for this material were loaded from the FEMAP material library and accuracy was ensured from the values for given in MIL-HDBK-5H.

2. Various small holes for venting and wire routing are present in the RIGEX structure. These holes were considered to be a level of detail not required for FEM analysis and, therefore, were left out of the model. All of the surfaces in the RIGEX FEM are meshed as solid pieces with no cutouts present. The holes present in the flight model structure will slightly reduce its mass and decrease its stiffness.

3. A variety of subsystem components are housed within the RIGEX structure. The mass of these items must be addressed for an accurate FEM, however, analysis of the subsystem components themselves is not necessary. Therefore, these items were placed into the model as point masses at their connection locations. This treatment of the subsystem components makes inclusion of their added mass possible without any added stiffness to the structure. This is a conservative approach because, in fact, each component attached to the structure will add some small amount of stiffness to that area, as well as mass. Table 1 lists all of the RIGEX subsystem components that were included in the FEM along with their mass and node number(s) where the point mass was placed.

<table>
<thead>
<tr>
<th>Component</th>
<th>Quantity</th>
<th>Mass (per unit) (lb)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Power Relay</td>
<td>3</td>
<td>0.22</td>
</tr>
<tr>
<td>Transformer</td>
<td>3</td>
<td>0.34</td>
</tr>
<tr>
<td>Press. Cyl. Mount</td>
<td>3</td>
<td>0.83</td>
</tr>
<tr>
<td>Oven Bracket</td>
<td>3</td>
<td>1.60</td>
</tr>
<tr>
<td>Oven</td>
<td>3</td>
<td>2.75</td>
</tr>
<tr>
<td>Computer</td>
<td>1</td>
<td>13.10</td>
</tr>
<tr>
<td>Camera</td>
<td>3</td>
<td>0.62</td>
</tr>
<tr>
<td>Power Distribution Plate</td>
<td>1</td>
<td>4.80</td>
</tr>
</tbody>
</table>

5. Nodes were individually created at the location of each bolt hole in order to represent accurate interaction between the plates. The bolt node was then tied to both adjoining surfaces. The detail of individual bolts and their respective bolt holes was not modeled. The loads transferred at each of these bolt locations can be used to derive the interaction of the actual bolt with the structure. Thus, a node at each of the 204 structural bolt locations was considered to be sufficient detail for representation of the union of plates.
6. The CAPE Mounting Plate provides a mechanical interface between RIGEX and the CAPE canister. This plate is a 1.5" thick aluminum 6061-T651 circular disk with a bolt pattern matching CAPE around the edge of its radius. The RIGEX Top Plate, 0.625" thick, is then attached to the CAPE Mounting Plate via another circular bolt pattern. The CAPE Mounting Plate was not included in the RIGEX FEM as it is considered a fully constrained surface. Therefore, the RIGEX Top Plate to CAPE Mounting Plate bolt pattern was assumed to be a set of fixed nodes. The term fixed refers to a prohibition of translation and rotation of the given node. Therefore, the RIGEX FEM was conservatively constrained in a fashion that allowed deformation of the Top Plate into the region where the CAPE Mounting Plate will be physically present. An analysis with Z axis translation constrained along the entire contact surface of the Top Plate was performed and found to be too much of a restriction, producing unrealistically stiff results. No solution was found that could restrict translation into the CAPE Mounting Plate while still allowing deformation away from the Mounting Plate. Therefore, the set of 28 fixed nodes representing the RIGEX Top Plate to CAPE Mounting Plate bolt pattern was implemented as the applied FEM constraint set. This constraint set is considered conservative as it only reduces stiffness and increases loads transferred along the constraints.

**FEM Method Validation Analysis**

Finite element modeling has evolved into a multifaceted discipline. The theory has grown rapidly since the 1950s. Many commercial software packages exist and many modeling options are available. An assortment of element geometries and sizes, in two dimensional or solid three dimensional configurations, can be obtained. The shape functions assigned to a set of elements can be linear or quadratic, among other higher-order options, and the degrees of freedom which govern the spatial variation of a field can be added or taken away. Therefore, after problem classification is completed, an appropriate method for model development must be selected. The results produced by a model are largely dependant on its method of development; poor modeling choices will lead to fallacious results. A series of comparisons between laboratory test results and finite element model examples was completed in order to determine an acceptable modeling method for the RIGEX FEM.

An engineering model (EM) of the RIGEX structure was fabricated by previous students and available for use in the AFIT laboratory. This structure, shown in Figure 4, represents the design of RIGEX before it was modified for the CAPE container.

Many changes have been made to the RIGEX structural design since the EM was built, including implementation of thicker aluminum plates, a larger cylinder radius, and a containment shroud. While the physical dimensions have been modified, the general design of the structure remains the same. The most recent RIGEX structural design is defined by detailed SolidWorks drawings, but has yet to be fabricated. The RIGEX structure is asymmetric and complex. It cannot be accurately represented by a simple cantilevered beam or cylinder because of the unique arrangement of structural rib plates. Therefore, the EM was used as a preliminary structure for validation of FEM methods because of its availability and similarity to the current design.

First, laboratory tests were completed to obtain natural frequency and basic mode shape data from the physical EM structure. Then, a series of FEMs representing the EM structure were built in FEMAP. These models are referred to as a set of preliminary FEMs, representative of the EM structure, as opposed to the RIGEX FEM which denotes the final FEM created and is indicative of the current RIGEX structural design. The main goal was to create a FEMAP model whose resultant eigenvalue analysis agreed with the natural frequency data found in the lab. Once found, the preliminary FEM method which best correlated with lab test data would be identified as the best method to use for the
RIGEX FEM. By determining methods of FEM construction that will accurately represent the natural modes of a physical test article, validation of modeling methods and FEM analysis for this specific problem is obtained. With confidence that the RIGEX FEM accurately represents how the structure will respond to external stimuli, the FEM can then be used to obtain natural frequency values and strength design validation.

Two main models were built to represent the EM structure. The first model was created by forming solid rectangular and circular geometries with the correct dimensions and positions. Then this 3-D structure was meshed with solid parabolic elements. These elements were created using the FEMAP auto mesh function. Each element was a solid tetrahedral containing ten nodes. The second model was formed by placing 2-D shapes, without a visible thickness, in the proper positions to form the 3-D structure. Then, the same FEMAP auto mesh function was used to create a plate element mesh within each surface. The plate elements were four-noded quadrilaterals with some three-noded triangles present in areas with curved geometry.

Each element has a linear or parabolic option. The linear versions have nodes only at the corners of the element shapes and are characterized by a linear displacement field equation. The parabolic versions have an extra set of mid-side nodes spaced in-between each corner node. This allows the displacement field equations for these elements to contain a complete quadratic function.

Solid elements are known to have problems with locking when used in thin plates. Locking causes a model to exhibit high-stiffness behavior which, in turn, influences the analysis solutions. Convergence on an accurate answer can be obtained with refinement of the mesh. Still, this is an unattractive solution because of the significantly longer computation time (on the order of hours instead of minutes) taken to solve for densely populated 3-D meshes. The method of 2-D plate elements has the ability to produce accurate results with fewer elements and a much smaller computation time. The 2-D plate element method, however, will diverge from reality as the thickness of the plate being modeled increases. Since the thickest plate being modeled in the FEM is only 5/8" thick, the plate elements still exhibit relevant solutions. In the end, four preliminary FEMs were developed to represent the RIGEX engineering model structure. These models included one plate version, and coarse mesh, intermediate mesh, and fine mesh solid versions. An eigenvalue analysis of these four FEMs was completed with fixed node boundary conditions mimicking the circular bolt pattern of the EM structure in the lab. The first and second mode results from the plate model preliminary FEM are shown in Figure 5. Each figure includes a color scale indicating the relative Von Mises stress distribution throughout the structure. The stress and displacement values shown are insignificant because eigenvalue analysis results are arbitrarily scaled in FEMAP for visual clarity. However, the mode shape and location were maximum stresses appear gives insight as to where the structure will be most harshly burdened during launch environment loading. The results varied widely between the four different preliminary FEMs.

To assess which of the preliminary results correlate well with the physical structure, baseline natural frequency values of the EM structure were obtained through ping testing and a 2-D laser vibrometer scan of the structure. SignalCalc software was used to gather data on the first two structure modes with a resolution of 1600 lines over a frequency span of 0-312 Hz. Three different accelerometers were placed at various positions to measure frequency response data from each hit of the ping hammer. Data was recorded when triggered by the ping hammer input voltage and 10 averages were used to produce the result graphs shown in Figure 6 (first bending mode) and 7 (second bending mode). These graphs also include coherence plots to assess the validity of the results over the frequency span. A 2-D scanning laser vibrometer was arranged to record modal data of the structure.

Figure 5. Preliminary FEM Plate Model – First and Second Mode Results

- Mode 1: 132.7164 Hz
- Mode 2: 185.0829 Hz
Validation Model Results and Conclusions

The set of data gathered in the lab, combined with eigenvalue analysis of the preliminary FEMs, provided an overall assessment of the EM structure. Tables 2 and 3 include a compilation of natural frequency results from all data sources mentioned.

**Table 2. Compilation of EM Structure Results for FEM Method Validation (Modes 1)**

<table>
<thead>
<tr>
<th>Model Description</th>
<th>Mode 1 (Hz)</th>
<th>% Difference From Ping Test</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ping Test</td>
<td>132</td>
<td></td>
</tr>
<tr>
<td>Laser Vibrometer Scan</td>
<td>131.5</td>
<td>0.38%</td>
</tr>
<tr>
<td>FEMAP 2-D Linear Plate Model</td>
<td>132.7</td>
<td>0.53%</td>
</tr>
<tr>
<td>FEMAP 3-D/Solid Quadratic Coarse Mesh</td>
<td>250.6</td>
<td>89.85%</td>
</tr>
<tr>
<td>FEMAP 3-D/Solid Quadratic Intermediate Mesh</td>
<td>159</td>
<td>20.45%</td>
</tr>
</tbody>
</table>

**Table 3. Compilation of EM Structure Results for FEM Method Validation (Modes 2)**

<table>
<thead>
<tr>
<th>Model Description</th>
<th>Mode 2 (Hz)</th>
<th>% Difference From Ping Test</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ping Test</td>
<td>170.1</td>
<td></td>
</tr>
<tr>
<td>Laser Vibrometer Scan</td>
<td>170.5</td>
<td>0.24%</td>
</tr>
<tr>
<td>FEMAP 2-D Linear Plate Model</td>
<td>185.1</td>
<td>8.82%</td>
</tr>
<tr>
<td>FEMAP 3-D/Solid Quadratic Coarse Mesh</td>
<td>297.2</td>
<td>74.72%</td>
</tr>
<tr>
<td>FEMAP 3-D/Solid Quadratic Intermediate Mesh</td>
<td>199.4</td>
<td>17.23%</td>
</tr>
<tr>
<td>FEMAP 3-D/Solid Quadratic Fine Mesh</td>
<td>142.4</td>
<td>16.28%</td>
</tr>
</tbody>
</table>

The ping test gave very clear, concise results and, therefore, was used as a baseline truth comparison for all other data in the percent difference column. The solid meshes exhibited locking behavior, as expected, and improved with mesh refinement. However, the analysis of the 3-D/Solid Quadratic Fine Mesh FEM took an unreasonable time to complete. The plate model analysis ran very quickly and resulted in the most accurate natural frequencies.

The final conclusion drawn from this series of method validation trials was to employ a plate FEM for all subsequent design work. The plate elements had many advantages over solids including fewer elements and nodes, shorter computation times, and results which more closely matched the test data. Therefore, the plate model approach to finite element modeling and analysis is used, with confidence, in development of a final RIGEX FEM.

**RIGEX FEM Design**

Once confidence was gained in the FEM development method, the final RIGEX structural model was created. This endeavor began with formation of the proper geometry in FEMAP. The RIGEX structure is composed of ten aluminum plates: a top plate, two inflation system mounting plates, four ribs, an oven
mounting plate, bottom square plate and a shroud. Each shape was drawn separately according to the dimensions specified in the RIGEX drawing package as developed in SolidWorks. A point was placed at each bolt location to aid in bolt node selection once meshed. Points were also placed at the center of mass positions for each subsystem component to use later as point mass locators. After all boundary surfaces and points were placed, three custom meshing options were designated. First, a meshing attribute was chosen to define the material property and plate thickness of each surface. The rib plates, pressure system plates and square bottom plate are all made of 0.375” thick 6061-T651 aluminum. The top plate and oven mounting plate were both made of a thicker 0.625” 6061-T651 aluminum. Then, a custom mesh pattern was specified along each surface edge. This allows for very fine meshes along rib connections and other bolt locations for higher fidelity results while leaving a coarse mesh flat, relatively unstressed surfaces to save in computation time. Finally, the mesh points on surface command was utilized to guarantee that nodes would coincide with all bolt and point mass locations within the outline of each surface. The custom mesh pattern on adjoining surfaces was set to match to ensure no discontinuities would arise, especially on the shroud. With these three specialized options applied uniquely to each plate surface, the geometry was ready to be transformed into a finite element model.

Each surface was meshed separately in order to form the RIGEX FEM. After a surface was meshed, the nodes at the bolt location indicator points were recorded. An intersecting plate would then be meshed, and a FEMAP check for coincident nodes was run to identify any nodes that were within a given distance from each other. This would produce an all-inclusive list, which would then be queried for nodes specifically representing bolt locations. The coincident nodes at bolt locations were then manually merged. This allowed for the plates to be connected only at the precise location of the actual bolts. When possible, the model was meshed with linear Quad elements. However, linear Tri elements populate the entire top plate and oven mounting plate to create a better fit due to their circular geometries. A unique set of three tube elements was also included in the RIGEX FEM. These elements represent the three inflation system pressure cylinders, sandwiched between the two structural inflation system plates. Insertion of the tube elements represented reality by tying together the two physically separated inflation system plates. The last basic item included in the RIGEX FEM was a boundary condition of nodal constraints. A cutaway view of the RIGEX FEM is shown in Figure 8 to show the pressure cylinder tube elements, the constraints, and the bolt pattern.

A property card for each subsystem component was formed with the correct mass included. Then a mass element was applied to each nodes corresponding to a subsystem location. All subsystem components were represented by a single point mass, except the computer. The computer is the heaviest component and is mounted on a mounting bracket at two locations nearly 14 inches from each other. Thus, the computer was modeled as two point masses. This allows for more accurate bolt loading analysis and only decreases the stiffness, thus increasing the conservative nature of the eigenvalue analysis. The shroud was created by extrusion, from the Top Plate to the Oven Mounting Plate, of two semicircular curves. The curves were then joined on their edges to create a cylinder. Nodes representing bolts along the shroud were then merged with their coincident nodes along the ribs and around the Top Plate and Oven Mounting Plate and the same fashion used to join the plate elements.

Results and Analysis

The results from the RIGEX FEM, were as expected. As development of the FEM progressed, eigenvalue analysis revealed that the shroud added stiffness to the structure, thus increasing its natural frequency, while adding point masses decreased the stiffness and lowered the natural frequency results. In addition, the shroud greatly dispersed the loads experience by
individual bolts. However, the loads experienced at some point mass locations drove design changes requiring larger, stronger bolts. Also, as expected, the RIGEX FEM presented higher natural frequencies than the EM structure. The first mode of the fully massed structure with shroud included came out to be 185.3 Hz. This is well above the 50 Hz first mode limit set by STP. Figures 9 thru 11 shows the first three expected mode shapes and values for the RIGEX structure.

Sixty-four different static loading cases were also analyzed. These sixty-four loads represent the maximum values from Table 3, which is a compilation of loads seen by side-wall mounted payloads in the orbiter during the various phases of flight.\textsuperscript{8} These maximum static loads are then used in determining the structural integrity of RIGEX’s bolts and ensuring the structure will not translate or deform in a manner that will negatively impact the experiment or any surrounding hardware.

<table>
<thead>
<tr>
<th>Flight Event</th>
<th>Load Factor g</th>
<th>Angular Acceleration Rad/s\textsuperscript{2}</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Nx</td>
<td>Ny</td>
</tr>
<tr>
<td>LIFT-OFF</td>
<td>± 7</td>
<td>± 7</td>
</tr>
<tr>
<td>Low Freq. Vibration</td>
<td>± 5.4</td>
<td>± 8</td>
</tr>
<tr>
<td>1</td>
<td>± 8.8</td>
<td>± 7</td>
</tr>
<tr>
<td>2</td>
<td>± 7</td>
<td>± 10.6</td>
</tr>
<tr>
<td>3</td>
<td>± 7</td>
<td>± 7</td>
</tr>
<tr>
<td>LANDING</td>
<td>± 6</td>
<td>± 7</td>
</tr>
</tbody>
</table>

Once identified, the 64 different maximum load combinations were assigned to a multiset analysis in FEMAP. The multiset analysis outputs the loads and translations experienced in the x, y and z directions at every node location for each of the 64 load combinations. As the FEM has 6252 nodes, this output file has 2,400,768 data points. Manually retrieving the relevant data would difficult at best, thus Matlab based algorithms were developed to aid in data reduction.

The first algorithm pulled the load values at each of the 204 bolt node locations. It then appropriately rotated the x, y, and z axes to reflect the actual orientation of the bolt in order to gain axial and shear load data. The Matlab code then sorted the data into 11 different bolt patterns. These included: constraint, bolts with their primary axis aligned with the global ‘x’, bolts with their primary axis aligned with the global ‘y’, bolts with their primary axis aligned with the global ‘z’, and seven ‘shroud’ bolt patterns, rotated 173º, 198.71º, 224.42º, 70.16º, 95.87º, 121.58º, and 147.29º about the z axis with respect to the global coordinate system. These 11 bolt patterns encompass all 204 bolts, and also ensure that the bolts of each pattern are the same size and material in the physical structure. The load data on the nodes carrying the larger point masses was also retrieved. Only the maximum axial and shear load of
each bolt pattern or type of point mass (oven, camera, etc.) is outputted from Matlab. This leads to only 32 data points (Table 4), which are further analyzed per the bolt analysis discussion below.

<table>
<thead>
<tr>
<th>Location</th>
<th>Value</th>
<th>Load (lbs)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Constraint Bolts</td>
<td>Max Axial</td>
<td>1341.4</td>
</tr>
<tr>
<td></td>
<td>Max Shear</td>
<td>5128.7</td>
</tr>
<tr>
<td>&quot;Z-axis axial&quot; bolts</td>
<td>Max Axial</td>
<td>3.9</td>
</tr>
<tr>
<td></td>
<td>Max Shear</td>
<td>140.1</td>
</tr>
<tr>
<td>&quot;Y-axis axial&quot; bolts</td>
<td>Max Axial</td>
<td>16.5</td>
</tr>
<tr>
<td></td>
<td>Max Shear</td>
<td>228.8</td>
</tr>
<tr>
<td>&quot;X-axis axial&quot; bolts</td>
<td>Max Axial</td>
<td>8.8</td>
</tr>
<tr>
<td></td>
<td>Max Shear</td>
<td>288.7</td>
</tr>
<tr>
<td>Shroud Coord 1 Bolts</td>
<td>Max Axial</td>
<td>4.8</td>
</tr>
<tr>
<td></td>
<td>Max Shear</td>
<td>52.2</td>
</tr>
<tr>
<td>Shroud Coord 2 Bolts</td>
<td>Max Axial</td>
<td>5.0</td>
</tr>
<tr>
<td></td>
<td>Max Shear</td>
<td>55.6</td>
</tr>
<tr>
<td>Shroud Coord 3 Bolts</td>
<td>Max Axial</td>
<td>9.1</td>
</tr>
<tr>
<td></td>
<td>Max Shear</td>
<td>64.0</td>
</tr>
<tr>
<td>Shroud Coord 4 Bolts</td>
<td>Max Axial</td>
<td>4.7</td>
</tr>
<tr>
<td></td>
<td>Max Shear</td>
<td>26.0</td>
</tr>
<tr>
<td>Shroud Coord 5 Bolts</td>
<td>Max Axial</td>
<td>8.3</td>
</tr>
<tr>
<td></td>
<td>Max Shear</td>
<td>48.6</td>
</tr>
<tr>
<td>Shroud Coord 6 Bolts</td>
<td>Max Axial</td>
<td>5.7</td>
</tr>
<tr>
<td></td>
<td>Max Shear</td>
<td>41.7</td>
</tr>
<tr>
<td>Shroud Coord 7 Bolts</td>
<td>Max Axial</td>
<td>3.9</td>
</tr>
<tr>
<td></td>
<td>Max Shear</td>
<td>36.9</td>
</tr>
<tr>
<td>Camera</td>
<td>Max Axial</td>
<td>9.3</td>
</tr>
<tr>
<td></td>
<td>Max Shear</td>
<td>145.9</td>
</tr>
<tr>
<td>Computer</td>
<td>Max Axial</td>
<td>84.8</td>
</tr>
<tr>
<td></td>
<td>Max Shear</td>
<td>1229.5</td>
</tr>
<tr>
<td>Power Distribution Plate</td>
<td>Max Axial</td>
<td>51.6</td>
</tr>
<tr>
<td></td>
<td>Max Shear</td>
<td>553.5</td>
</tr>
<tr>
<td>Oven</td>
<td>Max Axial</td>
<td>33.9</td>
</tr>
<tr>
<td></td>
<td>Max Shear</td>
<td>767.7</td>
</tr>
<tr>
<td>Oven Mounting Bracket</td>
<td>Max Axial</td>
<td>18.6</td>
</tr>
<tr>
<td></td>
<td>Max Shear</td>
<td>406.2</td>
</tr>
</tbody>
</table>

Due to the breathing nature of the 3rd Natural Frequency (Figure 11), concern was raised that the shroud may deflect into RIGEX internal components and jeopardize the science of the mission. The displacements shown in Figure 11 are scaled for easy viewing, but, due to the nature of an eigenvalue analysis, they can be used only for shape, not actual translation values. Thus, the maximum limit load combination output data was utilized again to find the maximum possible deflection of the shroud in towards the RIGEX hardware. The ovens were identified as the highest risk area because their corners come closer to the shroud than any other component, and their integrity is essential in order to reach the $T_g$ and allow the tubes to become malleable. Figure 12 shows a close-up view of the oven-shroud proximity issue.

![Figure 12. Oven Proximity to Shroud (Inches)](image)

For this analysis, the nodes around the shroud between bolt locations were selected, as they are the most likely to have the greatest deflections. This same Matlab algorithm was used to ensure the bumpers would not hit the inside of the CAPE under maximum loading conditions. While the bumpers are there to protect the CAPE from metal-on-metal damage that would occur if it was struck by RIGEX, an optimal flight would encompass no contact at all between any part of RIGEX and any part of CAPE. The bumpers are also located between the bolts that hold on the shroud, so only a few more nodes (those on the oven mounting plate, in between the shroud connection bolts) were entered into the Matlab code. The new Matlab code then found the maximum translations at the shroud nodes and bumper nodes, and then rotated them...
appropriately around the z-axis for their given locations. The output from this analysis revealed that the maximum deflection these critical locations, under any of the 64 maximum loading conditions, would be just over 1/16th inch, and therefore not cause any problems to the ovens or the CAPE.

**Finite Element Model Summary**

Finite element modeling and analysis are powerful payload risk mitigation tools that ensure adequate strength of design. Development of the RIGEX structural model has progressed through various plate thicknesses, increasing radii, and the addition of a shroud. While the flight model design has not yet been built, unique challenges were overcome in gaining confidence in the RIGEX FEM. Through extensive testing and analysis of the EM structure and a set of preliminary FEMs, confidence in the FEM modeling method was obtained for application in the final RIGEX FEM. A first natural frequency of approximately 185 Hz was determined for the RIGEX flight model. Maximum loading values at all bolt locations and maximum deflections of the shroud and bumpers were also determined. Analytical FEM documentation, along with future flight model acceptance testing, will provide AFIT and STP with adequate structural verification data for launch. The final RIGEX FEM, massed structure with shroud, will continue to be used for loads analysis and modal frequency comparison until all NASA requirements are met. The next step was to validate the fastening hardware, as described below.

**BOLT ANALYSIS**

To fly aboard the space shuttle orbiter, a payload’s bolts must have at least two locking devices to ensure maintenance of the joint’s integrity in the unique environment of space flight. On RIGEX, every bolts is assigned a preload, which will be recorded and maintained and will serve as the first locking device. The second locking device for the bolt is dependant on its location and accessibility on the RIGEX structure. For bolts that will likely need to be removed and retorqued multiple times, a locking helicoil is used. For bolts that will not be removed and are inaccessible within the hardware, patchlock is employed. Finally, locknuts are used on bolts that are easily accessible and where the nut will not interfere with inflation hardware. The RIGEX bolt analysis was completed in accordance with the NASA document governing preloaded bolts, NSTS 08307, Revision A. Use of criteria in this document will ensure that any preloaded bolt on the orbiter will exhibit adequate strength, a separation

factor of safety at limit load, and adequate fracture and fatigue life. The overall goal of the bolt analysis was to determine a range to which the bolts of a given bolt pattern may be torqued to, that will ensure those bolts will meet said criteria. With certain loads applied to certain bolts, the torque range can become either unreasonable or nonexistent, in which case a stronger bolt must be used or a change must be made to the bolt pattern to better distribute the load.

**Methodology**

An iterative process was used in the bolt analysis. The first step was to determine the type of bolts to be used at each location. Initial bolt sizes were governed simply by what would fit easily into a given configuration compiled with a reality check to ensure bolts would not be too small. Bolts were chosen only that complied with National Aerospace Standards (NAS) for Corrosion Resistant Stainless Steel (CRES). The material of choice for the bolts was A286 CRES due to its stellar spaceflight heritage. Being NAS fasteners, the bolts all are required to comply with Aerospace Standard (AS) 8879, which governs UNJ profile screw threads. Based on a bolt’s diameter and threads per inch (n0), this document reveals the tolerance on the major diameter and pitch diameter of the threads based on the diagrams provided (Figures 13 and 14).
Material data on A286 was then obtained from MIL-HDBK-5B, which provided the yield and ultimate shear and tensile strengths of the material along with the modulus of elasticity. Similar data was obtained for the structure’s 6061 aluminum.

Once an exhaustive list of properties for each type of bolt and for the tapped holes in the aluminum structure was developed, the analysis could proceed. The first step was to ensure the cross section of the bolt would be sufficient in the sense of a basic rod under an axial load. This was done by ensuring the axial load allowable ($P_{Ax}$), based on the size and material properties of the bolt, is greater than the maximum axial load the bolt could experience based on the FEA, ($P_{Ax,FEA}$), with a 1.4 factor of safety ($SF$). This criterion is met when Equation 2 is satisfied.\(^{12}\)

$$\frac{P_{Ax}}{SF \times P} - 1 \geq 0$$ \hspace{1cm} (2)

The second criterion that must be met is essential in ensuring the threads to not shear away from the bolt or tap. This criterion, Equation 3, insists that the shear load allowable ($P_{Ax}$) for the given bolt does not exceed the expected load as found in the FEA, ($P$), with a 1.4 factor of safety ($SF$). This criterion is met when Equation 2 is satisfied.\(^{12}\)

$$\frac{P_{Ax}}{SF \times P} - 1 \geq 0$$ \hspace{1cm} (3)

Finally, the analysis must prove that the shear load the bolt will feel under the load limit, ($V$), as determined in the FEA, will not exceed the shear load allowable ($VA$). $VA$ is determined based on the material properties of the bolt. Equation 4 must be satisfied to meet this criterion.\(^{12}\)

$$\frac{VA}{SF \times V} - 1 \geq 0$$ \hspace{1cm} (4)

If Equation 2, 3, or 4 was not satisfied, the bolt must be increased in size or changed to a stronger material, or the bolt pattern must be amended and the FEA redone to better disperse the load.

After determining that the chosen bolt would not fail directly under given loading conditions, an analysis was done to determine the minimum and maximum torque values could proceed. The maximum possible load (both axial and shear) that a bolt would need to endure was extracted from the load limit FEA discussed earlier in this paper. The NASA document, Torque Limits for Standard, Threaded Fasteners (MSFC-STD-486B), offered an average value starting point for minimum and maximum torque values that a bolt of a given size could comfortably carry.\(^{14}\) These values were used as initial values in the iterative process used to find actual minimum and maximum torque for the bolt using Equations 5 and 6.\(^{12}\) In these equations, $PLD$ represents the maximum or minimum preload that the bolt will experience given a specific input torque, $T$. $\Gamma$ is the uncertainty factor and $K$ is the typical nut factor, which for unlubricated bolts, such as those used in RIGEX, are 0.35 and 0.2 respectively. $D$ represents the basic diameter of the external (bolt) thread. Thermal loads are represented by $P_{thrtyp}$. The expected preload loss, $P_{loss}$ is defined as 5 percent of the maximum calculated preload. $T_p$ is the prevailing torque, defined as the torque needed to initiate rotation of the bolt given its locking device.

$$PLD_{max} = \frac{(1+\Gamma)T_{max}}{K_{typ} D} + P_{thrm}$$ \hspace{1cm} (5)

$$PLD_{min} = \frac{(1-\Gamma)(T_{min} - T_p)}{K_{typ} D} + P_{thr} - P_{loss}$$ \hspace{1cm} (6)

The input maximum and minimum torques were then varied until the preloads allowed for positive margins in the applicable preloaded bolt strength and preloaded bolt separation criteria. First, a bolt axial load resulting from the preload is calculated (Equation 7).\(^{12}\) The loading plane factor, $n$, is typically 0.5 for standard bolts. The stiffness parameter, $\phi$, is based on the stiffness of the bolt and nut (or tapped hole) as derived from their moduli of elasticity. A factor of safety ($SF$)
of 1.4 is used per NASA requirements. The external
load applied to the bolt, as found in the FEA, is \( P \) in the
equation. Finally, the preload value was adjusted by
changing the maximum input torque until Equation 8
became true.\(^{12}\)

\[
P_b = PLD_{\text{max}} + n\phi(SF \times P)
\]  

(7)

\[
\frac{PA_{s}}{P_b} - 1 \geq 0
\]  

(8)

The maximum torque was also dependant on satisfying
Equation 9, which is an essential step in ensuring
the threads of the bolt or tap will not shear away.\(^{12}\) \( PA_s \) is
the shear axial load on the bolt based on its size and the
strength characteristics of its material.

\[
\frac{PA_{s}}{P_b} - 1 \geq 0
\]  

(9)

The final criterion which drives the torque values is
preloaded bolt separation. This criterion ensures that
the bolt will not separate away from the structure. The
separation load is determined with Equation 10, which
uses the NASA separation factor of safety \( (SF_{\text{sep}}) \) of
1.2. A new \( P_b \) is calculated, using Equation 11.\(^{12}\) The
criterion in Equation 12 uses simple linear preloaded
joint theory as long as the bolt is not loaded above its
yield allowable.\(^{12}\) When Equation 12 is true,
confidence is gained that the bolts will not separate
away from the structure under any given load, thus
weakening and causing damage.

\[
P_{\text{sep}} = P \times SF_{\text{sep}}
\]  

(10)

\[
P_b = PLD_{\text{min}} + n\phi P_{\text{sep}}
\]  

(11)

\[
\frac{PLD_{\text{min}}}{(1 - n\phi)P_{\text{sep}}} - 1 \geq 0
\]  

(12)

The input minimum and maximum torques were varied
until Equations 8, 9 and 12 were all satisfied.

The final criterion of the bolt analysis is that fracture
and fatigue life must be adequate. While this portion of
the analysis is still pending, it may not be necessary at
all due to the brevity of the RIGEX mission. Additionally, all bolt torques will be carefully tracked
in a torque log and bolts with patchlock will be
replaced rather than re-torqued if the need arises.

**Assumptions**

In the bolt analysis, the following assumptions are made:

- The bolts, locknuts, and helicoils are all in
  compliance with their applicable NAS. This
  includes their dimensions all being within the
  published tolerances, which are used in the bolt
  analysis.
- The tapped holes in the RIGEX will have an
  internal thread in compliance with the threads in
  AS8879 and therefore fit their bolts within the
  required tolerances.
- The 6061 aluminum plates the RIGEX structure
  is made from is considered a perfectly uniform solid,
  in compliance with the specifications for yield and
  ultimate strength and modulus of elasticity as
  outlined in MIL-HDBK-5B.
- The fastener material (most often A286 CRES)
  made from is considered perfectly uniform, in
  compliance with the specifications for yield and
  ultimate strength and modulus of elasticity as
  outlined in MIL-HDBK-5B.
- The RIGEX computer is divided into two point
  masses, at it two mounting locations. While it is
  actually held to the structure by four bolts, it is
  modeled as being held by only two. All other
  RIGEX subsystem components are considered
  point masses, secured to the structure by a single
  bolt of the proper type. This is not the case in
  reality but it s very conservative estimate as, if one
  bolt can hold the structure, four certainly can.
- Bending loads are considered negligible and not
  applied to the bolt analysis.
- Prevailing torque for the helicoil inserts is
determined from the minimum locking torque after
the 15th cycle (worst case) as published from Heli-
Coil.
- Prevailing torque for bolts with patchlock is found
  from the minimum allowable breakaway torque
  (worst case) as published in MIL-F-18240E.
- Bolt yield is considered failure for the purposes of
  the RIGEX mission except where otherwise noted.
- Thermal loads are considered negligible.

While not all of these assumptions are conservative, the factors of safety applied to the calculations allow confidence in the analysis to remain high.

Validation

Physical test validation of the bolt analysis is pending. Bolts will be destructively tested at NASA Johnson Space Center (JSC) to ensure compliance with the specifications used in the analysis.

Results and Analysis

The analysis described in the Methodology section above was repeated for each of the 11 bolt patterns and for the bolts constraining each of the major subsystem components.

The constraint bolts, those which hold the RIGEX Top Plate to the CAPE Mounting Plate, proved the most interesting analysis. These bolts were originally slated to be 1/4-28 A286 CRES, spaced around the Top Plate in 28 locations at a 9.75” radius. Unfortunately, under this configuration, the bolts failed their shear loading strength criteria. This motivated a design change, and the bolts were increased to 3/8-24 A286 CRES. Unfortunately, with the increase in bolt diameter, the bolts needed to be moved inwards on the structure, to a radius of 9.5”. Once this change was implemented, the constraint bolts were able to pass all criteria, as shown in Table 5. The available torque range for these constraint bolts is 204 – 741 inch-pounds.

<table>
<thead>
<tr>
<th>Criteria</th>
<th>PASS/FAIL</th>
<th>Margin</th>
</tr>
</thead>
<tbody>
<tr>
<td>Min Cross-Section of Bolt Eqn 2</td>
<td>PASS</td>
<td>258.2</td>
</tr>
<tr>
<td></td>
<td>Eqn 8</td>
<td>0.004</td>
</tr>
<tr>
<td>Shear Pull-Out of Threads Eqn 3</td>
<td>PASS</td>
<td>579.7</td>
</tr>
<tr>
<td></td>
<td>Eqn 9</td>
<td>1.3</td>
</tr>
<tr>
<td>Shear Load Eqn 4</td>
<td>PASS</td>
<td>3.5</td>
</tr>
<tr>
<td>Separation Criteria Eqn 12</td>
<td>PASS</td>
<td>0.97</td>
</tr>
</tbody>
</table>

Similar results were obtained for the shroud bolt patterns and each of the large subsystem components. For each of those bolt patterns, a minimum and maximum preload was established as well as ensuring that all criteria were met.

Bolt Analysis Summary
Preloaded bolt analysis is a valuable tool for ensuring structural integrity during spaceflight. The RIGEX bolt analysis even motivated design changes to ensure that shearing would not occur on the bolts connecting the RIGEX structure to the CAPE Mounting Plate.

While the bolt analysis is only as accurate as the assumptions made, many of the assumptions were conservative and those that were not are overshadowed by the large factors of safety applied to the loading equations. By passing all of the criteria put forth in NASA’s Criteria for Preloaded Bolts, construction of RIGEX can proceed with confidence that the bolts will have adequate strength, a proper separation factor of safety, and adequate fracture and fatigue life in order to function properly as fasteners throughout the entire mission profile.

CONCLUSIONS AND RECOMMENDATIONS

The Rigidizable Inflatable Get-Away-Special Experiment concept was developed in 2001 and, after several design iterations, it is now undergoing construction for its flight aboard the Orbiter in summer 2007. Before any assembly could begin, a thorough structural analysis using analytical and numerical computer tools needed to be completed. Otherwise valuable time and financial assets could have been wasted on building an unsound structure.

Conclusions

The RIGEX FEM produced reasonable results. These results can be quantitatively assessed once the RIGEX flight model is assembled and tested in the lab. Model accuracy is currently based on method validation via EM lab data compared with a set of preliminary FEMs. The mode shapes and frequencies obtained from eigenvalue analysis verified the hypothesis of bending modes before axial or torsion modes. The FEA also showed a first natural frequency considerably higher for the new RIGEX FEM than those observed for the EM, which is to be expected due to the thicker nature of the structural plates and the addition of a shroud. The first natural frequency of 185 Hz easily meets the STP minimum first modal frequency requirement of 50 Hz. Finite element analysis also revealed that the RIGEX shroud will not strike any internal components., nor will RIGEX strike the inside of CAPE, even under the highest static loads. This allows the structure to fly with minimal impact protection. The FEA also revealed maximum expected loading conditions at all bold locations, which could then be used in the RIGEX bolt analysis. This analysis, driven by NASA’s Criteria for Preloaded Bolts, leads to a high level of confidence that the RIGEX bolts will not fail under any spaceflight loading condition. Therefore, RIGEX will not jeopardize its internal science experiment, the CAPE, the orbiter or the astronaut crew. The information presented in this document supports the plan to continue use of the full RIGEX FEM for any future analyses.

Recommendations

The cyclic intent of FEM design will come into play when the RIGEX flight model is built and undergoes acceptance testing. Then lab data for the exact structure represented in the RIGEX FEM will become available. Correlation with the acceptance test results will provide an opportunity for revision of the RIGEX FEM. If the results do not agree, then refinement and modification of the RIGEX FEM should be done until it accurately represents the physical specimen. Per the Payload Verification Requirements document, NSTS 14046, an analytical model should match test result to within 5% of the primary modes and 10% of the secondary modes. This standard should be used to show a sufficient quantitative assessment of the RIGEX FEM when compared to lab data.

SUMMARY

A RIGEX FEM was created using FEMAP software. Eigenvalue analysis of the model showed a margin of over 130 Hz with respect to the required first modal frequency minimum. Based on the FEA, the RIGEX structure will meet requirements for response to in-flight loading, thus preventing damage to itself and the CAPE. Furthermore, bolt analysis based on FEA and NASA criteria revealed a high level of confidence that the structure will remain fully intact at the limit loads.

Many current satellite technology concepts involve the use of large space structures that are limited by launch vehicle size and weight constraints. Inflatable, rigidizable structures can “potentially revolutionize the design and applications of large space structural systems.” Development and launch of the RIGEX payload will increase knowledge of inflatable, rigidizable structures by providing on-orbit reliability data and an assessment of ground test methods for future applications.

Acknowledgements

The authors would like to thank Maj. Eric Swenson and their coworker, Captain Jeremy Goodwin, for all of their guidance, support and motivation.
References


