EMBRY-RIDDLE Aeronautical University
SCHOLARLY COMMONS

McNair Scholars Research Journal

Volume 3 | Article 6

2016

Material trades and structural analysis of a composite structure

Jon Willems
Embry-Riddle Aeronautical University

Material trades and structural analysis of a composite structure

Student Scholar: Jon Willems
Advisor: Dr. Ilteris Demirkiran
McNair Scholars Program
Embry-Riddle Aeronautical University
600 S. Clyde Morris Blvd.
Daytona Beach, FL 32114
August 3rd, 2015

Abstract

In order to design a composite system specific for a particular mission, it is necessary to perform a materials trade study and structural analysis first. Through the use of software such as: Analam, 6strut, and Strut Design Tool the best material makeup can be identified. Then, by using Patran, MSC Nastran and Sigmax a finite element model based on a real structure can be set up and used to find stresses and margins of safety as well as modal frequencies. The results of this analysis are that the structure's first mode was found at 20.41 Hz. The lowest margin of safety in the composite skins was found to be 2.6180, which indicates no failure and room for improvement. Also, forces in the struts of the kinematic mount were found and compared to a rough analysis from 6strut. The max force from sigmax in any of the struts was found to be 612.0 lbs and for its given loading pattern had a percent deviation from what 6strut expected for said strut of 57.29%.

Introduction

The project being discussed in this paper is a materials trade and structural analysis of a composite structure. The composite structure being analyzed is a composite box housing sensitive optics. This composite box is strut mounted to the bottom of a Metering Structure, through the use of a kinematic mount consisting of composite components. This box is to be comprised of a honeycomb sandwich with two composite facesheets, with materials that will be later discussed. The struts are comprised of many components, however, only the main two components will be analyzed in this paper. This is because they are the parts that drive the performance of the entire strut. These two components are a composite hollow tube and two metal flexures. The two metal flexures are screwed into either end of said tube.

This research effort is looking to accomplish a few main goals. The first is to find the maximum force produced in each strut. These forces will be the results of being exposed to expected radial and axial load cases that would be seen during its flight into space. This would allow designing for the worst case scenario for each strut. The second main focus of this project is to determine at what frequencies the first 6 modes occur, for the system of the struts and the box. The last goal of this project is to ensure that positive margins are obtained when using the Tsia Wu failure theory for composites **[1]**. Having positive margins of safety are representative of a composite structure that did not fail under the stress placed on it. Before getting into the methodology of how these results were obtained, some background information is needed on both composite theory as well as kinematic mount theory.

A composite is a material that is made by combining two or more material **[2]**. Usually composites are a combination of a fiber and a resin matrix. Composites are often used for their light weight and potential for a wide variety of material properties. The fiber's role is to handle the majority of the axial loading and stress. This is because they are usually very stiff and strong in the fiber direction **[3]**. While the matrix's job is to transmit the load and distribute it among the other fibers, while also protecting the fibers which are generally brittle. Some examples of well-known composites are concrete, wood, bone and fiberglass **[2]**. Concrete is a combination of small stones and gravel and sand. While wood on the other hand is a combination of long cellulose fibers which are very strong. These fibers are then held together by a weaker material called lignin **[2]**. Bone is made of a strong and hard but brittle material called hydroxyapatite which is mainly comprised of calcium phosphate. The calcium phosphate is mixed with collagen which is a type of protein **[2]**. Fiberglass is a modern example of a composite and is comprised of glass fibers with a plastic matrix.

Composites come in many shapes and sizes and one way it is delivered is in the form of prepreg unidirectional. The unidirectional description indicates that all the fibers are parallel to one another. Prepreg is a reinforcing fabric that has been “pre-impregnated” with a resin system therefore consisting of fiber, a matrix and a curing agent **[4]**. One ply of this material is known as a lamina and is very strong along the fiber direction. However, it is weak when loaded perpendicular to the fibers **[5]**. In order to combat this, a laminate is formed by stacking many lamina plies. This is done with each ply having their fibers directed in different directions relative to a defined zero degree or one direction **[5]**. This turns the orthotropic lamina into a quasi-isotropic laminate. Orthotropic is a material with different properties when loaded in different directions. While quasi-isotropic materials have consistent material properties regardless of direction of loading. In order to further strengthen the laminate, it is set to cure in an autoclave. An autoclave is essentially an oven that can be both heated and pressured to desired values **[6]**. The autoclave’s main purpose is to cure the composite lamina by causing the curing agent to set. This greatly increases the strength of the laminate by turning it from a flimsy prepreg stack up, into a rigid and robust laminate.

There are many ways to make a laminate match the material properties needed for your mission. These material properties are largely driven by the fiber to resin volumes. These ratios can be altered by placing bleeder material inside the vacuum bag before it is placed into the autoclave. This bleeder material will suck out resin depending on how much and what material you use. Also, the layup scheme chosen and number of plies used can also alter the material properties of the laminate. Common layup schemes are +60°/-60°/0° and +45°/-45°/0° **[7]**. This means that once you have 0 degree direction defined, you layup a lamina with its fibers at a 45 degree angle to the zero. Then, you lay up a negative 45 degree laminate and continue until the amount of plies satisfies the properties you seek. One key thing to keep in mind is that an asymmetric layup will often cause warping and loss of strength. In order to satisfy a symmetric layup, take the case of the +45°/-45°/0° layup schemes. The first three layers would have to be 0°/-45°/+45° and then it would repeat in reverse, +45°/-45°/0° to make a balanced laminate. This would be the same idea for the 60°, -60°, 0° layup schemes.

The next and final thing is to give some background on kinematic mounts. A kinematic mount constrains all six degrees of freedom making them a great interface **[9]**. Also, known as a kinematic coupling they are precision interfaces used to connect two objects. In the case being discussed in the paper, the kinematic mount is the interface between the MS and the composite box **[8]**. The benefits of these kinematic mounts are they are statically determinate structures. This means the components of the mount can be designed independently of the objects they mount to. They also have very predictable behavior and can precisely position components, which are the main advantages of using them. However, they can be heavy, costly and have a lower first mode than non-kinematic mounts.

Methodology

The first step in order to conduct the analysis was to create a layup recipe through a materials trade study. This layup recipe requires that a high elastic modulus and very low, near zero coefficient of thermal expansion (CTE) is achieved. This is done through the use of a tool that was designed by Exelis, now Harris, named Analam. Once Analam is launched choosing the fiber, resin, film adhesive and honeycomb materials is the next step. The fiber and resin combination chosen for this mission was M55J with a 996 resin. These are both manufactured by Hexcel and were provided by the design engineer. The next step was choosing a film adhesive to adhere the honeycomb to the composite face-sheets. This was decided to be FM73 by the same design engineer. The honeycomb type however, was not provided and would need to be chosen through trial and error.

The first honeycomb chosen was CR III 1/8”-5052 1.6 pound per cubic foot also manufactured by Hexcel. This first honeycomb selection was an arbitrary one as it would be updated once the official layup scheme was selected. The honeycomb type would be systematically updated in order to fine tune the CTE values, which drive the whole mission. Now that materials had been selected for each quantity, the layup scheme would need to be derived. While there are many layup schemes, a +60°/-60°/0° layup scheme was chosen due to its well-known ability to achieve very low CTE values. The next step was to use Analam to find the amount of plies of M55J/996 lamina that were needed. This would be done through the repetition of a symmetric layup of 60°, -60°, 0°, 0°, -60°, and 60° and so on until the CTE values were lowest possible. Once the optimal layup scheme and number of plies were inputted into Analam, the honeycomb type was updated through a systematic approach. This approach was comprised of selecting successively lower density and smaller core diameter honeycomb types. This approach was implemented until it was arrived that the CR III 1/4"-5052 1.6 pound per cubic foot honeycomb type gave the lowest values necessary. The last step was to export a Nastran card consisting of a property and material card to be later used during the analysis. These cards are known as Pcomps and contain Nastran friendly text. This text has embedded in it the material properties of the lamina, honeycomb and film adhesive and angles at which they are layed up. Now that the composite structure of the box was created, the struts would have to be designed.

The design engineer provided the Analam layup for the struts. The strut layup was achieved by following the same steps above to create the scheme for the hollow tube. The design engineer also provided a file known as a 6 strut file. The 6 strut file possesses the number of struts needed, their lengths, along with end and beginning point locations. The next step to further design the struts was through the use of a program called the strut design tool. This tool was also developed at Exelis, now Harris. The first step was to import the Analam layup for the strut which would serve as the material for the hollow tube, while the flexures would be made of invar. Next, the wall thickness and length of hollow tube were selected. This was done in concert with the outer diameter of the flexure, length of the flexure and thread type of the flexure. These values were systematically updated until all margins of safety were within acceptable values. The acceptable values were any value greater than zero. Next, was to set up the finite element model.

A finite element model is needed to accurately represent reality and needed to calculate the results. The first step in creating the finite element model was to import the computer aided design (CAD) parasolid model. This was provided by the design engineer, and was imported into a program called Patran. Once the CAD model had been imported, surfaces were created at each of the mid planes of the panels of the model. These surfaces would serve as the geometry of the box. Next, the strut geometry would need to be created by making lines and using the end point locations from the 6 strut file so that they were located correctly. Once the struts and the box geometry had been recreated using Patran, the next step is to apply beam elements to the struts and grid to the box. The elements and grid are necessary for mapping the equations that are necessary for solving for the stresses and forces. They are also needed so that properties and materials can be precisely assigned. The next step could be started now that beam elements are applied to the struts and quad, and tria (squares and triangles) elements were applied to the box. For the next step, it was necessary to assign materials and properties to the elements and grid. Using the strut design tool, the material properties calculated for the tube and the flexures were assigned to the struts in Patran. The next step is to create multiple different coordinate systems, one for each box panel. Each coordinate system would have its origin on the box and its x-direction parallel to the intended fiber direction. Its y- direction would then be perpendicular to the fibers but in plane for each specific panel. With these coordinate systems in place, "dummy" panel properties could be assigned to each panel. This was all done making sure to reference its specific coordinate system. These are labeled as dummy properties because later in the process, when Patran outputs the model code, the Pcomp cards made earlier will replace the dummy properties. This is common practice. The below figures are included to help visualize the FEM model that was created. Due to this project being proprietary, a full scale photo of the structure cannot be attached.

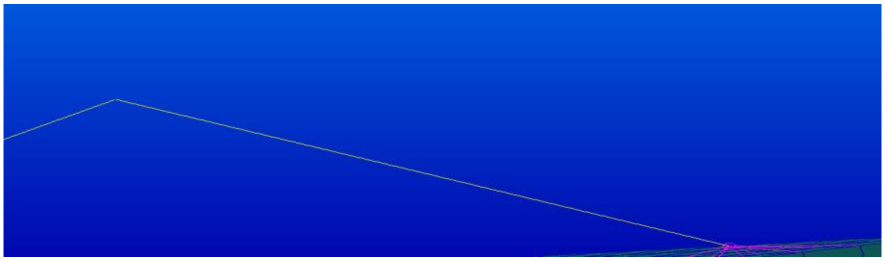

Figure 1: Picture of the strut geometry for an individual strut

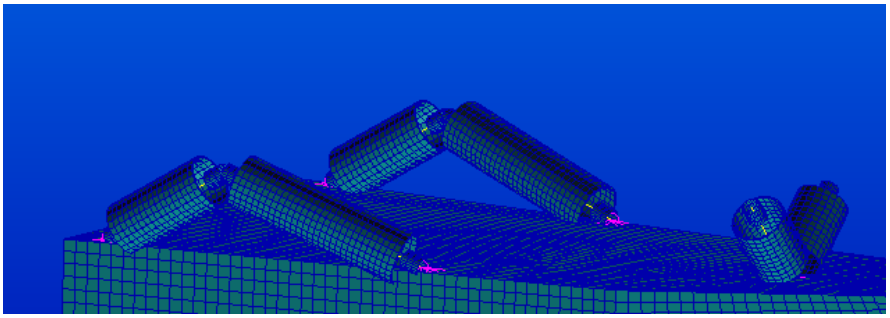

Figure 2: Picture kinematic mount struts with beam elements, grid and properties assigned

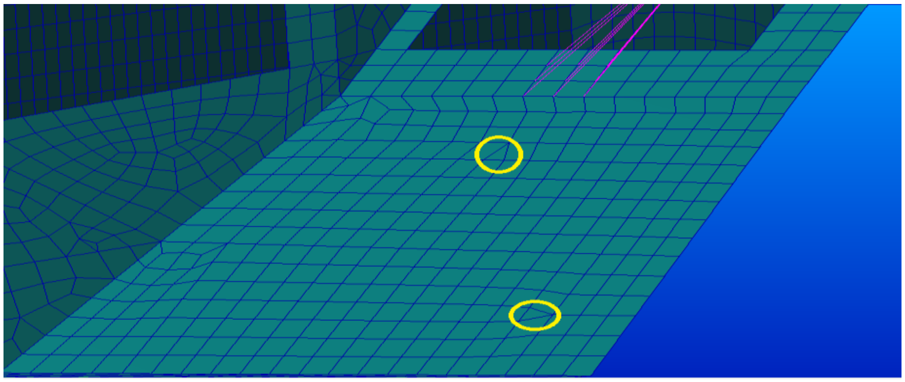

Figure 3: Picture of triangular elements known as "trias'

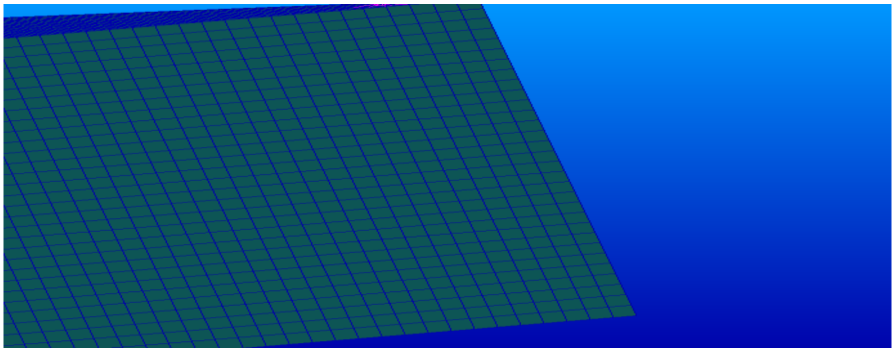

Figure 4: Picture of square elements known as "quads"

The next step was to attach the struts to the box through the use of Rigid Body Elements (RBE) 3 multipoint constraints (MPC). An RBE3 is an element that takes the average motion of all the defined independent nodes, and forces the single dependent node to move at that same average motion. This results in a connection with infinite stiffness that transmits loads without losses between the connections. The next step was modeling the two sensitive optics as lump or point masses. This is done first by creating a node at the center of gravity for each optic. Then, you are able to define its mass and moments of inertia in the material and properties sections. Once the nodes are created plus the materials and properties are assigned, they are then connected to the box through the use of RBE 3s. The connections attach at the places on the box defined by the design engineer. Now that the geometry has been created, the constraints and loads would need to be set up. The below figures depict RBE 3 connections and the lump masses.

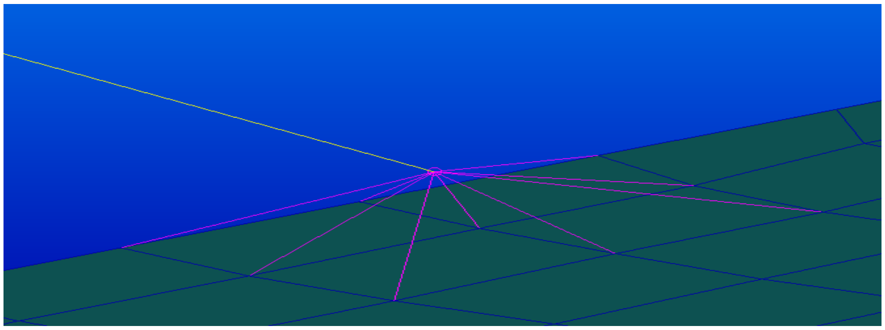

Figure 5: Picture of a RBE3 MPC

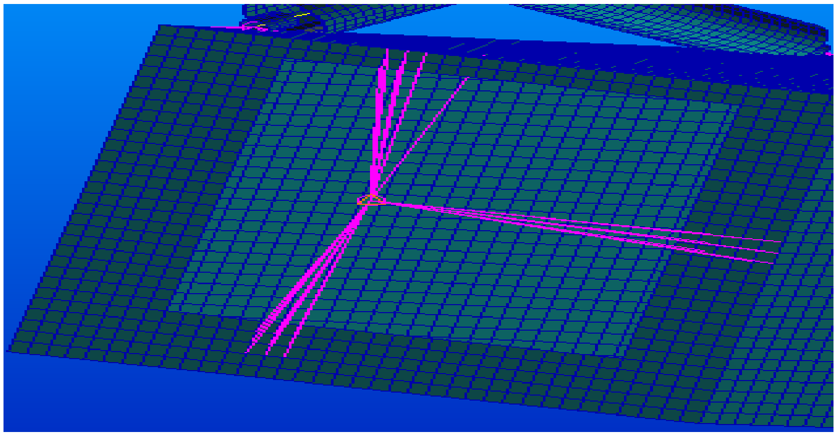

Figure 6: Picture of rear optic with its RBE3 MPCs

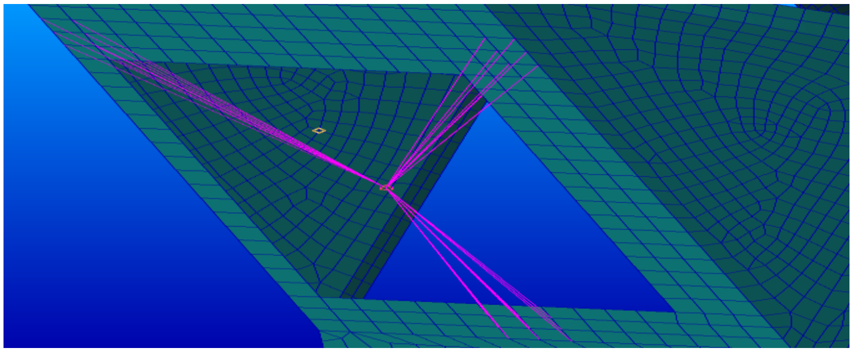

Figure 7: Picture of rear optic with its RBE3 MPCs

For the constraints, single point constraints (SPC) would be used at the tips of the struts as shown in the picture below. All six degrees of freedom are constrained, translation in the x, y and z-direction and rotations about the x, y and z-directions. This is done to ensure that a kinematic mount is simulated. For the load cases, a unit gravitational force is set up in the x, y and z-directions and will later be scaled to match the mission parameters in another software. Once this has been done, Patran is ready to output a bulk data file containing the model in a text form. This text is in a form that Nastran can interpret, run a simulation and output results.

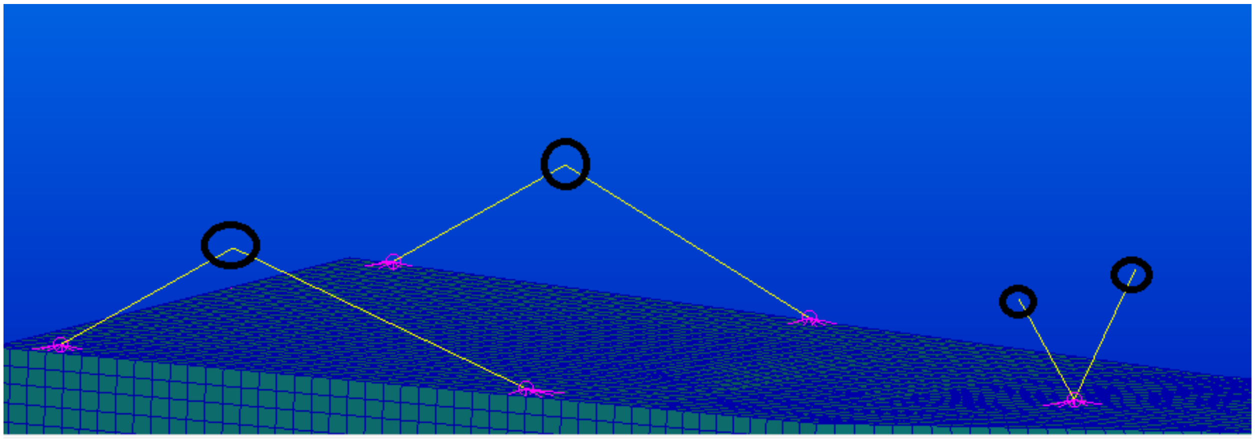

Figure 8: Picture of strut geometry with black circles indicating SPC's placement

Once this bulk data file has been created, the next step is to open the file and replace the dummy panel properties. This is done with the pcomp cards by deleting the originals and pasting all the text in Pcomp cards into the bulk data file. Next is to run the bulk data file through Nastran so that a punch or .pch file and a .f06 file are created. The .f06 file is used to find and locate any fatal errors that halt the .pch file from being created. The .f06 file is also used to validate that the system's weight is correct and that the reaction forces are equal to the weight of the box. This is because it is only being loaded by a unit gravitational force in this simulation. The .pch file will then be used with a software called Sigmax that scales and combines multiple load cases and outputs the desired values. Once the .pch file is created for both the panels and the struts, the next step is to make a include file or an .in file. This is a file that will tell Sigmax what to do with the .pch file. The .in file will include the scaling of the radial and vertical g loads. For this paper's analysis, 26 load cases were analyzed. These included 13 cases with 7 gs in the positive vertical, and sweeping radial loads of 11 gs at increments of 30 degrees from 0 to 360 degrees. The other

13 cases have the same sweep but with a negative 7 g vertical load. Once the .in files are complete, they are run through Sigmax using the Microsoft Disk Operating System, or MS DOS, also known as the command prompt.

Once these .in files have been run through Sigmax the resulting output files are Max files, or .max files. These files contain two thirds of the results sought in this project. The .max file for the panels possess the margins of safety for the composite plies. As long as this number is positive it means that the plies did not fail under the stress induced. The .max file for the struts contains the max forces in each strut at for each load case. In order to obtain the modal characterization of the system, the original bulk data file needs to be altered. It needs to be changed from a sol 101 to a sol 103, which is the type of solution sought.

Once the file is altered to a sol 103 and a few of the inputs are changed, it can be re-run through Nastran to receive another .f06 file. Once this file has been created and opened, the modal frequencies and corresponding modes are listed. All results are displayed in the following section.

Results

Table 1: Table of modal frequencies for composite system

Mode	Frequency (Hz)
1	20.41
2	30.33
3	58.83
4	97.3
5	111.74
6	121.54
7	139.01
8	202.17
9	316.89
10	348.32

Table 2: Max Force in each strut member with angle of applied radial g load found from Sigmax

Strut	Max Strut Force (lbs)	Theta of Applied Radial G load (°)
1	612.00	180
2	366.10	300
3	575.30	360
4	274.00	270
5	356.60	180
6	366.20	30

Table 3: Max Force in each strut member with angle of applied radial g load found from 6strut

Strut	Max Strut Force (lbs)	Theta of Applied Radial G load (°)
Strut 1	339.457	189.0
Strut 2	280.249	289.0
Strut 3	620.536	359.0
Strut 4	306.988	269.0
Strut 5	568.631	163.0
Strut 6	457.204	36.0

Table 4: Margins of safety for the composite laminate

Rank	Eid	Lid	Ply	Mid	FI	MS
1	374	14	26	13111111	0.1936	2.6550
2	372	14	26	13111111	0.1864	2.6180
3	434	2	28	13111111	0.1856	2.6480
4	418	20	3	13111111	0.1762	2.7910
5	416	20	3	13111111	0.1659	3.1810
6	935	7	27	13111111	0.1647	3.0120
7	376	14	26	13111111	0.1605	3.2890
8	406	7	26	13111111	0.1564	3.3920
9	247	1	26	13111111	0.1553	3.4210
10	408	7	26	13111111	0.1477	3.6100

Analysis

The results shown in Table 1 are the modal frequencies for the system. These values indicate the first 10 modal frequencies for which a mode exists. These modes indicate the multiple resonance cases. This means that if a modal frequency is encountered then the system of the box and struts will be subjected to resonance which can be catastrophic. This is the importance behind a modal characterization.

The results in Table's 2 and 3 indicate the maximum loads in the struts given a specific radial load case. Table 2 shows the values from sigmax while Table 3 contains the expected values from 6 strut. Below are tables of the percent differences between what was expected from 6strut and what was obtained from sigmax.

Table 5: Percent difference between sigmax and 6strut max strut force values

Max Strut Force from 6strut (lbs)	Max Strut Force from Sigmax (lbs)	Percent Difference
339.457	612	57.29
280.249	366.1	26.56
620.536	575.3	7.57
306.988	274	11.36
568.631	356.6	45.83
457.204	366.2	22.10

Table 6: Percent difference between sigmax and 6strut angle of applied radial load

Theta of Applied Radial G load from 6strut (°)	Theta of Applied Radial G load from Sigmax (°)	Percent Difference
189	180	4.88
289	300	3.74
359	360	0.28
269	270	0.37
163	180	9.91
36	30	18.18

As you can see from Table 5, there is fairly large percent differences between what was expected and what was found for the maximum strut forces. This was likely due to the fact that 6strut does not account for the bending in the flexures which is critical to the calculation. As you can see from Table 6, the percent differences between sigmax and 6strut were fairly low, which was expected. This is likely due to the fact that the loading is more consistent and the method by which the forces are found are not as stated above.

Table 4 contains data for the composite plies. It ranks the data 1 through 10 based off the values sigmax found for the margins of safety (MS). It list the element identification numbers (Eid) that correspond to the Patran model, the ply number that the lowest margins were found in, the material identification number (Mid) which corresponds to the M55J/996 lamina, the Load case identification number (Lid) and the Failure index (FI) which if it reaches the value of 1 there is a failure. According to this table, the composite layup that was created in Analam will not fail under the stress produced by the multiple different load cases. This is indicated by the fact that the lowest margin of safety turned out to be 2.618, which being positive means the composites did not fail. It also indicates that there can be a lot of weight shaved off this design. This is due to the fact that the lowest margin is much higher than zero. An ideal margin of safety is one that is higher than zero but not much higher. This indicates a system that won't fail and one that is as light as it can possibly be.

Conclusion

Through background knowledge of composites and kinematic mounts, it is possible to create from scratch a finite element model for analyzing a structure under given loadings. First, by selecting materials and determining a layup "recipe" or layup scheme, which could be a 45/-45/0 or 60/-60/0. Then, you can tweak things until you have the desired material properties through the use of Analam. Once these properties have been created for both the box and the struts, the finite element model can be created. This was done through the use of Patran in conjunction with the exported properties created from Analam. Then, this model can be run through both sigmax and Nastran in order to find the desired results based on the solution methods ran. Then, the results for the modes and the margins of safety can be examined. After this has been done, results from the 6strut file can easily be compared to the sigmax results through the use of a percent difference. As stated above, the reason for the large percent difference between sigmax and 6strut results is likely due to the fact that 6strut does not account for the bending of the flexures. While the first mode and the margins of safety looked normal and indicate that the created composite will not fail, there are likely errors in the founded results.

The main errors of this analysis are likely due to errors always encountered when using numerical and computational methods. Examples include conversions from decimal to binary and back. Also, there is error from which solution criteria you are using and the assumptions that the creator of the solution implemented. Also, there was likely human error due to the fact that the experiment was conducted in a very short period of time. This, combined with the fact that the researcher was learning the software while conducting the experiment, creates more error. There could also be error from the mishandling of data and incorrectly inputting values from one file to the next.

Bibliography

[1] Gray, C. E. (n.d.). Optimal Design of a Thermally Stable Composite Bench. 684.

[2] The Royal Society of Chemistry. (n.d.). *Composite Materials.* Retrieved from www.rsc.org: http://www.rsc.org/Education/Teachers/Resources/Inspirational/resources/4.3.1.pdf

[3] Taj, S., Ali Munawar, M., & Khan, S. (2006). Natural Fiber-Reinforced Polymer Composites. 1-3

[4] Fibre Glast Developments Corporation. (n.d.). What are Prepregs? Retrieved from Fibre Glast Developments Corporation: http://www.fibreglast.com/product/about-prepregs/Learning_Center

[5] Fibre Reinforced Plastic. (2010, December 3rd). Lamina and Laminate, What is that? Retrieved from http://www.fibre-reinforced-plastic.com/: http://www.fibre-reinforced-plastic.com/2010/12/lamina-and-laminate-what-is-that.html

[6] Engineer's Handbook. (2004-2006). Manufacturing Processes - Autoclave Molding. Retrieved from www.engineershandbook.com: http://www.engineershandbook.com/MfgMethods/autoclavemolding.htm

[7] Rhee, S. Y., & Cho, M. (2004). Layup optimization with GA for tapered laminates with internal plydrops. 2.

[8] MIT. (2004). *About Kinematic Couplings*. Retrieved from http://www.kinematiccouplings.org/: http://www.kinematiccouplings.org/

[9] Fellowes, D. (2006). Kinematic and Quasi-Kinematic Constraints: What They Are & How They Work. 1.