Finite element analysis of a multi-functional composite house

William Scholten\textsuperscript{a}, Dr. Darren Hart\textsuperscript{a}, Gabriel Esquivel\textsuperscript{b}

\textsuperscript{a}Department of Aerospace Engineering, Texas A&M University, College Station, TX, 77843, USA
\textsuperscript{b}Department of Architecture, Texas A&M University, College Station, TX, 77843, USA

E-mail: wds498@tamu.edu (W Scholten)

Abstract

The purpose of this investigation was to develop a technique using finite element analysis in Abaqus to analyze a geometrically complex architectural design of a multi-functional composite house that implements fiber elements to stiffen the structure. In order to analyze the design, realistic boundary conditions and loads were applied, quasi-isotropy was assumed for the material properties due to the orientation of the lamina, and Tsai-Hill failure criteria was implemented on an Abaqus model of a previous version of the design. Throughout this investigation, many parametric studies were performed on the model without fiber elements in order to observe how variations in loads, material properties, and thickness affect the model. As a result of this investigation, a Python code was developed that would create and then tie fiber elements onto the surface of the house. In order to test the accuracy of the model, test specimens made out of a similar composite as the one used for the Abaqus model were fabricated, and then tension and bending tests were performed on the specimens.

Nomenclature

\begin{align*}
E &= \text{Isotropic Young’s modulus (GPa)} \\
E_1 &= \text{Longitudinal Young’s modulus (GPa)} \\
E_2 &= \text{Transverse Young’s modulus (GPa)} \\
F_{1t} &= \text{Longitudinal tensile strength (MPa)} \\
F_{1c} &= \text{Longitudinal compressive strength (MPa)} \\
F_{2t} &= \text{Transverse tensile strength (MPa)} \\
F_{2c} &= \text{Transverse compressive strength (MPa)} \\
G_{12} &= \text{In-plane shear modulus (GPa)} \\
G_{13} &= \text{Out-of-plane shear modulus (GPa)} \\
G_{23} &= \text{Out-of-plane shear modulus (GPa)} \\
S &= \text{In-plane shear strength (MPa)} \\
\nu_{12} &= \text{Major in-Plane Poisson’s Ratio} \\
\nu &= \text{Poisson’s Ratio} \\
A &= \text{Area of application (m}^2) \\
C_D &= \text{coefficient of drag} \\
\rho &= \text{density of air (kg/m}^3) \\
\nu &= \text{velocity (m/s)} \\
F_D &= \text{Force of drag (N)}
\end{align*}
1. Introduction

There have always been differences in the approaches of how an architect designs and how an engineer designs. The architect focuses on being artistic in his designs whereas the engineer focuses on safety and efficiency. This research was born from these differences.

The Department of Aerospace Engineering at Texas A&M was approached by Mr. Gabriel Esquivel of the Department of Architecture to help him analyze a design of a multifunctional composite house that would be cantilevered from a cliff. (See Figure 25 in Appendix A) This design was unique due to the fact that throughout this geometrically complex shape, fiber elements were imbedded within the composite layup. These elements not only served artistic purposes, but also served many other purposes such as electrical, plumbing, and structural elements. The fiber elements were placed into the design using a method called non-linear agent base design which changes the position of the elements based on the structure around it. [1]

The original version of this design was created in the design programs called Rhino and Maya. The problem was that the architects had no real means of analyzing a composite structure. This led to the involvement of the Department of Aerospace Engineering because of its ability to perform analysis with composites. Another problem arose because the fiber elements could not be directly imported from the original design programs into Abaqus, the finite element analysis program which would be used to analyze the house design. The house design without fiber elements was able to be imported into Abaqus though. This resulted in the question of whether or not Abaqus could use its own features to accurately model the fiber elements and therefore accurately model the entire design itself. This investigation not only looked at the feasibility of the design; it investigated the ability to analyze it.

This paper is organized into three main parts. The first part will cover what was done in the setting up of the model without fiber elements for analyze such as the application of loads, boundary conditions and material properties. This first section will then discuss some general analyze such as mesh convergence and parametric studies. The next section will delve into the process that created a method which would implement fiber elements into the design. Finally the last section will discuss the preparation and testing of composite specimens, and the comparison of experimental data obtained from the tests to Abaqus models of the tests.

2. General Analysis of House Model

2.1 Developing an Improved Model

The original Abaqus model of the house used for this study was based on an orphan mesh of the model. An orphan mesh is an extracted mesh of a part or assembly of parts and can have loads and material properties applied to it. (The Abaqus documentation was used multiple times during this investigation so it will be referenced once) [2] The problem with this version was that selecting areas to apply loads was tedious because each element had to be selected, and the mesh could not be refined in any way. In order to alleviate this, an improved version of the model was created. This was done by first importing a part file containing the “parts” of the house into Abaqus as shells and then by stitching them together into one large shell part with a tolerance of 1. The new house part was modeled as a shell because the orphan mesh was originally comprised of the shell element type, STRI3. For this new house part, the metric unit system was chosen to describe its dimensions. In order to ensure that the stitching had no significant effects on the geometry of the model, the same loading, boundary conditions, and material properties were applied to both models in order to compare the contour plots (See Figures 26 and 27 in Appendix A) The contour plots of both models seemed to be the same. This meant that the stitching had no significant effect. As a result, the new improved model of the house was used for the rest of the study.

2.2 Loads and Boundary Conditions

With an improved model, boundary conditions and loads were applied in a more efficient way. Keep in mind though that the placement of the loads and boundary conditions
was subjective. Their placements were chosen in what was believed to be the most realistic location.

The boundary conditions for the model needed to simulate house being cantilevered from cliff by fiber elements. It was decided that the best way to model the cantilever would be to use the Encastre boundary condition on the edges where fiber elements extend from the house to the cliff face. (See Figure 28 in Appendix A) This fixed the model in place at the specified edges. The edges were chosen by examining artistic renditions of the house. [1]

Since this house was cantilevered from a cliff, it would be subject to many different loads. The loads chosen for this model were gravity, vertical wind force, lateral wind force, and snow. The gravity load was implemented simply by applying gravity acting in the U3 (vertical) direction with a magnitude of -9.81N/m² for the metric unit. The vertical and lateral wind forces were implemented by applying surface traction loads in user defined directions (U3 for vertical and U1 for lateral) to chosen surfaces on the house part (See Figure 29-30, Appendix A). The magnitudes of both loads were based off of the drag equation:
2.3.2 Isotropy

It was decided that the orientation of the lamina within the composite layup would be [0°/45°]n, where n was half the number of plies. This allowed for the assumption of quasi-isotropy to be made because the engineering elastic constants would be identical in all directions. [3] This meant that the material properties could be simplified to those seen in the Assumed Isotropic Values column of Table 1. The original idea for modeling the composite layup was to use the composite layup feature in Abaqus and specify the lamina properties $E_1$, $E_2$, $G_{12}$, $G_{13}$, and $G_{23}$. However, it became apparent with run time and memory usage that this approach was possibly not efficient for the investigation. With the assumption of quasi-isotropy, the layup was simplified into a single isotropic section with only $E$ and $v$ defined, saving a considerable amount of time and memory.

2.3.3 Tsai-Hill Failure Criteria

Since the model was relatively new, the conditions at which the model would fail were unknown. Unlike most aerospace structures, stress and strain were not good indicators of failure for this model because the complex geometry caused stress and strain concentrations, and the safety factor for the model needed to be large. It was unknown how the house would react to different loads and how much those loads would vary. Due to these unknowns, there needed to be a high safety factor. The structure could be experiencing relatively low stress. However, in reality, it could have already become unsafe.

For this reason, it was decided that Tsai-Hill Failure criteria would be implemented into the model. This failure criteria is a modified form von Mises yield that is used to determine failure of a composite material. [3],[5] What made this failure criterion useful was that it was already a built-in output in Abaqus and it requires an additional six material properties which were in the book that the original properties were taken from. These additional properties were $F_{1t}$, $F_{1c}$, $F_{2t}$, $F_{2c}$, and $S$. (see Table 1).

2.4 Mesh Convergence Tests

Mesh convergence tests are done to determine the type and the least number of elements needed for Abaqus to converge to a solution. For this study, four element types were tested. These element types were Quadratic Triangle (STRI65), Linear Triangle (S3R), Quadratic Quad-dominated (S8R and STRI65), and Linear Quad-dominated elements (S4R and S3R). In order to perform the tests at the same point for any mesh, a datum point was created on the surface of the house part because a node would be generated at that location every time. This allowed for the measurement of displacement in U1, U2, and U3, to be taken at the same point for any mesh and would in turn allow the convergence to be observed at that point. The number of elements was varied by changing the global seed size because the seed size directly affected the number of elements generated. As the global seed size decreased, the number of elements increased. For each element type and global seed size, the displacement was measured at the datum point.

The results of the mesh convergence in the U3 direction are seen in Figure 1 below. Based on the figure, both Quadratic elements appeared to converge to a displacement of approximately 0.198m, while the Linear Quad-dominated element appeared to be approaching 0.198m. The Linear Triangle elements failed to converge in the U3 direction. Generally, a larger number of elements meant a longer run time, so the element type that converged with the minimum global seed needed to be used for the model. However it was difficult to determine from Figure 1 what the seed size and element type should be. For this reason Figure 2 was created. Figure 2 shows the percentage error away from 0.198m (assumed converged value of displacement) vs. global seed size. Based on this graph, all element types except Linear Triangle converged within 1% of 0.198m. A smaller seed size would improve the model’s convergence when using Linear Triangle elements. However, the smaller seed size would be inefficient both for run time and in memory due to the increased number of elements. In order to be both efficient and accurate, it was decided that the model should have a 1% error. From Figure 2, Quadratic Triangle elements converged within
Figure 1 Mesh Convergence Test: Displacement in U3 vs. Global Seed Size

Figure 2 Percentage error of displacement in U3 vs. Global Seed Size

Figure 3 This figure shows the results of the parametric study of how shell thickness affects maximum Tsai-Hill Failure
1% at a larger global seed size (approximately 0.55) than the other element types. For this reason, Quadratic Triangle elements were selected for the model with a seed size of 0.55. However, due to an earlier mistake, displacement in U2 was used for the mesh convergence. This led to the temporary usage of Linear Triangle elements with a global seed size of 0.23. This mistake was not detrimental because at 0.23, the error in displacement in U3 was about 3.8%. However, all the parametric studies used this element because the mistake was noticed after the studies were performed.

2.5 Parametric Studies

After performing the mesh convergence tests, parametric studies on the loads, material properties, and the thickness of the house were performed. Only one value was changed at a time while everything else in model was held constant.

2.5.1 Thickness

A parametric study on the thickness of the house was performed in order to observe how the thickness affected Tsai-Hill failure and thus the safety factor of the house. Due to the fact that the model was a house, the safety factor needed to be relatively high. This study was done by varying the value of section thickness of the house part. The results of this study are shown in Figure 3.

Based on this figure, it appeared that as the thickness of the house increased, the maximum Tsai-Hill failure (referred to as maximum failure) decreased. The one exception to this trend occurred around a thickness of 0.31m where the maximum failure increased slightly and then continued to decrease at a slower rate than before. The reason for this “jump” and another “jump” around 0.05m was that the maximum failure changed locations. Before the first jump, the maximum failure occurred at the same location on the house, but when the jump occurred, the maximum failure occurred at a different location until the next jump. The lowest value of maximum failure obtained in this study was approximately 0.13 at a shell thickness of 0.31m. It was decided that this shell thickness would be used for the model because it had the lowest maximum failure. A smaller value could possibly have been obtained, however that could only be done by increasing the shell thickness to an unrealistic value. The one downside to using the maximum failure to determine the thickness was that certain areas on the model had failure criteria that were orders of magnitude lower than maximum. Eventually, the thickness of these areas should be reduced in order to avoid wasting material. However this was not done in this study.

2.5.2 Material Properties

A parametric study of the material properties of the E-glass/Epoxy material was performed in order to understand how variations in the different strengths affected the maximum failure. The strengths only affected Tsai-Hill failure for this model so variations in each of the strengths were observed while keeping everything else constant. Each strength was varied by a percentage amount. It was necessary to perform these studies, because the properties of the composite material are different for each manufacture. Does a slight change in one of the properties produce a drastic change in stress, strain, etc.?

Based on Figure 4, it was clear that shear strength had the most effect on the maximum failure than the other strengths while the longitudinal strengths had almost no effect on maximum failure. The transverse strengths appeared to cause variations when those strengths changed ±20% of the original value. It appeared that for this model there was an exponential relationship between shear strength and failure. From -30% to +30% of the shear strength, there was approximately a 42% difference in the maximum failure.

Like the results from the shell thickness parametric study, there were visible “jumps” in the maximum failure due to changes in location of the maximum. There also appeared to be location changes of the maximum failure that did not cause visible “jumps”. For instance, there was a change in location between -32% and -33% for F2. The jumps also appeared to affect how F2c, F2t, and F1c influence the variation of maximum Tsai-Hill failure. There was a change in location for S; however, it did not appear to affect the relationship.
The maximum failure remained constant for changes in $F_{1c}$ until -35% when the location of maximum failure changed. After this, the maximum failure slightly varied with changes to $F_{1c}$.

For changes in $F_{2t}$, the maximum failure slight varied. From -32% to 15% change in $F_{2t}$, the maximum failure decreased by approximately 5%. Around -32%, the maximum failure changed location which caused it to vary more with $F_{2t}$. Also, at 17% difference, there was a visible change of the maximum failure, but at this location, $F_{2t}$ has no effect on the maximum failure.

For changes of ±10% for $F_{2c}$, maximum failure was not affected. However, at a -10% difference, there was a change in location of the maximum, resulting in an inverse relationship between $F_{2c}$ and the maximum failure. As $F_{2c}$ decreased to -30%, there was a sharp increase in the maximum failure due to a location change. After this jump, the maximum failure appeared to follow the same trend as before.

The properties of the material that eventually will be used to construct this house will no doubt vary from properties used in the model. However, due to this parametric study, it was safe to assume that as long as the strengths do not vary more than 15%, the current model should be able to describe the failure of the house.

2.5.3 Loads

A parametric study of the loads acting on the model was necessary because they could easily vary. For example, one day there could be no wind and on other days there could be strong winds due to a storm. For each load, the change in the maximum Mises stress and maximum Tsai-Hill failure was observed. The force gravity was varied by changing the density a percentage amount because it directly affected the force. Both of the wind loads were varied by changing the velocity of the wind while keeping the other factors affecting the load constant. The snow load was varied by changing the thickness of the snow.

Based on Figures 5 and 6, density had a linear relationship with the maximum Mises stress and the maximum failure. This made sense because density and mass are linearly proportional. An increase in density increased the mass which in turn increased weight. The result of this would be an increased maximum stress and maximum failure. There was also a “jump” in maximum failure due to a change in location. Based on the results in both figures, it seemed that density was a major factor in the stress and failure. From -10% to +10% of 1900kg/m², there was approximately a 22% difference in both the maximum Mises stress and maximum Tsai-Hill failure.

The results of both the lateral and vertical wind forces are presented in Figures 7 and 8. From Figure 7, it appeared that the wind speed of both the lateral and vertical winds have
a parabolic relationship with the Mises stress. The reason for this relationship was that the velocity of the wind was squared when calculating the $F_D/A$. The difference between the two winds in this figure was that stress decreased as the vertical wind speed increased while the opposite occurred for the lateral wind. Mises stress increased as the lateral wind speed increased because there was more force acting on the side of the structure. Stress decreased as the vertical wind speed increased, because it counteracted some of the force due to gravity and snow. In essence, the load due to the vertical wind was a “lift” force. It also seemed that both of the loads due to wind did not have much influence on the stress of the house. From 0mph to 40 mph, the stress increased approximately 3% for the lateral wind load, while the stress decreased 1.4% for the vertical wind load.

From Figure 8, it appeared that under 20mph, the loads due to wind had almost no effect on the maximum failure. However, between 20mph and 25mph, there was a change in location of the maximum failure for both wind loads. After this jump, it seemed that both wind loads had more influence on the maximum failure than before the jump, but the failure only changed slightly. For instance for the lateral wind load, the maximum failure remained constant from 0mph to 20 mph while from 25mph to 40mph there was approximately a 1.88% percent change.

From Figure 9, the snow load appeared to have a direct proportional relationship with the maximum Mises stress. With each 0.02m increment of thickness, the stress appeared to increase 0.3MPa. From having no snow to having 5cm of snow, there was approximately a 4.3% increase. Based on Figure 10, the thickness of the snow appeared to have a linear relationship with the maximum failure from 0.0m to 0.01m and from 0.02m to 0.03m. Between 0.01m and 0.02m there was a jump discontinuity due to a location change. From 0.02m to 0.05m there was only a 2.7% increase in the maximum. It seemed that for both stress and maximum failure, the snow thickness had only a minor influence.

Based on the parametric studies of the loads, it appeared that density (the gravity load) affected the model more than the other loads.

The thickness of the house in these studies was 0.31m which resulted in a heavy house meaning that there was a large force acting on the house due to gravity. Changing the thickness might possibly change the influence of the loads on the model. However, due to time constraints, this theory was not tested.
3 Implementing Fiber Elements

Up to this point in the investigation, the model had been analyzed without fiber elements. However the main goal of this investigation was to develop a way to implement the elements. The approach taken in this investigation was based on a feature in Abaqus called a tie constraint. This feature allows Abaqus to “tie” to parts with dissimilar meshes together, in a sense combing them together. For the feature to function there needed to be a master and a slave surface. The slave surface was chosen to be the fiber element while master surface was chosen to be the surface on which the element would be tied to. Also for this investigation, it was decided that 3D wires would be used as the fiber elements because wires resemble the fiber elements used in the designs the most.

This section will discuss the procedure that was done to develop the method to create the Python code. Creating this code was one of the most important aspects of this investigation because it would allow for the creation and implementation of fiber elements into the Abaqus. As stated earlier, the fiber elements could not be directly imported into Abaqus. A more indirect method would have to be used.

3.1 Flat Plate

The first step in this process was implementing tie constraints on a cantilevered 2m by 1m flat shell plate, a “simplified” version of the model. The plate was cantilevered on one side while the other side was given shell edge load. It was also given a shell thickness of 0.1m. Various wires (straight and curved) were created as individual 3D planar wire parts and then tied to the flat plate. Different layouts of wires were
arbitrarily implemented on the flat plate. This allowed for the observation of how different layouts of wires effect the stress redistribution. Figure 11 shows one of the configurations. This configuration was five horizontal wires and two diagonal wires. As seen in this figure, the wires are deformed with the plate and have stress. In order to get an idea of how much the wires actually redistributing stress, values of stress were extracted along the center of the plate (where the middle wire is) for varying wire thickness. As seen in Figure 12, as the thickness of the wire increased, the lower the stress is on the plate. A thicker wire would obviously not deflect as much as a thin wire. Also it seemed that as the thicker the wire gets, the more the stress is reduced.

3.2 Non-flat Plate
After learning how to implement wires onto a flat plat, it was decided that wires would be tied onto a non-flat shell plate because it was a slightly more complicated geometry than the flat plate. Like the flat plate, the non-flat plate was cantilevered on one side while the other was loaded. The wires at first were created in the same manner as before. However it became apparent that 3D planar wire parts would not be efficient enough for creating wires whose coordinates depend on a 3 dimensional system because 3D planar wires were created in a 2D sketch plane. This led to the use of another feature in Abaqus called Create Wire: Point-to-Point. This feature would create a wire feature for a part based on input of three dimensional coordinates. For this investigation it was decided that a reference point part would be used to create Point-to-Point wire features on, because trying to create a wire feature on the surface, to which it would be tied to, occasionally resulted in a partition of the surface. With this new method, different configurations of wires were implemented onto to the non-flat plate. (See Figure 32 Appendix B for an example)

3.3 Creating the Code
After successively implementing fiber elements onto both the flat plate and the non-flat plate, it was time to create a code that would automatically create a wire and then tie it to the desired surface. This was done by examining the journal file for the non-flat plate in Notepad++

---

**Figure 11** Picture of contour plot of flat plate with wires tied to it.

**Figure 12** Stress vs. Distance for varying wire thickness
in order to find the methods Abaqus used to create and tie the wires. After finding the necessary lines of code, a Python script was created and then iterated upon until it could perform the desired task. The main theme behind these codes was the creation of wires using points. The programs that created the original model had to have used coordinate systems. This meant that every fiber element within that model was defined by points. These points could then be extracted and placed into a text file. This text could then be read and a fiber element could be generated in Abaqus with as much automation as possible.

The first iteration of the code created a wire feature based on a set of coordinates within the code. The main purpose of this code was to create a foundation for the iterations of codes after it. The problem with this script was that the coordinates had to be typed into the code in a specific format before running it.

The second iteration of the code used a for loop to generate points in a matrix based on a mathematical function that had to be typed into the code. It then created a wire feature for every two points. The downside to this code was that multiple wire features were be created for one part as well as multiple sets based on those features. However, each wire feature was set to merge with part geometry, so while the wires were individual at the feature level, they were merged with each other at the part level.

The third and final iteration of the Python code read in points from a text file, and inserted them into a matrix and created a wire feature for every two points. It also merged all the sets together, creating one set that spanned the length of all the wire features for that part. After creating the wire features, the code generated a mesh, applied material properties and tied the part to the surface. If the user needed to change the part name, element type, number of elements, etc., they would have to change it in the code themselves. The one flaw with this code was that the points defining the wire needed to be sorted so that they flow with the fiber element. If the points from the fiber element were stored randomly, it would create a wire, but it would not be the intended wire. With this last Python code, it was now possible to apply fiber elements onto the surface of the house part. (See Appendix B for code)

3.4 Applying Fiber Elements to House Part

Since there were no text files containing actual coordinates for a fiber element from the design of the house, it was decided that a few fiber elements would be implemented into the

Figure 13 Picture of house model with fiber elements
model by arbitrarily selecting points from which to create a wire. These points were obtained from either points already existing on the house part or by using a datum plane cut to partition the house and obtain more points. These partitions were done to an exact copy of the house in order to insure that the partitions did not affect the house in any way. After obtaining these points, they were sorted so that the code could actually generate the desired wire. The script was then run for each fiber element. These fiber elements were given the material properties of steel A36 so that they would serve as structural elements. (See Table 5 in Appendix B for properties). A job was then run with the house part and the wires in order to see stress redistribution if any. As seen in Figure 12, the wires were taking on stress however there was almost no stress redistribution. This was probably because there were only six fiber elements implemented currently when in reality there would have been over a hundred fiber elements. Since no stress redistribution could be seen, it was decided to see how the fiber elements affected the deflection of the house. A node was selected at the tip of the house and then both the thickness of the house and fiber elements was varied. As seen in Figure 14, the way the fiber elements influence the deflection depended on the thickness of the house itself. For this figure, positive change meant that the deflection was increasing while negative change was the opposite. If the house was too thin, the fiber elements would start to weigh the house down and if the house is too thick, the deflection wasn’t reduced as much.

4 Experimental Analysis

The final part of this paper discusses what was done in the testing of the assumptions used for the composite layup in the Abaqus model. Recall that the assumption was that since the lamina would be at $[0/45]^n$, a state of quasi-isotropy with one section could be used. This was done by comparing experimental data to Abaqus models that used either an actual composite layup or the assumed isotropic section used for the house model.

4.1 Fabricating Composites

In order to perform tension and bending tests that would test the validity of the assumptions and properties used for the composite layup, test specimens made out of a similar material as the one used in the Abaqus model needed to be fabricated. The composite that was fabricated was E-glass/Epoxy with a 2x2 twill weave. (See Appendix C Figure 33) Ideally, an E-glass/Epoxy composite with an 8-harness satin weave would have been fabricated however both weaves allow for the assumption of quasi-isotropy so the material properties should not be drastically affected.

The Architecture Department ordered both the E-glass fabric and the epoxy. The composites were fabricated at Texas A&M Riverside Campus using one of the department’s facilities. It was decided that three composite plates each with a different number lamina (3, 4, and 5 lamina) would be fabricated from which test specimens would be made from. Approximately, twelve 16 by 24cm rectangles were cut out of a 1 yard$^2$ E-glass cloth with 5 rectangles cut at a 45° angle (with respect to a side of the cloth) while the others were cut at a 0° angle. Starting with the 3 lamina composite, a mixture of epoxy and epoxy hardener was poured onto one layer of cloth and then spread across the cloth allowing it to soak in. After this another layer of glass cloth with a different orientation was placed onto the epoxy soaked layer, and the process would begin again. The 3 layer composite had lamina orientation of $0^\circ/45^\circ/0^\circ$. The 4 layer composite had lamina orientation of $0^\circ/45^\circ/0^\circ/45^\circ$. The 5 layer composite had lamina orientation of $0^\circ/45^\circ/0^\circ/45^\circ/0^\circ$. Once this process was...
complete for one of the composites, the epoxy soaked layers of E-glass were placed unto a flat surfaced coated with a releasing wax, and allowed to cure overnight. During the application of the epoxy, safety masks and gloves were worn due to the toxins released from the epoxy/epoxy hardener mixture. Once the composites were cured, they were easily removed from the flat surface on which they had been curing.

4.2 Preparing Test Specimens

The original plan for the specimens was to create multiple dogbone specimens out of the composite plates by using a press mold of a dogbone. However, pressing out a specimen proved to be unsuccessful so it was decided that rectangular stripes would be cut out of the composite plates using a band saw. Specimens were cut out at three different directions, 0°, 90°, and 22°. The goal of this was to test the assumption of quasi-isotropy in different directions. For each plate, 2 specimens were cut out at a 0° and a 90° angles while only one specimen was cut out at a 22° angle. Had the plates been larger, more specimens could cut. Due to grooves in the flat surface which had been used for curing, many of the test specimens excess epoxy on the side that was laying on the flat surface. This excess epoxy was shaved off using a sander. After this the dimensions of each specimen was measured. (See Figure 35 Appendix C for example of specimen)

4.3 Tension Tests

Tension tests were performed on three specimens (0, 90, and 22) from each composite plate. The purpose of these tests was to estimate Young’s modulus for the isotropic section and to calibrate the Young’s and shear moduli for the composite layup. The tests were performed using a load machine which would pull one side of the specimen. A laser extensometer was also used to measure strain. The program operating the machine, Testworks, recorded the load, time, crosshead extension, stress, strain, laser extension, and laser strain for each test. Before each test, the crosshead and the laser were zeroed so that strain would be zero at the beginning of the test, and gage length and grip separation were measured and inputted into the program. The specimen was then be pulled until fracture.

The results of these tests are presented as stress/strain curves in Figures 15-17. These figures only show strain up to 1% so that only the linear region could be analyzed. Each graph contains the stress/strain values for all specimens with the number of lamina. The laser extensometer was only able to accurately measure strain up to the thousandths decimal place. This resulted in the cluster of stress values for each 0.001 strain increment which was clearly seen in the figures. There were also initial values of stress at 0 strain for each specimen. This was due to the grips applying a load to specimens so that specimens did not slip during the test. In order to estimate Young’s Modulus for each specimen, linear trend lines were created. The slope of the trend line was approximately the value of Young’s Modulus for that specimen due to the relationship between stress, strain and Young’s Modulus.

The estimated values of Young’s Modulus for each specimen are presented in Figures 15-17 as well. Once the Young’s Modulus was estimated for each specimen, an average was taken for each ply number. The table below (Table 2) shows the average E and standard deviation for each lamina.

<table>
<thead>
<tr>
<th># of Lamina</th>
<th>Average E (GPa)</th>
<th>Standard Deviation (GPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>9.375</td>
<td>1.334</td>
</tr>
<tr>
<td>4</td>
<td>9.825</td>
<td>0.922</td>
</tr>
<tr>
<td>5</td>
<td>9.778</td>
<td>1.611</td>
</tr>
</tbody>
</table>

The standard deviation for each layer was high probably because of all three values of the average Young’s modulus were less than the assumed value of 24.5 GPa which was used for the model of the house. This was to be expected due to material flaws and differences in manufacturing. The important thing was that the values of the average Young’s Modulus had a standard deviation of approximately 2.63%.

The average value of Young’s modulus obtained for each set of specimens would be used for the isotropic section. However, in order
Figure 15 Stress vs. Strain data for 3 lamina tension tests from both experimental data and the composite layup. Each equation is matched with its corresponding data points.

Figure 16 Stress vs. Strain data for 4 lamina tension tests from both experimental data and the composite layup. Each equation is matched with its corresponding data points.

Figure 17 Stress vs. Strain data for 5 lamina tension tests from both experimental data and the composite layup. Each equation is matched with its corresponding data points.
to calibrate the composite layup, an Abaqus model of a tension test was made because the value obtained from experimental data was for the entire composite and not an individual lamina. It was decided that the shear moduli would be varied the same percentage amount as the Young’s moduli. For example, if the Young’s moduli were 12.25GPa (value from [3] was 24.5GPa), G13 would be half of 4.7 GPa. A virtual test specimen with averaged dimensions based on the dimensions of the four lamina specimens was created in Abaqus as a 3D shell part and given a composite layup which used lamina properties. It was then given an Encastre boundary condition at one end (symbolizing the fixed grip) and a Displacement/Rotation condition that gave the specimen a displacement of 0.001m in the U1 direction (distance along specimen).

After performing the job, the data for force vs. time in the U1 direction at one end of the specimen was summed and exported to Excel. Then the original coordinates and displacement in U1 vs. time for two nodes in the middle of the specimen approximately 2.5cm apart were obtained. Once these values were obtained, stress was calculated by dividing the force with the cross-sectional area (thickness x width) and strain was calculated by dividing the change in distance between the two selected nodes with the original distance. These stress and strain values were then plotted on the same figure as the experimental data (Figures 15-17) was on and a linear trend line was created so that the slope could be compared to the average value. If the slope was larger than the average value, the model was too stiff and the properties needed to be reduced. The opposite was true when the slope is less than average. This process continued until there was less than 1% difference between the slope and the average. Table 3 shows the value of E1 and E2 for each lamina.

The average of these values was approximately 11.36GPa with a standard deviation of 0.7. This averaged value will be used for E1 and E2 in the lamina properties used in the bending tests. As stated before, the reason why this value for Young’s moduli was higher than the Young’s Modulus used for the isotropic sections was that the average was based on the entire composite while E1 and E2 were used for an individual lamina's properties.

### 4.4 Bending Tests

The purpose of these tests was to test the assumption of quasi-isotropy and single section by comparing the estimated values of stiffness (Load/Displacement) from these tests to the values estimated in the tension tests for both the isotropic section and composite layup cases. Bending tests were performed on two specimens (0 and 90) from each composite plate. The type of bending test that was performed was a 3-point bending test. This was done by using the same machine and program used for the tension tests. However, the testing apparatus was set up to perform 3-point bending and the laser extensometer was not used (Figure 18). Before each test, the crosshead extension was zeroed for the same reason that it was in the tension test. Also, the distance between each grip, the distance between the two edge cylinders, and the diameter of the cylinders were measured.

![Figure 18 Testing Apparatus of 3-point bending](image)
Figure 21 Load vs. Displacement for the 4 lamina 0° Specimen

F/D = 5.1908 N/mm (Experimental)
F/D = 4.0879 N/mm (Composite)
F/D = 3.4165 N/mm (Isotropic)

Figure 19 Load vs. Displacement for the 3 lamina 0° Specimen

F/D = 2.8255 N/mm (Experimental)
F/D = 2.8064 N/mm (Composite)
F/D = 2.3581 N/mm (Isotropic)

Figure 20 Load vs. Displacement for the 3 lamina 90° Specimen

F/D = 4.227 N/mm (Experimental)
F/D = 4.2125 N/mm (Composite)
F/D = 4.4206 N/mm (Isotropic)
Figure 22 Load vs. Displacement for the 4 lamina 90° Specimen

Figure 23 Load vs. Displacement for the 5 lamina 0° Specimen

Figure 24 Load vs. Displacement for the 5 lamina 90° Specimen
Table 4 Comparing Experimental Stiffness to Estimated Stiffness

<table>
<thead>
<tr>
<th>Specimen</th>
<th>Experimental Stiffness (N/mm)</th>
<th>Estimated Stiffness: Isotropic (N/mm)</th>
<th>Estimated Stiffness Composite (N/mm)</th>
<th>Percent Error: isotropic</th>
<th>Percent Error: Composite</th>
</tr>
</thead>
<tbody>
<tr>
<td>3 lamina 0°</td>
<td>5.1908</td>
<td>3.4165</td>
<td>4.0879</td>
<td>34.18</td>
<td>21.25</td>
</tr>
<tr>
<td>3 lamina 90°</td>
<td>2.8255</td>
<td>2.3581</td>
<td>2.8064</td>
<td>16.54</td>
<td>0.68</td>
</tr>
<tr>
<td>4 lamina 0°</td>
<td>4.227</td>
<td>4.4206</td>
<td>4.2125</td>
<td>4.58</td>
<td>0.34</td>
</tr>
<tr>
<td>4 lamina 90°</td>
<td>5.0692</td>
<td>5.1417</td>
<td>4.8654</td>
<td>1.43</td>
<td>4.02</td>
</tr>
<tr>
<td>5 lamina 0°</td>
<td>8.2983</td>
<td>9.1727</td>
<td>9.9418</td>
<td>10.53</td>
<td>19.81</td>
</tr>
<tr>
<td>5 lamina 90°</td>
<td>56.289</td>
<td>45.384</td>
<td>49.119</td>
<td>19.37</td>
<td>12.74</td>
</tr>
</tbody>
</table>

For this test, load, time, extension, stress, and strain were measured. However, the values of stress had to be disregarded because the area of contact was unknown which means the values of stress are inaccurate. Since stress/strain curves could not be accurately created, another method had to be used to estimate \( E_1 \).

This alternate method was comparing the experimental data to an Abaqus version of 3-point bending. The results of these tests are presented in Figures 19-24. There is an initial load in each figure for the experimental data due to the crosshead being in contact with the specimens before the test began. For each load vs. displacement, a linear trend line was created for the linear region.

The Abaqus model of 3-point bending was created by using a 3D shell part for half of each specimen and 3D analytic rigid shells for two cylinders with reference points at the center of the cylinders. Only half of a specimen and two cylinders would be used because the testing apparatus is symmetric so a symmetry boundary condition was used. For each specimen, the cylinders were positioned so that they matched the positions in the actual experiment exactly. (See Figure 35 Appendix C) Since shell thickness was only used in calculations, both cylinders had to be moved by half of the shell thickness so that the cylinders lay on top of the shell. The following boundary conditions were applied to the model: the edge cylinder was given a Displacement/Rotation boundary condition that constrained translation and rotation in all directions, the center cylinder was given a Displacement/Rotation condition that constrained translation and rotation in all directions except for translation in the U3 direction, the half specimen was given a Displacement/Rotation boundary condition at edge with the center cylinder which constrained movement in U1 and U2 and at the far edge of the specimen which constrained movement in the U1 direction. Also a symmetry condition was applied at the “center” of the specimen. Each specimen was displaced -0.01m by the center cylinder. After this, reaction force and displacement in the U3 direction was taken from the reference point on the center cylinder, and exported to Excel. The load vs. displacement was then plotted on the same graph as the experimental data. (See Figures 19-24)

Excluding the data for the 4 lamina specimens, the composite layup was always stiffer than the isotropic. Also as seen in Table 4, the estimated stiffness for the composite layup was generally more accurate with respect to the experimental data than the estimated stiffness from isotropy. The reason for isotropy being less accurate and stiff was probably because it assumes that all the lamina within the composite have roughly the same stiffness. However, in reality, some of the lamina s were weaker in tension because they were oriented at 45°. This resulted in an overall lesser value of Young’s moduli for the entire composite in tension. As seen in Table 4 and Figures 19-24, there was still some error using the composite lamina however it was not as severe as the isotropic. This error was more than likely due to a multitude of subtle differences between specimens such as defects, delamination, and warped weaves. Based on this data though, the isotropic assumption was wrong. Instead, lamina properties should have been used. In fact the tests prove the accuracy of the lamina properties.
5 Conclusion

Fiber elements could not be directly imported into Abaqus from the design programs they were originally created in. This meant that in order to complete analyze the design of the multifunctional house, an alternative method of implementation had to be developed.

Before this was done though the current Abaqus model with no fiber elements needed boundary conditions, loads and material properties applied to it. It was here that the assumption of quasi-isotropy was made because of the chosen orientation of the lamina. Then mesh convergence tests and parametric studies were performed on the model.

After this general analysis, a Python code was created that would read points from a text file, and create a fiber element out of individual wire features based on the read in points. With this working code, stress redistribution was attempted to be observed, by applying a few fiber elements to the model. However, due to the sheer size of house, there was no significant stress redistribution.

The final part of this project was testing the validity of the assumptions made for the composite layup. This was done by performing tension and bending tests and comparing the experimental data to Abaqus model’s of each specimen with either the isotropic section or the composite layup. As a result of these tests, the original assumption made about quasi-isotropy was wrong because it did not take into account the weakness of lamina at other orientations.

Despite the quasi-isotropy assumption being wrong, the most important result from this investigation was in fact the code. The next step would be to create multiple fiber elements based on actual fiber elements from the design. With more wires, more meaningful analysis can be performed on the house. Also with the assumption of quasi-isotropy disproven, lamina properties should be used again meaning that some of the parametric studies would need to performed again in order to reflect the new properties. Finally, more composite specimens should be created in order to obtain more consistent results for material properties and a better understanding of those materials. Hopefully as a long term result of this study, both departments will increase their collaboration with each other in order to pursue more innovative designs.

Acknowledgements
Thanks to Kevin Maxwell for the assistance in Abaqus and Python. Antonty Zacharatos assisted in the tension and bending tests. “The Ranch” at Riverside Campus assisted in the fabrication of composites.
Appendix A

Artistic rendering of House with fiber elements

Figure 25 Artistic render of Design Provided by Gabriel Esquivel

Comparison of Original Model and New Model

Figure 26 Contour Plot for Original Abaqus Model
**Figure 27** Contour Plot for Improved Abaqus Model

*Screenshot of Boundary Conditions*

**Figure 28** Screenshot of Boundary Condition for Abaqus House Model
Screenshots of surfaces for loads

**Figure 29** Screenshot of surface for which the vertical wind load is applied

**Figure 30** Screenshot of the surface for which the lateral wind is applied
Figure 31 Screenshot of the surface for which the snow load is applied

Appendix B

Screenshot of Non-flat plate

Figure 32 Screenshot of a non-flat plate with fiber elements
from part import *
from material import *
from section import *
from assembly import *
from step import *
from interaction import *
from load import *
from mesh import *
from job import *
from sketch import *
from visualization import *
from connectorBehavior import *

part_name='Top1_metric'
surface_instance_name='surface_metric-1'

file_name='Top1.txt'
def linecount(filename):
    with open(filename) as f:
        for i, l in enumerate(f):
            pass
    return i + 1

lines=linecount(file_name)
infile= open(file_name,'r')
table=[]
for x in range (0,lines):
    #data=infile.readline().split(',
    data=infile.readline().split()
    #print data
    for i in range (0,3):
        data[i]=float(data[i])
    table.append(data)
infile.close()

mdb.models['Model-1'].Part(dimensionality=THREE_D, name=part_name, type=DEFORMABLE_BODY)
mdb.models['Model-1'].parts[part_name].ReferencePoint(point=(0.0, 0.0, 0.0))

for i in range (0,lines-1):
    previousx= table[i][0]
    previousy= table[i][1]
    previousz= table[i][2]
    currentx= table[i+1][0]
    currenty= table[i+1][1]
    currentz= table[i+1][2]
    mdb.models['Model-1'].parts[part_name].WirePolyLine(mergeWire=ON, meshable=ON, points=(
((previousx, previousy, previousz), (currentx, currenty, currentz))

set_name='Wire-1-Set-' + str(i+1)
mdb.models['Model-1'].parts[part_name].Set(edges=
    mdb.models['Model-1'].parts[part_name].edges.getSequenceFromMask((
        '[#f ]', ), ), name=set_name)

mdb.models['Model-1'].parts[part_name].seedEdgeByNumber(constraint=FINER
    , edges=
    mdb.models['Model-1'].parts[part_name].edges.getSequenceFromMask((
        '[#f ]', ), ), number=10)

mdb.models['Model-1'].parts[part_name].setElementType(elemTypes=(
    ElemType(elemCode=T3D2, elemLibrary=STANDARD), ), regions=(
    mdb.models['Model-1'].parts[part_name].sets[set_name] )
)

previous_set='Wire-1-Set-1'
merge_set='merge-set'
for x in range (0, lines-2):
    current_set='Wire-1-Set-' + str(x+2)
    mdb.models['Model-1'].parts[part_name].SetByMerge(name=merge_set, sets=(
        mdb.models['Model-1'].parts[part_name].sets[previous_set],
        mdb.models['Model-1'].parts[part_name].sets[current_set])
    )
    previous_set=merge_set
mdb.models['Model-1'].parts[part_name].generateMesh()

mdb.models['Model-1'].parts[part_name].SectionAssignment(offset=0.0,
    offsetField='', offsetType=MIDDLE_SURFACE, region=
    mdb.models['Model-1'].parts[part_name].sets[merge-set],
    sectionName='fiber', thicknessAssignment=FROM_SECTION)

for x in range (0, lines-1):
    delete_set='Wire-1-Set-' + str(x+1)
    del mdb.models['Model-1'].parts[part_name].sets[delete_set]

instance_name=part_name+'-1'
constraint_name='constraint-'+instance_name

mdb.models['Model-1'].rootAssembly.Instance(dependent=ON, name=instance_name
    , part=mdb.models['Model-1'].parts[part_name])

mdb.models['Model-1'].Tie(adjust=ON, master=
    mdb.models['Model-1'].rootAssembly.instances[surface_instance_name].surfaces['entire_surface']
    , name=instance_name, positionToleranceMethod=COMPUTED, slave=
    mdb.models['Model-1'].rootAssembly.instances[instance_name].sets['merge-set']
    , thickness=ON, tieRotations=ON)
Properties for Steel A36

<table>
<thead>
<tr>
<th>Material Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s Modulus (GPa)</td>
<td>207</td>
</tr>
<tr>
<td>Poisson’s Ratio</td>
<td>0.3</td>
</tr>
<tr>
<td>Density (kg/m$^3$)</td>
<td>7850</td>
</tr>
</tbody>
</table>

Appendix C

Picture of 2x2 twill E-glass

![Image of 2x2 twill E-glass weave]

**Figure 33** Picture of 2x2 twill E-glass weave

Picture of E-glass/Epoxy Specimen

![Image of E-glass/Epoxy test specimen]

**Figure 34** Picture of E-glass/Epoxy test specimen
Abaqus Model of 3-point Bending

![Figure 35 Bending test assembly in Abaqus](image)

References


