GFR15 Monocoque FEA Report

Sam Conklin

Gabe Gray, Project Advisor

0. Table of Contents

- 0. Table of Contents
- 1. Project Description
  - 1.1 Introduction
  - 1.2 Rules and Constraint Analysis
  - 1.3 Requirements
- 2. Current State Analysis and Benchmarking
  - 2.1 Current State Analysis
  - 2.2 Benchmarking
- 3. Design Analysis
  - 3.1 Simulating Physical Testing
  - 3.2 Various Modeling Techniques
    - 3.2.1 2D Laminate Model
    - 3.2.2 3D Core Elements with 2D Faces Sheets
    - 3.2.3 3D Elements to Model Core and Face Sheets
  - 3.3 Definition of Layup
  - 3.4 Meshing
  - 3.5 Definition of Layup
  - 3.6 Design Overview
- 4. Design Selected
  - 4.1 Rationale for Selection
    - 4.1.2 Threats
  - 4.2 Technical Specification
    - 4.2.1 Meshing and element type
    - 4.2.2 Suspension links
    - 4.2.3 Hardpoint Connections
  - 4.2.4 Roll Hoop Integration
- 5. Implementation
  - 5.1 Documentation
  - 5.2 Element Specification
    - 5.2.1 Element Orientation
    - 5.2.2 Element Direction
  - 5.3 Roll Hoop Connection Points
  - 5.4 Material Failure

5.4.1 Max Strain
6. Application to the Alternate Frame Rules
6.1 AF 4.1 Main Roll Hoop, Bracing and Bracing Supports
6.2 AF 4.2 Front Roll Hoop
7. Conclusion
7.1 Future Work
7.2 Personal Reflections
8. Works Cited

1. Project Description

1.1 Introduction

Global Formula Racing is a world class team comprised of students from Oregon State University (OSU) and Duale Hochschule Baden Wurtternberg (DHBW). Each year, the team cooperatively designs and manufactures two high-performance race cars to compete in the Formula SAE (FSAE) and Formula Student (FS) design series. Both cars share the same chassis design, however, one is powered by a combustion engine (cCar) and the other utilizes a fully electric powertrain (eCar). GFR competes with both cars in events in Europe and the United States where the teams are scored based on ingenuity of the design, quality of the final product, team member knowledge of the car, and performance in a variety of dynamic events which test the car's abilities.

Figure 1: 2013 Formula Student Germany

Many technical developments have been made since OSU's first entry in 2004 which have built the foundation for the incredible success of GFR in recent years. In 2009, the team developed a hybrid chassis utilizing a mix of steel tubing and carbon fiber reinforced plastic (CFRP) [1]. This replaced the traditional steel tube frame constructed for all prior cars. In the years following, GFR evolved to a fully carbon monocoque frame constructed with honeycomb cores and carbon fiber laminates.

While there are many benefits associated with using a composite chassis, the design process is rigorous and requires a deep knowledge of the anisotropic materials. In addition to material selection and advanced manufacturing practices, a sufficient method of analyzing the structural properties must also be developed. The goal of this report is to mature GFR's capabilities in Finite Element Analysis (FEA) of composite materials in order to design a lighter chassis with known stiffness characteristics. Designing with FEA also permits the analysis of more complex designs and adherence to the Alternate Frame rules provided by FSAE. These are an optional set of rules which allow a team to validate the structural integrity of the chassis using FEA rather than the customary Structural Equivalency Spreadsheet (SES) [SES Tutorial].

After assessing various finite element analysis programs, Siemens FEMAP was selected for its high quality composites package which is widely used in the aerospace industry. This investigation of composite Finite Element Analysis presents a valuable opportunity to advance the design of GFR's chassis.

1.2 Rules and Constraint Analysis

FSAE has developed rules that guide the design of structurally sound chassis to protect the driver in the event of a crash. This has traditionally been guided by the Structural Equivalency Spreadsheet which mandates that competition chassis must display equal or greater strength than a baseline steel tube chassis. The baseline chassis is broken up into critical zones and for a composite monocoque, a representative test panel of each zone must be compared with physical testing to the steel tubes it is replacing.

In 2011, the Alternate Frame (AF) rules were presented by FSAE which is an optional rule set superseding the SES that each team can elect to follow. The AF rules guide the FEA of a chassis and mandate specific analysis conditions. The results of the analyses are compiled and submitted on a single form called the Structural Requirements Certification Form (SRCF). An additional step mandated by the AF rules is the submission of an Alternate Frame Rules - Notice of Intent which requires finite element analysis of a sample frame as a qualification test of the team's FEA capabilities.

The AF rules require that a NASTRAN based FEA package is used. NASTRAN (NASA Structural Analysis System) was developed by NASA for the Saturn V rocket and is the required solver deck for all NASA funded projects [2]. FEMAP is a program that facilitates model input to the NASTRAN deck and provides visualization and analysis of the results. At FSAE Michigan 2014, only seven out of the 120 registered teams elected to follow the AF rules [Gabe Gray via a conversation with Bill Riley, Chief Design Judge].

Similar to the SES, The Alternate Frame rules mandate a minimum structural integrity of the submitted chassis design. FSAE specifies eight loading cases with load directions and magnitudes, load applications points, and a maximum deflection the frame can exhibit. Additionally the frame must not exhibit any modes of failure under each load case. The specific conditions can be found in Article 2 of the AF rules section of the FSAE rulebook [3]. These requirements are used for the analysis of the final frame design as well as the qualification frame submitted with the Notice of Intent. For teams designing a composite chassis, two of the tubes on each side (four total) must be replaced with composite panels (indicated with red arrows).

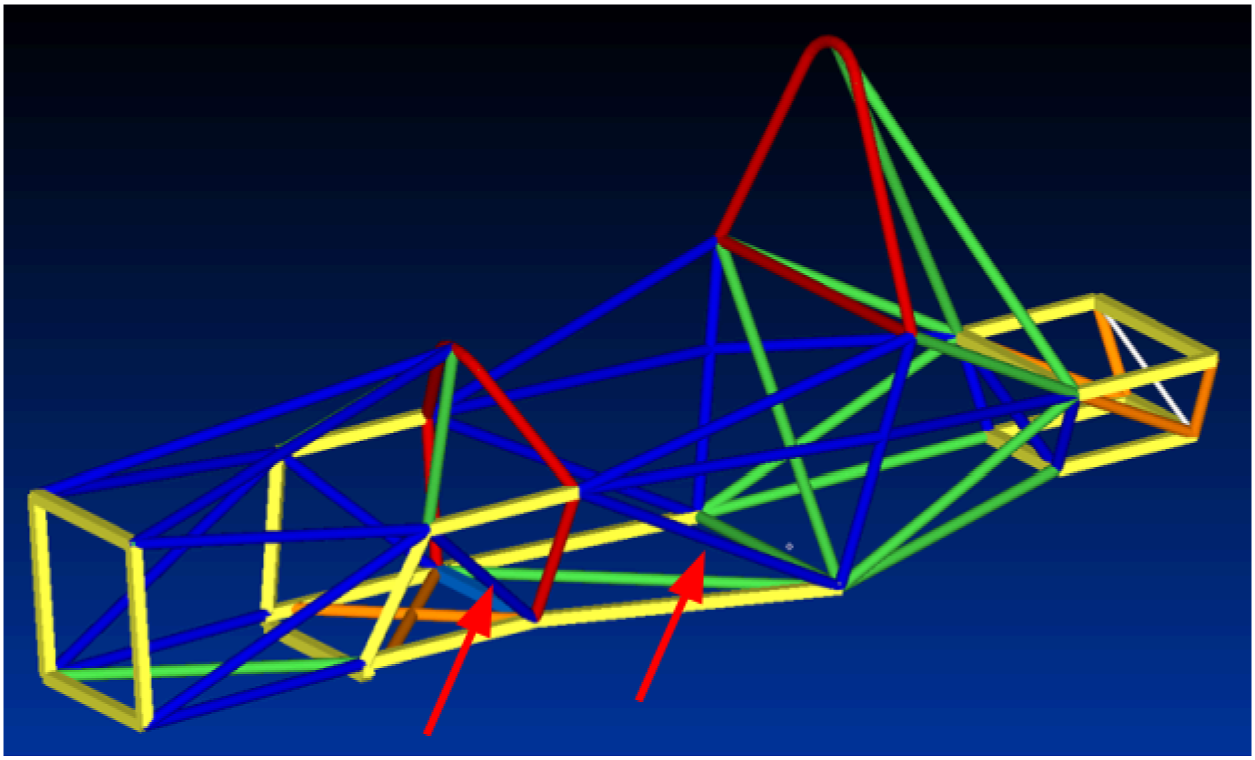

Figure 2: FSAE Notice of Intent Sample Frame [7]

When designing a chassis for a race car, an in-depth understanding of the rule book is mandatory. Part T-General Technical Requirements covers most of the structural requirements under the traditional SES ruleset. When following the AF rules, Section T3 is superseded by section AF. The AF rules still require some physical testing to validate the composite material characteristics. These tests are similar to the equivalency tests outlined in sections T3.28-T3.34 and a three point bend test is required for each laminate used on the chassis. The main elements of the AF rules are contained in section AF4 and define the eight loading cases required for the chassis analysis. An example of a loading case is shown below:

AF4.2 Front Roll Hoop

AF4.2.1 Load Applied: Fx=6.0 kN, Fy=5.0kN, Fz=-.9kN

AF4.2.2 Application Point: Top of Front Roll Hoop

AF4.2.3	Boundary Condition: Fixed displacement (x,y,z) but not rotation of the bottom nodes of both sides of the front and main roll hoops.
AF2.3.2	Max Allowable Deflection: 25mm
AF4.2.5	Failure must not occur anywhere in structure

By following the AF rules, additional points could be possible in the static events in two ways. First, adherence to the Alternate Frame rules would be a strong talking point in the Design Report and during the Design Event. At 2014 FSAE Michigan, GFR placed fourth in the Design Event with 129/150 total possible points [10]. This leaves a potential 21 points for the taking. Second, reliable FEA of the chassis could allow a reduction in unnecessary material which directly translates to cost savings. The Cost Report is traditionally an event that GFR does not excel in due to the high cost of the car and any reduction in expensive composite materials is helpful. Third, completing the SES is a time consuming process that requires numerous physical tests. Recognizing the limited time and resources available to GFR, it would be beneficial if adhering to the AF rules can reduce the man hours required in qualifying the frame design.

Electing to follow the AF rules could translate into point gains during the Dynamic Events as well. Since the Alternate Frame rules offer more design freedom, chassis weight could be reduced by applying only the necessary strength and stiffness where it is required. The SES specifies generic "zones" on the chassis and requires that each zone is made with a uniform layup schedule, even if strength is only needed in a small area. If the AF rules allow layups with non uniform ply coverage, the unnecessary reinforcement can be removed. One specific example can be found in the SES test of the Side Impact Structure. A minimum load must be reached before panel failure in a three point bend test of a representative panel. The results and setup of this test for the 2015 chassis can be viewed on the testing page. While the GFR panel met the minimum load requirement, it is likely that the total weight of the panel could have been reduced by removing plies from the ends of the beam and adding them to the middle of the panel where the moment is the greatest. The loading case AF 4.3.2 defines the analysis requirements for the side impact structure.

A reduction in weight improves cornering and acceleration which is beneficial in every event. Based on the work of Trevor Takaro equating design parameters to points, a reduction of one kilogram from the total weight of the car can be equated to 1.50 points at FSAE and 1.97 points at FSG. Additionally, with the proper utilization of FEA, chassis stiffness can be increased to allow the suspension system to function as designed.

Static Events:		
	Presentation	75
	Engineering Design	150
	Cost Analysis	100
Dynamic Events		
	Acceleration	75
	Skid-Pad	50
	Autocross	150
	Efficiency	100
	Endurance	300
Total Points		1,000

Figure 3: A 1.4.1 FSAE Points Summary (Formula Student competitions may differ) [3]

1.3 Requirements

This report, in conjunction with the FEMAP tutorials and other GFR resources, should succinctly detail the steps and knowledge required to pass the AF Rules, submit the SRCF, and accurately analyze a composite monocoque chassis for competition in FSAE events. If successful, future GFR teams should not have to allocate excessive resources to the analysis and students learning the process should be brought up to speed quickly.

Collaboration between all team members involved in the composites design and manufacturing will be essential for the success of this project. In the immediate future, the FEMAP tutorials will need to be finished by Ammar Al-Habsi to facilitate quick transfer of knowledge to GFR students in the upcoming years. Information from Paul Weitzman regarding composite properties, testing, and manufacturing will be crucial, and additional support and guidance from Gabe Gray will make this project possible. Consultation with our friends at Wasatch Composite Analysis and Predictive Engineering/Applied CAx will also be necessary as they professionally analyze systems for other companies and have offered to share their expertise. Predictive Engineering/Applied CAx is based in Portland and has been very generous in supporting our efforts. A positive relationship should be maintained in this and future years as they are able to provide high level recommendations. This is an exceptional opportunity to push the limits of GFR's capabilities.

2. Current State Analysis and Benchmarking

2.1 Current State Analysis

Robert Story completed a Master's thesis [5] in 2014 detailing a computational analysis of sandwich panels and determination of structural properties for the SES. This work will help provide the foundation of composites knowledge for moving over to the AF rule set. It is recommended that any future students pursuing this project read Robert Story's Thesis.

Adoption of Alternate Frame Rules

	Positive	Negative
Internal Factors	**Strengths** • Reduction in physical testing of laminates • Permits analysis of more chassis designs	**Weaknesses** • Increased learning requirements for new team members • Potential increase in time for completion
External Factors	**Opportunities** • Reduced chassis weight • Improved chassis stiffness • High level design discussion opportunity	**Threats** • Additional qualification step before competition

Three point bend tests are the focus of the thesis and the method developed predicts stiffness and failure characteristics of sandwich panels by combining classical laminate theory to classify the composite skin and shear deformation properties of the core material. A typical sandwich laminate consists of two, multi-ply, high stiffness composite laminates on either side of a lightweight, low stiffness core. In past chassis Hexcel Nomex and Kevlar honeycomb cores were used (Hexcel Core Data Sheets), however, on the 2015 chassis an aluminum honeycomb core is being evaluated due to its higher specific strength and stiffness.

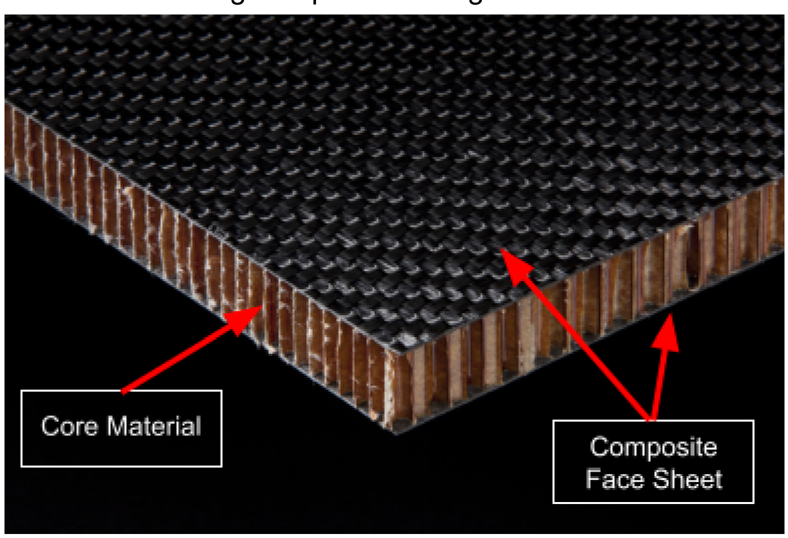

Figure 3: Example of Sandwich Panel Construction [6]

For analysis of composite materials it is helpful to establish a coordinate system to describe the orientation and location of laminate elements. The figure below demonstrates the standard convention for naming the directions of typical unidirectional tape (a) and 0°-90° woven fabric (b). Unidirectional material, commonly referred to as tape, consists of fibers running in only one direction. The term fabric indicates woven fibers laying in varying directions. The warp fibers (1) lay perpendicular to the stock roll and extend the entire length of the material while the fill fibers (2), also referred to as the weft, run parallel to the roll and only extend the width of the fabric.

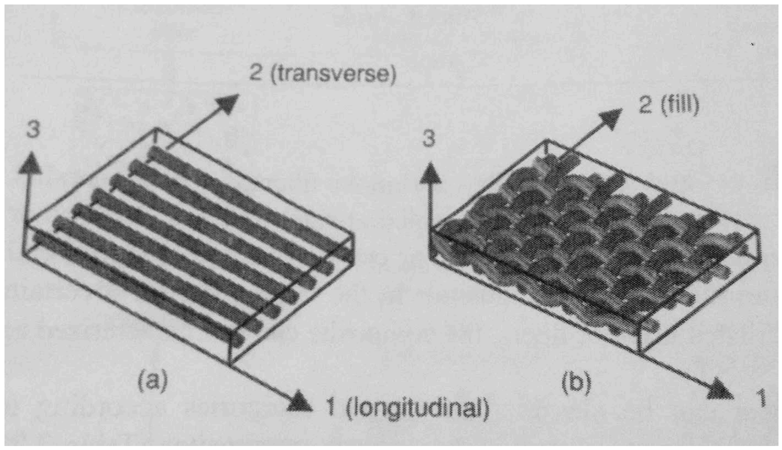

Figure 4: Convention for Composite Coordinates [8]

When lamina (one single ply of material) are combined into a laminate (multiple plies stacked together), they are often applied at varying angles to create a balanced laminate capable of providing stiffness and strength in multiple directions. To describe the orientation of a single ply, the angle θ is taken between the 1 direction of the ply and the x direction of the laminate. This convention is demonstrated in the figure below.

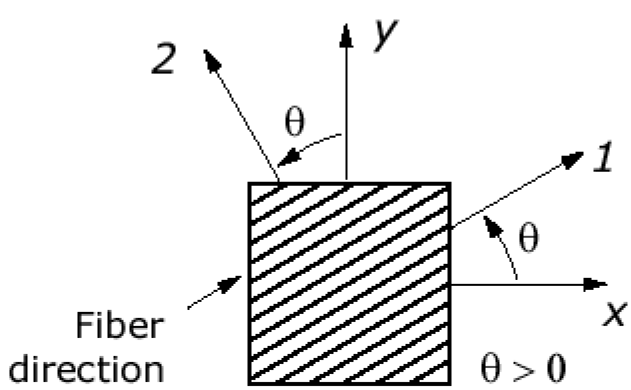

Figure 5: Rotation of ply orientation from standard coordinates [9]

The table below outlines the useful variable names and properties common to a 3D composites analysis. The table was adapted from Robert Story's thesis and Daniel, Ishai (book).

Necessary Material Properties for Analysis				
			Required for Method	
Material	Symbol	Description	Rules	GFR
Lamina	E_1	Young's Modulus, 1 Direction	✔	✔
Lamina	E_2	Young's Modulus, 2 Direction	✔	✔
Lamina	E_3	Young's Modulus, 3 Direction		?
Lamina	v_{12}	Major In-Plane Poisson's Ratio	✔	✔
Lamina	v_{23}	Out of Plane Poisson's Ratio		?
Lamina	v_{13}	Out of Plane Poisson's Ratio		?
Lamina	G_{12}	In Plane Shear Modulus	✔	✔
Lamina	G_{23}	Out of Plane Shear Modulus		?
Lamina	G_{13}	Out of Plane Shear Modulus		?
Lamina	F_{1t}	Tensile Strength, 1 Direction	✔	✔
Lamina	F_{1c}	Compressive Strength, 2 Direction	✔	✔
Lamina	F_{2t}	Tensile Strength, 2 Direction	✔	✔
Lamina	F_{2c}	Compressive Strength, 2 Direction	✔	✔
Lamina	F_{3t}	Tensile Strength, 3 Direction		?
Lamina	F_{3c}	Compressive Strength, 3 Direction		?
Lamina	F_6	In-Plane Shear Strength		✔
Lamina	F_4	Out of Plane Shear Strength		?
Lamina	F_5	Out of Plane Shear Strength		?
Core	G_L	Shear Modulus, L Direction		✔
Core	G_W	Shear Modulus, W Direction		✔
Core	F_L	Shear Strength, L Direction		✔
Core	F_W	Shear Strength, W Direction		✔
Core	F_C	Compressive Strength		✔

Table 1: Necessary Material Properties for Composite Analysis

In Robert Story's thesis, three different methods of characterizing sandwich panels are compared to experimental results from 3-point bending tests. The three methods are summarized here. The first method is derived from standard beam bending equations combining the properties of the facesheet and the core material. This is the method used by the

FSAE in the composites section of the Structural Equivalency Spreadsheet. The derivation can be viewed in Section 2.3.1 of Story's thesis and the final equation simplifies to:

$$\frac{W}{\Delta} = 24\frac{E_f btd^2}{L^3} \qquad \text{[Eq. 2.1]}$$

Where

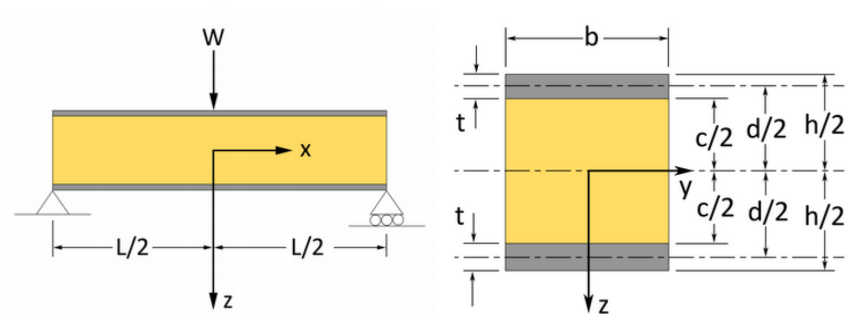

Figure 10: Beam Deflection Diagram [5]

$$W = applied force$$
$$\Delta = Deflection\ at\ center\ of beam$$
$$E_f = Effective\ facesheet\ stiffness$$

The method for determining facesheet stiffness is well outlined in Section 2.3.1 of the thesis. To summarize the approach, a stiffness matrix is generated for each ply relative to the global coordinate system (x,y,z) and then a summation of the oriented ply stiffnesses is completed with consideration taken for the distance from the ply to the midplane of the laminate. This method was only able to make a marginal prediction of the stiffness and the results became less reliable as panel stiffness increased.

To improve the accuracy, Story amended this method by adding a term that accounted for the shear deflection of the core material. This shear deformation term can be directly added to the original beam bending equation, resulting in the following equation:

$$\frac{W}{\Delta} = 24\frac{E_f btd^2}{L^3} + 4\frac{G_c bh}{L} \qquad \text{[Eq. 2.2]}$$

Story named this the GFR method and it provided a much better prediction of the sandwich panel's stiffness.

The third method analyzed in the thesis was the CATIA FEA module. This method was the most commonly used tool at the time of the report. In the simulation the panel was simplified to a 2D model with linear quad elements. A 9.525mm mesh size was decided upon as it was one half the width of the load application foot.

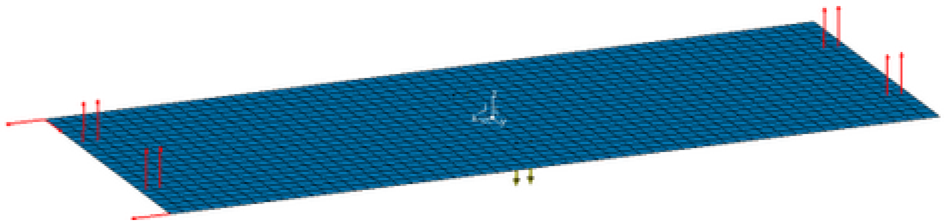

Figure 11: CATIA mesh of the 3-point bending test [5]

The ability to forecast failure is just as, if not more, important than determining laminate stiffness. All three methods relied on the Max Normal Stress failure criterion for facesheet failure. However, the GFR method and associated Matlab code created with the logic does check for core shear failure and core compression failure. It was duly noted that another potential failure mode is intra-cell buckling. This is not a significant concern for panels using a Nomex honeycomb as the cells are of relatively small size, however, the aluminum honeycomb core being evaluated this year has a larger cell size and may pose a risk of intra-cell buckling. Composites failure is an integral part of successfully completing the AF rules. Each load case specifies a maximum deflection along with the requirement that the structure can not fail in any mode. Potential modes of failure listed by the AF Rules Committee include Von Mises, Tsai-Wu, Eigenvalue buckling, and column buckling. This is not an inclusive list and an overview of composite failure theories can be found beginning on page 26 in this Composites FEA Presentation from industry. A very approachable but deeper understanding of failure theories can be explored in chapter 6 of Daniel, Ishai [5].

2.2 Benchmarking

Currently, GFR does not have a tool that can effectively run FEA on the entire chassis. Instead, each panel is designed and analyzed individually, then checked against the structural requirements established by the SES. This allows the chassis to be certified for FSAE/FS events, but FEA is desired for all of the reasons mentioned earlier in this report. Again, the currently available methods are CATIA FEA, the SES equations, and the matlab code designed by Robert Story. Each of these methods can offer a certain level of accuracy as benchmarked in Robert Story's Thesis and are depicted in Figure 12 and Figure 13 above.

The accuracy of the three methods were compared by determining the absolute error between the predicted stiffness and the experimentally confirmed stiffness of the panel. The results are displayed in Figure 12 below.

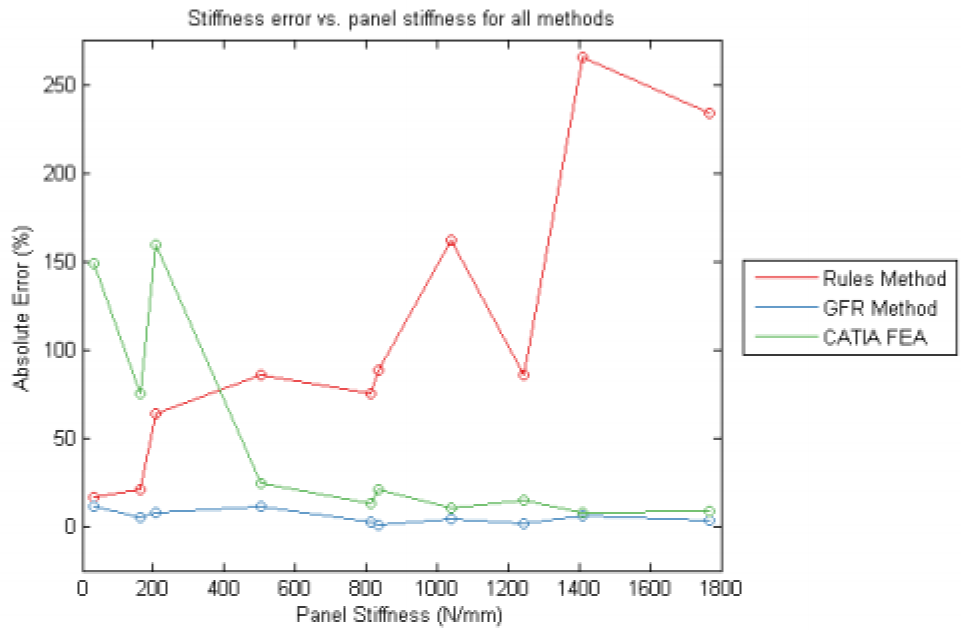

Figure 12: Stiffness Error vs. Panel Stiffness[5]

The rules method provided an inconsistent prediction that deteriorated with increasing panel stiffness CATIA demonstrated the opposite by starting off with an absolute error of 150% and finally converging to an error closer to 12% once the panel stiffness exceeded 500 N/mm. The GFR method displayed the most reliable results in this test and all predictions were within 10% of the true stiffness and converged to 5%.

The thesis also compares the prediction accuracy of ultimate strength between the three methods. It was determined that CATIA was a poor predictor of failure at any stiffness, the rules method slightly better and the GFR method the best. For panels with stiffness above 250 N/mm, the GFR method was able to predict failure within at least 22% and as close as 1.6%.

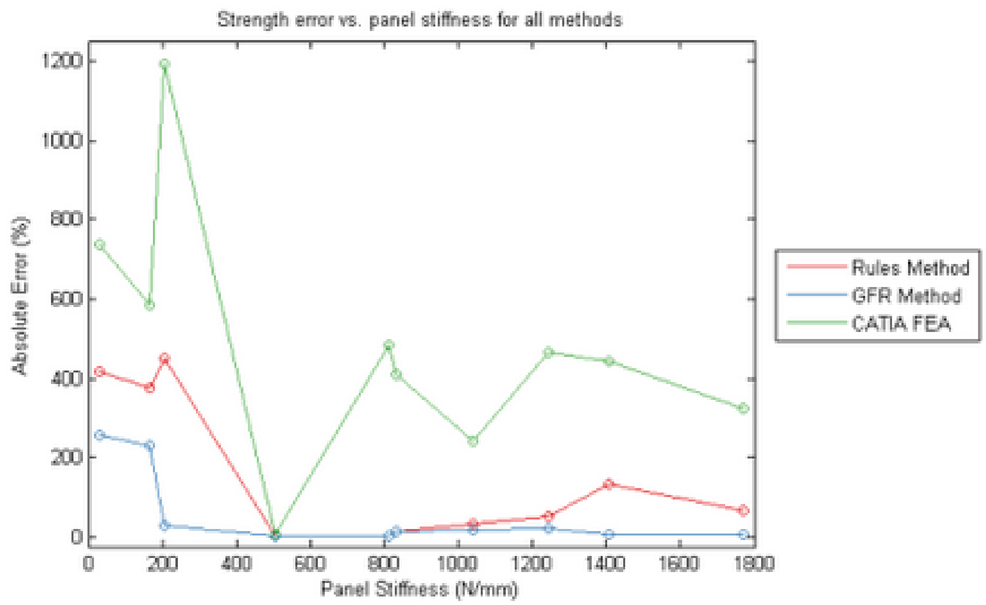

Figure 13: Strength Error vs. Panel Stiffness[5]

Additionally, the work that Ammar Al-Habsi completed in the previous term has provided a base level assessment of various methods for simulating simple geometry with FEMAP. In the FEMAP analyses that have been run, the most consistent problem is setting up the constraints to accurately model the real world conditions. The physical test method employed by Robert Story utilizes pivoting supports to allow free rotation of the test specimen as recommended by ASTM standard C393 [11]. This complicates the analysis as the rotation of the supports adds to the total translation of the sample. This error can be seen in Robert Story's CATIA analysis. At the end of the panel the simulation displays a secondary, negative, curvature where it is constrained horizontally. This is not the case in the physical test and the panel should display no curvature beyond the support point. These results are displayed in Figures 14 and 15 below. The blue arrow in Figure 15 indicates the point of inflection of this secondary curvature.

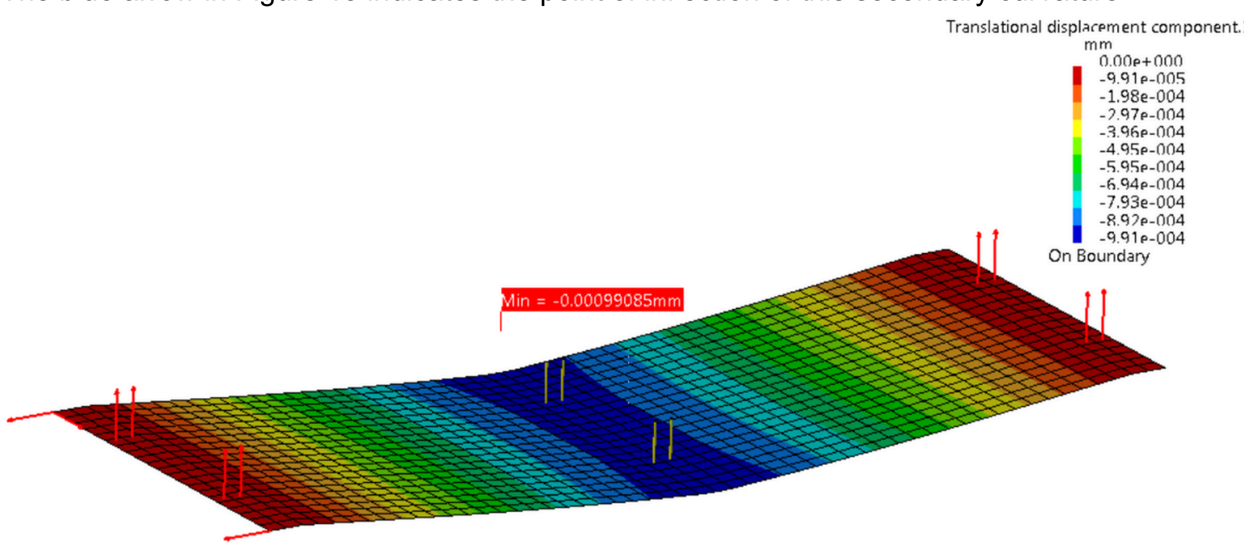

Figure 14: Z direction translational displacement, GF2013.BB.03 [5]

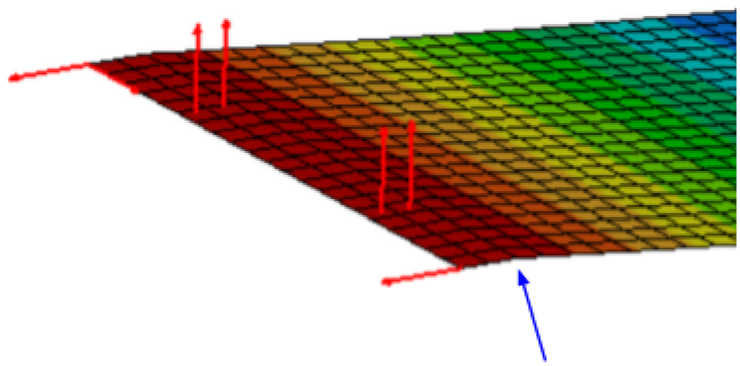

Figure 15: Detail view of Figure 14 showing curvature inflection point

It is critical in all simulations to perform visual and mathematical verifications of the results. For complicated simulations where a mathematical check is unrealistic, a good practice is to build a simple model with the same element structure and verify the analysis is behaving appropriately.

3. Design Analysis

FEMAP is an expansive software package capable of modeling complicated reactions of forms and systems. Recognizing the limited resources available to GFR and the fast design cycle of the Formula Student series, future members of GFR will probably not have the luxury of becoming experts in FEMAP. Therefore, consideration must be given to each aspect of Finite Element Analysis to determine how it benefits the team and decide what our study of FEMAP should focus on, concentrating on what is absolutely necessary for the AF Rules and SRCF submission. This will determine where the largest competition point gains can occur within the scope of this project.

3.1 Simulating Physical Testing

An important part of developing a robust FEA method involves reproducing physical tests of representative structures. Tests that have been run by GFR include three point bend tests of sandwich panels and tensile tests of thin laminates. This is a logical starting place as the material tests represent a distilled loading case of simple structures and the lessons learned in these basic simulations provide a strong foundation for the higher levels of complexity that will be encountered in the future. Additionally, it is imperative that the method developed is validated against test data to provide confidence in the analysis results. That being said, the quote that “all models are wrong, but some models are useful” by the mathematician George

E.P. Box is valuable to keep in mind. Excessive time can be spent perfecting small details of a model but the end goal must be kept in focus and work prioritized to achieve that goal as quickly as possible. Realistic results must be achieved but only within a necessary range of accuracy

This aspect of developing our FEMAP capabilities is where most of the effort to date has been applied. Reasonable methods for modeling the three point bend tests and tensile tests have been developed and primary development has been focused on modeling connections at the supports and the loading foot. This work has not been completed as the deflection of the three point bend tests have only been simulated to within ±%20.

To better understand the dynamics of the physical test, the compressive modulus of the contact material was determined using the Instron. This material was placed under the loading foot and on top of the supports prior to 2015, and any compression would be measured additively with deflection of the panel. Results and test setup can be viewed on the testing page.

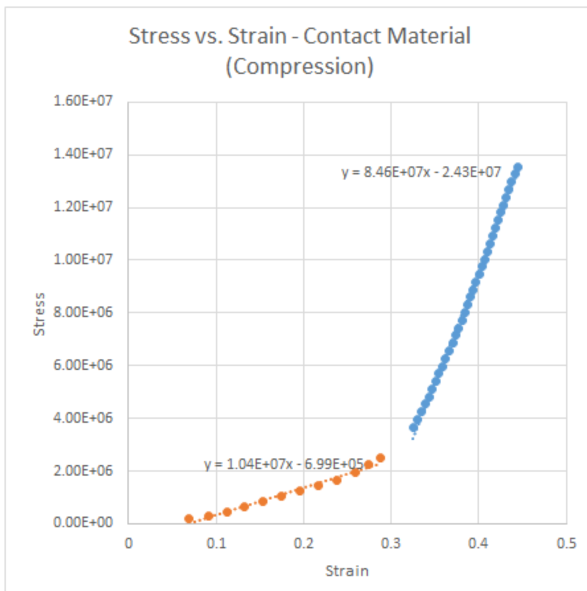

Figure 18: Stress Strain curve of contact material

Under the given loading conditions it is expected that the contact pads would compress a total of .56mm. The calculations can be found here. This equates to 5% of the total deflection and needs to be accounted for when analyzing tests prior to 2015.

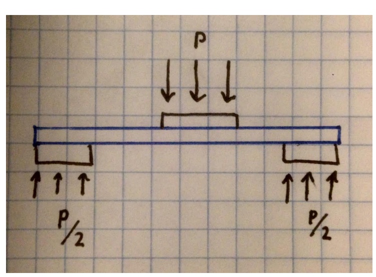

Figure 19: FBD of contact pads

The amount of rotation exhibited by the support bars was also calculated to determine its significance in the overall simulation. It was determined that this contributed to 1.7% of the total vertical deflection. It should be noted that this value is specific to this test and would change with different panel stiffnesses. The value is small but not negligible, and unlike the contact material mentioned earlier, this applies to all tests using this fixture. It is unlikely that the simulations will fall within 2% of the target value but it is useful information and opens up the possibility of removing this complication from the model and adding it to the results after the analysis. The angle of the supports, and consequently ends of the panel, was also determined which provides a useful inspection point for the analyses. This rotation of the supports is a function of the panel deflection, as the panel length is fixed and does not slide along the supports.

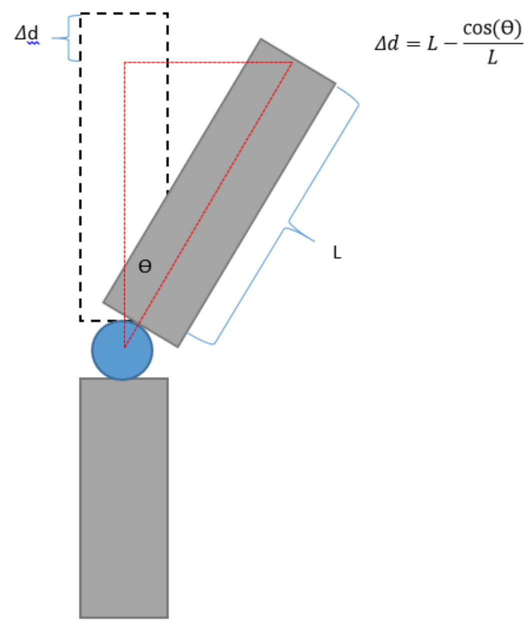

Figure 20: Calculation of deflection provided by rotating supports

In summary, the motion of the supports is a function of the panel deflection and the compression of the contact material is a function of the force applied. These were not accounted for in the “GFR method” but could easily be included. This information helps determine what aspects of the physical test make significant contributions to the final result. Future models can be simplified further and adjusted with a known value that compensates for the test fixture deflection. A future step in matching simulations to test data would be the testing and subsequently simulating metal tubing, as used in the roll hoops. This would benefit the analysis of the GFR frame as well as the analysis of the AF qualification frame which is primarily composed of steel tubes.

While matching physical tests is a valuable tool for validating FEA methods and practicing the wide array of FEMAP's available features, it does not provide intrinsic value on its own. The greatest potential offered by FEA is in analysis of complicated geometry and it should be used as such. That being said, future readers should remember that one of the most valuable tools when learning FEMAP is to run a bare bones simulation with a single element to verify their understanding of the mechanics of a feature.

3.2 Various Modeling Techniques

When developing a model for analysis, one of the most important considerations is determining the appropriate geometry to represent each structure. Simplifications can be made in the geometry to reduce the effort required to mesh, constrain, and analyze the model, however, the representation must be accurate enough to provide useful results. Although this consideration applies to structures of all types, this study of FEMAP will focus on structures made with tubing, plates, and composite sandwich panels for the chassis and may eventually include thin wall composite bodies to model aero components.

The following sections are a discussion of three potential methods that could be used to simulate composite sandwich panels. These are discussed in the "Composite Laminate Modeling" white paper released by Predictive Engineering. The various methods include a 2D representation, a combination of 2D plate elements and 3D solid elements, and a fully 3D solid model. The tradeoffs that must be considered include the effort required in modeling, effort required in meshing, accuracy of the results, and the computational cost of running each iteration. Similar to optimizing mesh size, where the mesh size is reduced until reaching a point of diminishing returns, an appropriate modeling technique can be selected by testing the various representations and determining the method of least complexity which also provides sufficient accuracy. This practice, as applied to mesh size is visualized in Figure 21 below.

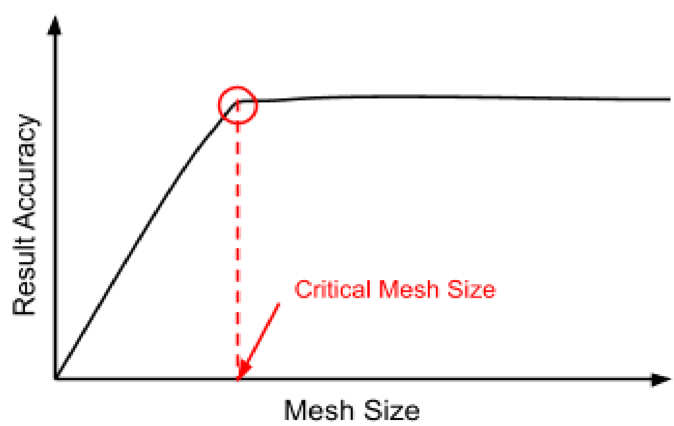

Figure 21: Visualization of Mesh Size vs. Result Accuracy from CATIA FEA Tutorial

3.2.1 2D Laminate Model

FEMAP has the ability to simplify a sandwich panel into a 2D surface which significantly simplifies the modeling and meshing of the structure. This is often sufficient for axially loaded laminates and can provide a quick analysis. Where the 2D plate elements break down is in out of plane loading and failure. FEMAP utilizes classical laminate theory to model 2D laminates which does not account for the through thickness properties of the laminate. The consequence of this simplification is the loss of stress, strain, and failure data in the direction normal to the laminate, as well as interlaminar shear failure.

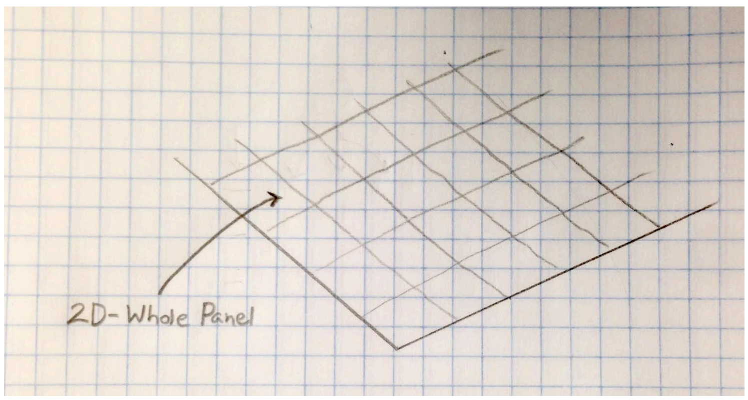

Figure 22: 2D Representation of Panel

3.2.2 3D Core Elements with 2D Faces Sheets

The next step to increase the accuracy of the simulation requires modeling the core as a 3D solid element while keeping the face sheets modeled as 2D laminate representations. This allows inclusion of the 3D mechanical properties of the core which permits simulation of through thickness deflection and shear. Care must be taken when modeling the laminate as both face sheets must be offset by one half the thickness of the laminate from the core [12]. When defining a composite material for analysis, a Layup Property Card must be generated for each unique laminate that defines the materials and ply orientations. If multiple face sheets are modeled, each face sheet must be defined by its own layup property card in order to collect analysis results for each ply.

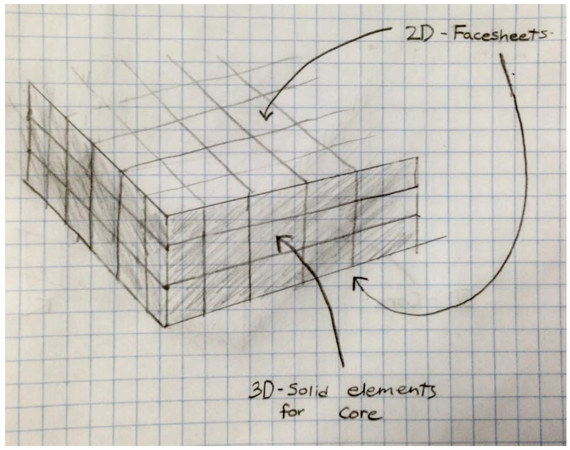

Figure 23: 3D Core, 2D Face Sheets

3.2.3 3D Elements to Model Core and Face Sheets

This final iteration models the true thickness of the face sheets in addition to the core. With this version the through thickness properties of the laminate can also be included, accounting for all constituents of the sandwich panel. With proper application this will provide the most representative results. When considering the element size for this method, keep in mind that a 3D element should never have an aspect ratio between any two edge lengths greater than 10 (element height: element width). The typically thin face sheets used in sandwich panels will force relatively small elements which increases meshing complexity and computational cost.

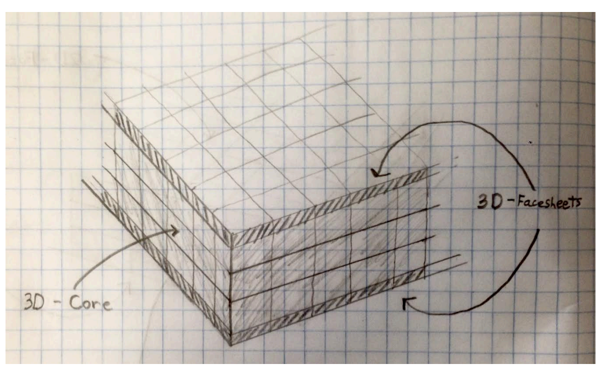

Figure 24: 3D Core and Face Sheets

3.3 Definition of Layup

One possible advantage that could be realized with advanced finite element analysis is the characterization of laminate behavior around hardpoints. This is significant because the current method of applying reinforcing plies around connection points relies heavily on experience and estimation. The current hardpoint reinforcements are shown in Figure 25 below, where LH is aligned with the hardpoint, which is aligned with the load exerted on the chassis This is an iteration of the previous hardpoint reinforcement, a circular patch, and significant analysis was done in Matlab to achieve this design, but composites FEA was not performed and physical testing has not been performed yet.

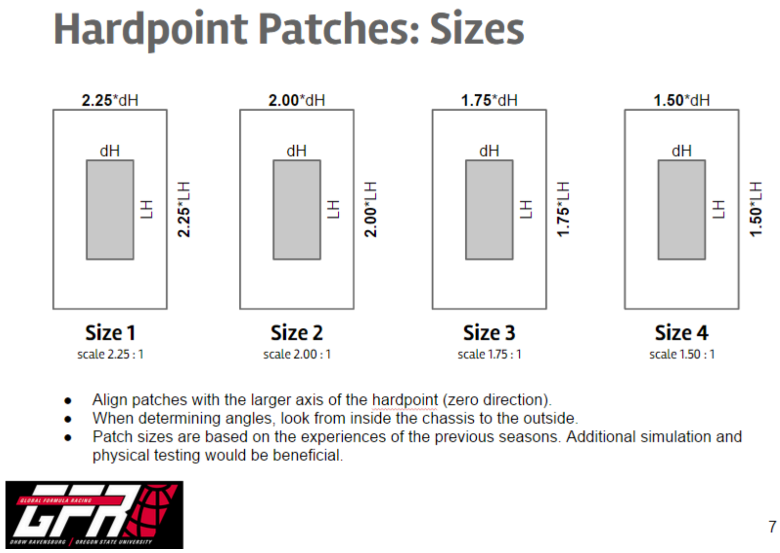

Figure 25: Hardpoint Patch Definitions from 2015 Ply Book

By successfully developing a method for defining the adjusted layup schedule around hardpoints, analysis could be conducted for each hardpoint by applying the expected loads and assessing if the reinforcement is excessive, appropriate, or insufficient. As the current design is based on empirical data from successful chassis, it is most likely that the patches are either appropriate or excessive. The 2015 chassis utilizes 31 hardpoints indicating significant room for weight reduction. Developing this proficiency would require research in defining layups for small sections and is deeply rooted in the modeling method used to simulate the entire chassis, be it a 2D, 3D, or a combination.

It should be recognized that the capabilities of FEMAP extend to isotropic material analysis as well. In a hardpoint assembly, the composite reinforcement represents only a fraction of the weight provided by the metal hardpoint itself. Simon Bettsheider completed a valuable study on hardpoint design that correlated the hardpoint exterior profile with the applied load using a MatLab script and is described in his report [14]. This study could be expanded upon using FEMAP to determine the reaction of the hardpoint and the surrounding laminate to determine the lightest weight design.

3.4 Meshing

In determining the value of electing to follow the AF rules and/or run a finite element analysis on future chassis, careful attention must be given to the necessary time and manpower.

In the work that has been done so far on the 2015 analysis, a significant amount of time has been dedicated to meshing the chassis. The current process involves meticulously meshing each section by hand which generates a robust, consistent mesh at the cost of skilled labor and delayed analysis. FEMAP offers a number of auto-meshing tools which should be evaluated to minimize necessary time. This could be tested by comparing the accuracy of results from each meshing technique, and should greatly increase the analysts efficiency

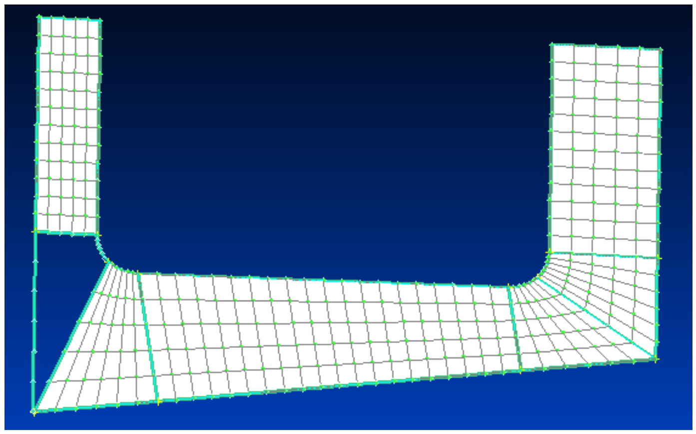

Figure 24: Image from the Chassis Meshing Tutorial

3.5 Definition of Layup

Another important step in developing a finite element model for a composite structure is the definition of layup elements. For sandwich panels, this is closely tied to the method selected in section 3.2 and can be accomplished in a variety of ways. Predictive Engineering's "Composite Laminate Modeling" white paper provides a useful overview of the various methods. Further research is needed on this topic as it holds significant potential to affect the analysis of the current chassis and more importantly, the design of future chassis.

3.6 Design Overview

Each of these design considerations play an important part in developing a successful chassis analysis but none are useful on their own. A holistic approach must be taken where each of these are studied to the point that they can provide useful results but do not take a disproportionate amount of time to develop. Once these decisions have been made, a productive course of action will be to apply the loads specified in the Alternate Frame rules to the 2015 chassis and determine areas where the chassis has been over- or under-built. With this information, recommendations can be made to help design a new layup schedule for the 2016 car using the same geometry and molds to produce a lighter, rules compliant frame, whether that be through the AF Rules or the SES.

4. Design Selected

4.1 Rationale for Selection

The knowledge gained in this study of the Alternate Frame rules has indicated that an AF rules submission is still a viable option worthy of further investigation. More work needs to be done as the team does not yet have the capability of successfully submitting a Structural Requirements Certification Form, but no obstacles have been encountered that would foreseeably terminate progress. A preliminary analysis of the 2015 chassis was generated and used to estimate a torsional rigidity value for the initial design report. This initial analysis, displayed below in Figure 25, was a major milestone and demonstrated some of the potential benefits of FEA. Additionally, the analysis acted as a proving ground for various modeling techniques that have been under development.

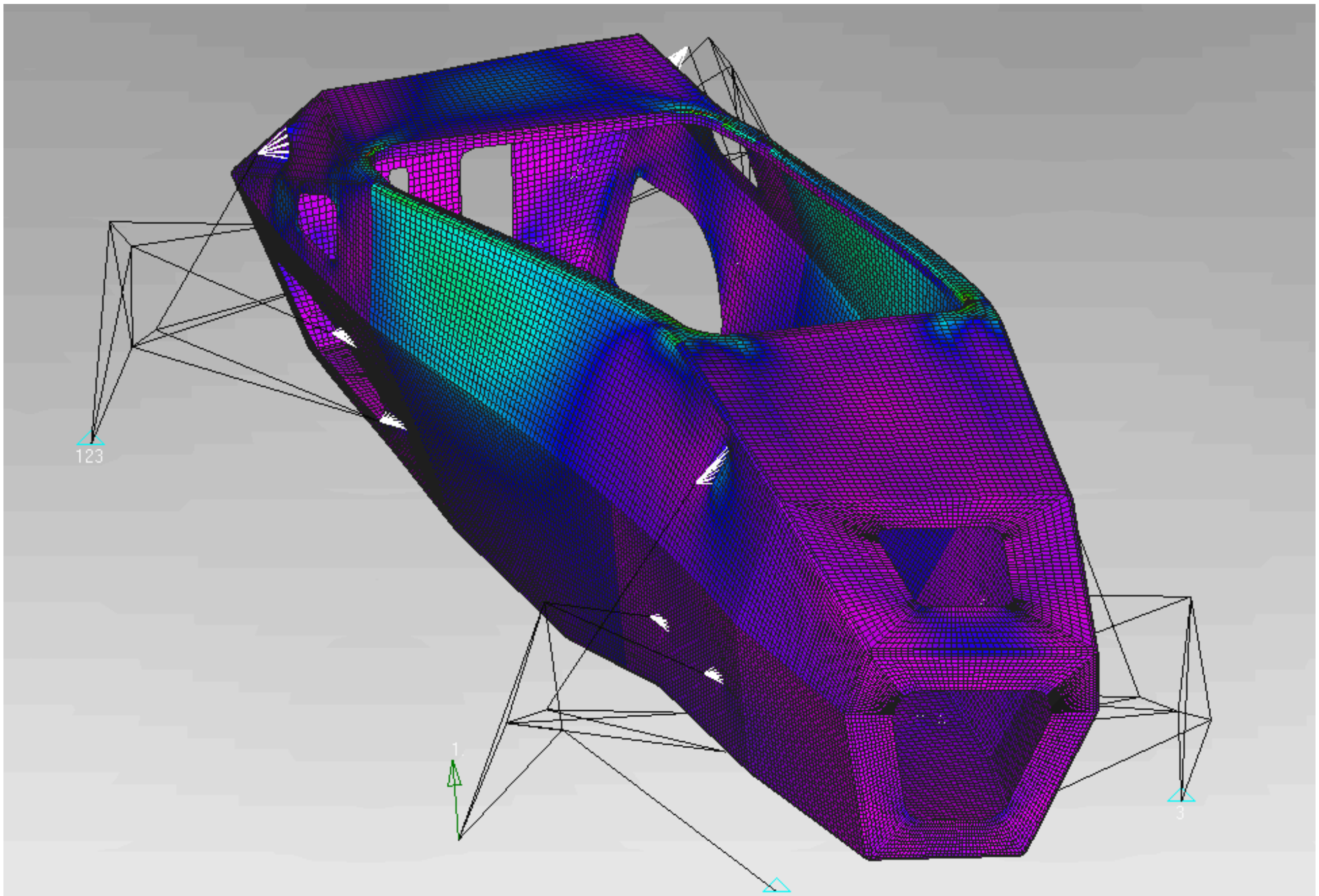

Figure 25: Initial Chassis Analysis for Torsional Rigidity

The Torsional Rigidity test indicated a chassis stiffness of 1608.3 Nm/deg which lies within an order of magnitude of accurate stiffness values. This was the goal of this exercise, while further learning, as well as a physical test, will be performed before the completion of this report.

Among many other lessons, one significant takeaway was the importance of specifying and checking the element orientation and material direction for the model. The contour map in Figure 25 above shows a discontinuity in the strain field at each end of the side impact panel. This was remedied by updating all elements orientations with to face outward using a FEMAP command.

4.1.1 Opportunities

A Finite Element Analysis of the chassis provides numerous benefits in the design phase. One of the main benefits is the ability to test a variety of laminate designs with no material costs. Once a functional model of the chassis has been meshed and constrained, the layup schedule can be easily changed in FEMAP. While the AF rules still require physical testing (discussed in next section) the knowledge gained in simulating three point bend tests can improve the estimated stiffness of panels and potentially reduce the number of failed tests due to under- (or over-) designed laminates.

While a final interpretation of the rules is still being developed, it appears that the AF rules will permit the use of reduced reinforcement in certain areas of the chassis. The SES is a conservative ruleset that requires teams to match chassis strength to a steel frame rather than design to the actual loads that will be seen by the car. The Alternate Frame rules offer more realistic loading scenarios, an example of which can be found in AF4.3, which specifies the loading scenario required to analyze the Side Impact Zone. The load specified in the AF rules (7kN) is allowed to be distributed over a 10 inch diameter circle which significantly reduces the stress seen by the panel. For comparison, the corresponding requirement in the SES demands that a representative panel is tested in a standard three point bend test fixture with a much narrower loading foot. The approved Side Impact Panel submitted in the 2015 SES supported a maximum load of 8kN in this setup. The large loading foot of the AF Rules is a more accurate representation of a side collision and may reduce the necessary reinforcement at many locations on the chassis.

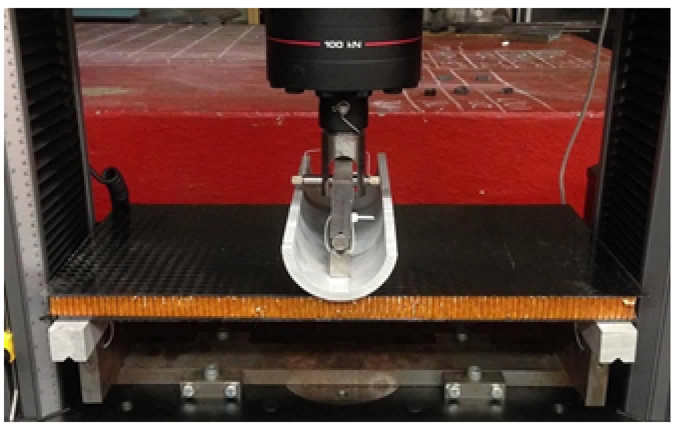

Figure 26: Side Impact Structure GFR2015.BB.01 Test for SES

The Alternate Frame rules are still relatively new for FSAE (2011) and as such, the path is not yet well defined. Interpretation of the rules is an ongoing activity for GFR and the findings will have a significant effect on the potential benefits of adhering to the AF rules.

4.1.2 Threats

While producing a functional model revealed a number of benefits, two significant factors in determining the value of following the AF rules are the time and resources required when compared to the SES. The meshing and analysis of the chassis took a significant amount of time but this is expected to go down in future years with the help of GFR tutorials and with the plan to use the same molds for the next several years. In addition to time spent in FEMAP, an array of physical testing of the composite layups is required with the SRCF submission. These involve beam bending tests for each laminate type used on the Side Impact Structure, Front Bulkhead, Front Bulkhead Support, Front Hoop Bracing, Main Hoop Bracing Support, Impact

Attenuator Anti-Intrusion Plate, and Shoulder Harness Bar. Additionally, perimeter shear testing must be completed for all panels that include an attachment to the Main or Front Roll Hoop, or Main Hoop Bracing [15]. These tests are laid out in the FSAE rules in sections T3.28-T3.34. Panels with duplicate ply schedules do not need separate tests which adds value to chassis designs that have little variation in laminates.

One potential design brought up in the development stage was to develop multi-thickness panels that use more plies towards the middle, where moments are greatest. While this appears to fall within the rules laid out by the Monocoque General Requirements and the Alternate Frame Rules, the potential weight savings need to be compared with the increased time and materials required to design and test these specialized panels. The Alternate Frame Rules additionally offer greater flexibility in the design of the Main Roll Hoop and braces, Front Roll Hoop, and Front Bulkhead by removing geometric restrictions specified in section T3.

There is high confidence that FEMAP is a competent and approachable program suitable for GFR's composite analysis needs. The primary concerns associated with implementation and adherence to the AF rules are primarily time based. Better estimates of necessary time for analysis will be developed once the learning curve has leveled off, tutorials have been finished, and more time can be applied directly to the chassis analysis.

4.2 Technical Specification

The Torsional Stiffness Analysis tested the application of many of the FEA tools under review and quickly established which methods could be implemented. When submitting the SRCF, the AF Rules (AF1.4) recommend including a brief report defending the validity of the analysis. This clause specifically asks for a discussion of the element types, mesh quality, and boundary conditions. These factors are discussed in the report below to begin guiding the development of the analysis with a focus on the aspects of analysis which are important to the AF Rules Committee. The sections below refer to the state of the analysis used in the Torsional Rigidity Test.

4.2.1 Meshing and element type

The chassis has been meshed by hand with a predominant mesh size of .01 m. When running an element check on the model, 1661 of 39698 elements failed under the standard test parameters inspecting the aspect ratio of the elements, internal angle, taper, and other factors that affect the quality of the mesh. The mesh took approximately 12-15 hours of labor to complete. The method developed an extremely regular mesh with consistent elements around the corners as can be seen in Figure 27 below.

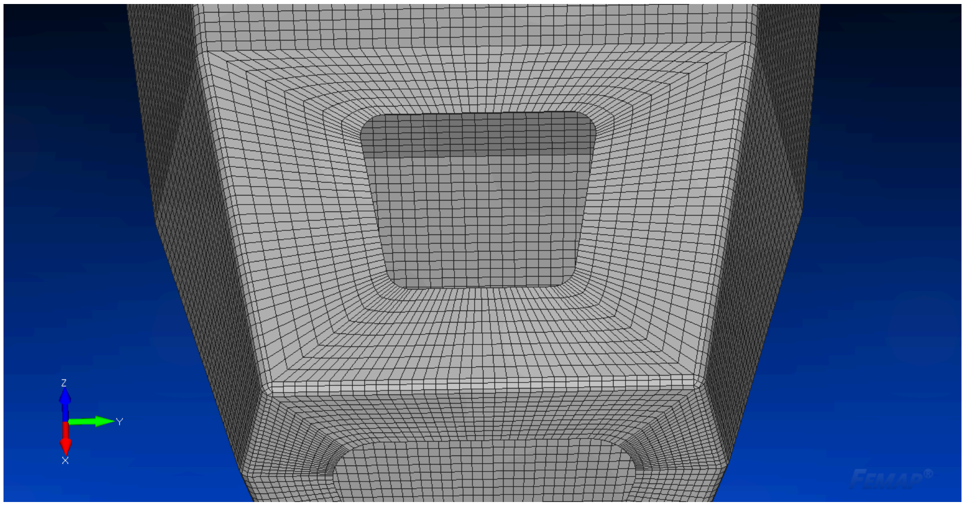

Figure 27: Detail View of Chassis Mesh

The laminate is modeled with 2D, planar elements using a uniform global ply. This method ignores the out of plane effects in the composite and is a significant simplification of the ply schedule.

4.2.2 Suspension links

A significant factor in the validity of this test was appropriately defining the suspension geometry and constraints. RBE2s were initially considered, however, RBEs cannot be connected in series. Each Rigid Body Element (RBE) rigidly links two nodes and the nodes must be specified as either the dependent or the independent node. A single node cannot be given conflicting definitions from multiple rigid body elements. Instead, rod elements were selected and specified with an unrealistically large modulus of elasticity and diameter such that any axial deflection of the link would be negligible. The suspension was modeled by importing the suspension geometry skeleton and applying coincident nodes and elements. See Figure 25 for a depiction of the suspension links. This was a successful method which could be repeated.

4.2.3 Hardpoint Connections

The hardpoints were modeled using an array of RBE2s with the independent node located at the spherical bearing which joins the suspension link and the clevis. The dependent nodes were distributed over the approximate footprint of the hardpoint.

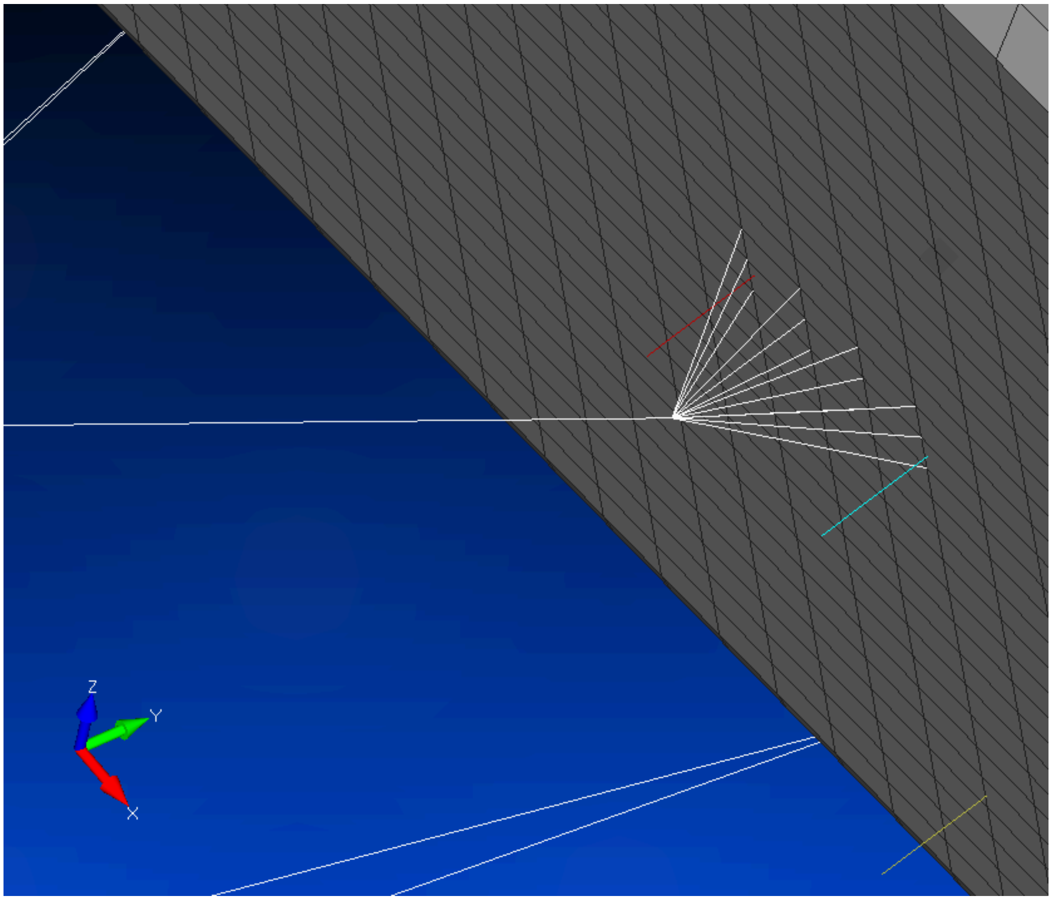

Figure 28: Detail View of Simulated Clevis Connection

By including the clevis geometry, this method applies the appropriate moment and shear force to the surface surrounding the hardpoint. This method works for the 2D elements although it will need to be verified for 3D elements. Drill guide geometry was imported to the model to help locate the hardpoints. Future progression of these connections could use the actual silhouette of the hardpoint to better represent the load distribution to the laminate.

It should be noted that this method of constraining the suspension geometry will not be the primary method used for boundary conditions in the AF rules analysis as the specified boundary condition for all tests is to fix the attachment points of the front and/or main roll hoops to the chassis. However, this method may prove useful in constraining the roll hoops to the chassis, and the modeling of the hardpoints at these attachment locations will be investigated.

4.2.4 Roll Hoop Integration

Moving past the Torsional Stiffness Test, roll hoops are being added to the chassis mesh to generate a more representative and AF Rules-ready model. A variety of element types were examined and Beam Elements appear to be the most suited to the application. There is the option to use Curved Beam Elements, however, most sources including the FEMAP Help Guide, indicate that using a series of straight elements is more common and provides accurate results. The Main Roll Hoop has been constrained to the chassis using RBEs in the same way as described above and is visible in Figure 29 below. The Front Roll Hoop will be the next addition to the model and will provide the opportunity to run all of the AF tests.

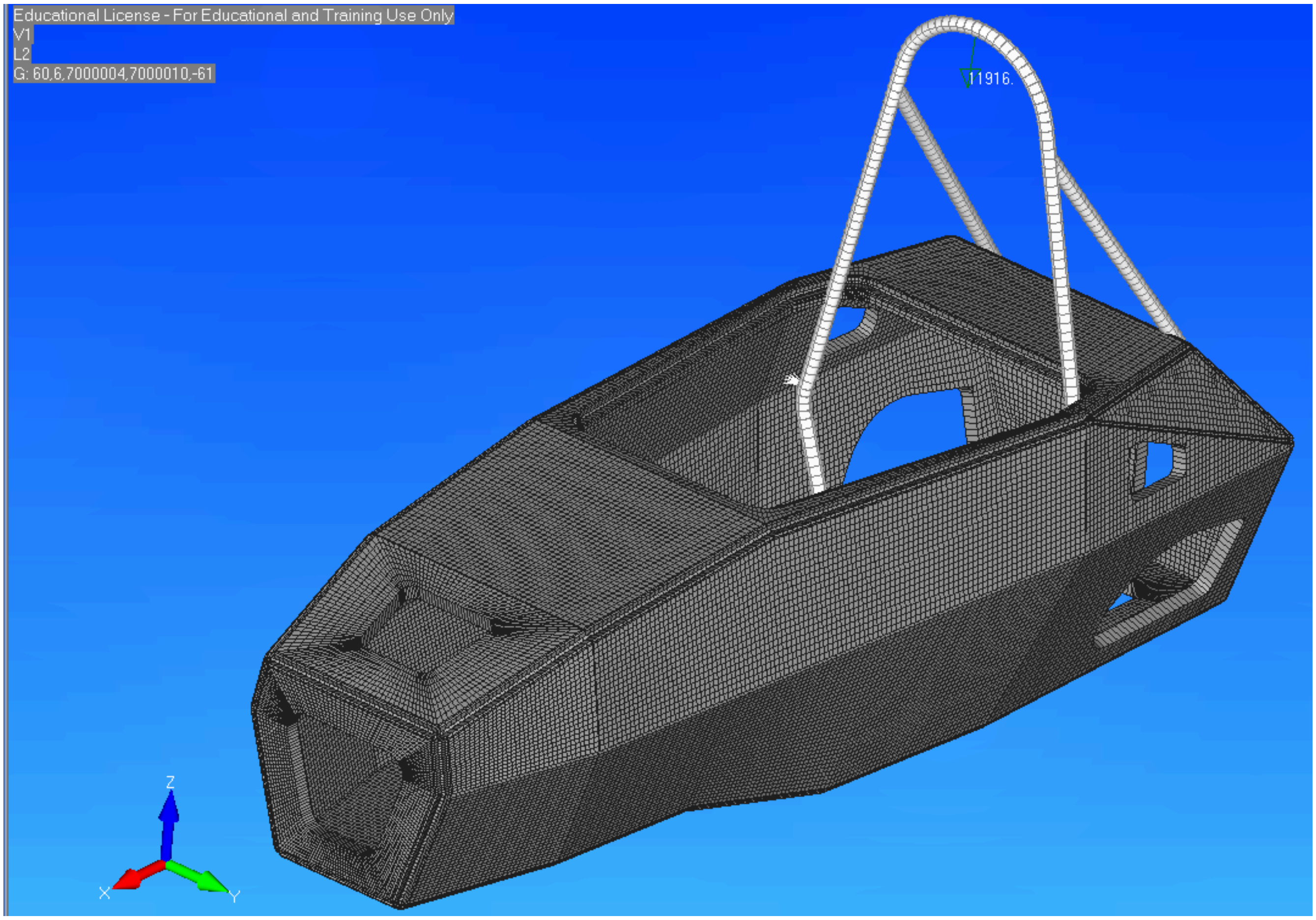

Figure 29: Main Roll Hoop Added to Chassis Mesh

5. Implementation

A detailed manufacturing/implementation record. Description of construction issues encountered and design changes made. Includes pictures and engineering analysis to support and describe all design changes made during the manufacturing / testing process. Engineering methodology justifies selection and sizing of tools used to manufacture or implement the design. Equations and sample calculations used included and explained. Assembly and process tutorials included in appendices when appropriate.An example of a good visual process flow chart for the manufacturing of the Formula chassis can be found HERE. (From Formula Chassis 2012 Report)

5.1 Documentation

As the complexity of the analysis model grows and more team members begin to contribute, it is crucial to maintain appropriate documentation to minimize duplication of work. Materials, processes, and revisions all need to be recorded.

5.1.1 Materials

To ensure that the most current material properties are being used and to verify that changes to materials have been adopted into the most recent chassis mesh, a folder and log has been created to store a copy of the file. When a material is created or updated a Femap neutral file containing only the material of interest must be exported and placed in the materials and layups folder using the file name format of “material name YY-MM-DD.neu”. The same process applies to the creation or change of layups. The change must also be documented in the Materials and Layup Log with a date and description of change.

5.1.2 Analysis Files

Currently, all Femap analysis files are stored in the Geometry, FEMs, and Results folder. The most commonly used sub folder is the chassis mesh folder where the full chassis mesh is stored. Every time a chassis mesh is saved, upload the file to this folder and note the change on the appropriate tab of the Femap File Iteration Log. Make change descriptions as detailed as possible.

With multiple contributors there is a risk of losing work due to using an out of date chassis mesh file. To minimize this risk, always delete the chassis mesh file off of the local hard drive after it has been uploaded to Google Drive and download the latest revision when you resume working. Also be aware of others working on the file simultaneously as there is currently no file vault system in place. If multiple contributors need to be working at once, a successful method is to work in separate files and combine created geometry, materials, or analyses, etc. by exporting a Femap neutral file.

An array of benchmark models are also necessary to gain a full understanding of the behavior of elements and the Benchmark Models folder has been created to store these files. Use the Tests and Results Log to record the reason for the study as well as a detailed account of the results.

5.1.3 Rules Questions

The Alternate Frame rules are still very young and a number of unclear sections and mistakes still exist. To ensure transfer of knowledge between team members a copy of the AF Rules and SRCF has been moved into a google docs page where comments can be made to record questions and answers.

5.2 Element Specification

The appropriate selection of element types and definition of element properties is a critical step in any finite element model. This is especially true for orthotropic composite materials as the material direction and element orientation must be appropriately assigned. Do not assume that any element property will automatically be assigned correctly.

5.2.1 Element Orientation

Element orientation is a pertinent problem with 2D Laminate Plate elements and was an option that created problems early in the development of the chassis mesh. The orientation is determined by the order in which the defining nodes are selected and defines the material application direction and thickness extrusion direction. The primary surface of the 2015 chassis mesh is defined using the mold surface and as such, the thickness of the laminate must extend towards the interior of the car. One indicator that the elements were poorly oriented was significant discontinuities in the surface contour plot, even when displaying criteria such as displacement which is not dependent on the laminate definition.

To reorient the elements, use the Femap “Reverse Normal/ Orient First Edge” command. All chassis elements should be directed inward.

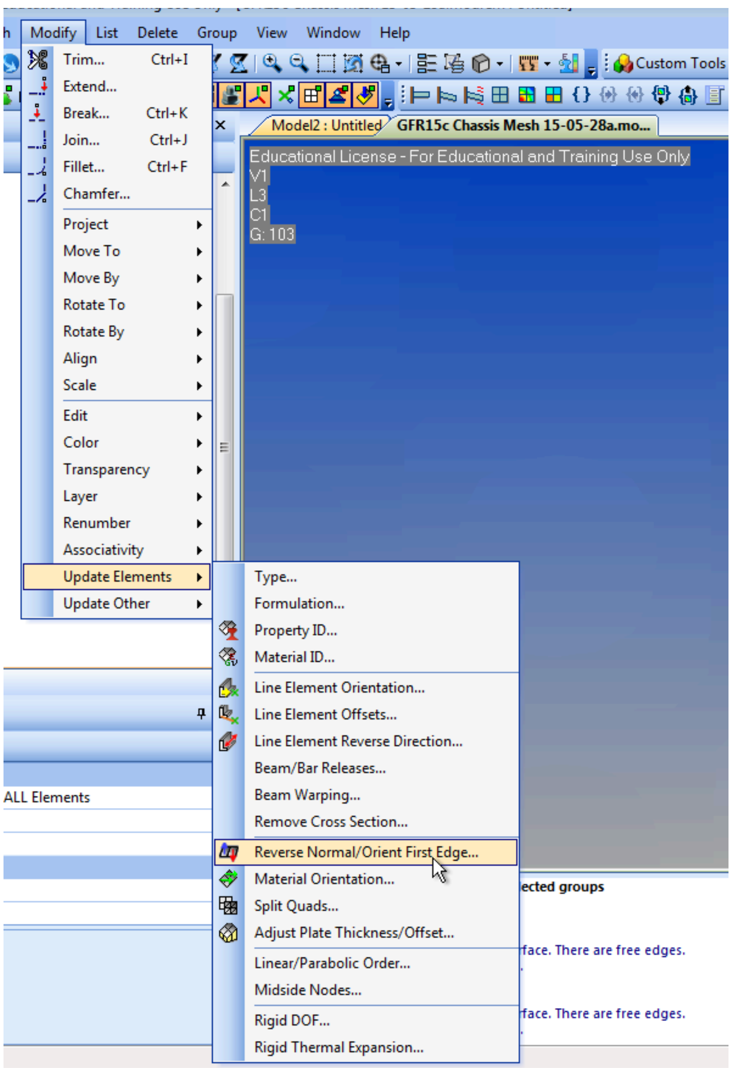

Figure 30: Re-orienting elements

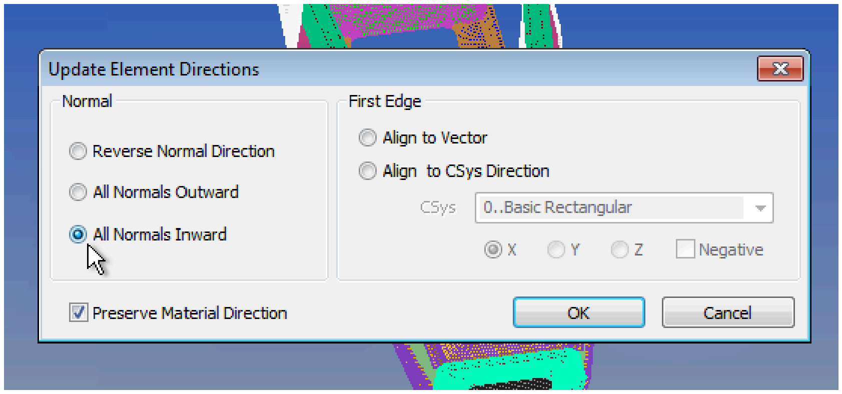

Figure 31: Select: All Normals Inward

5.2.2 Element Direction

.A problem that can manifest similar results is the specified offset of the laminate property. This defined option indicates that the laminate should extend in only one direction from the surface, rather than use the surface as a midplane. When this option is changed, a noticeable This is found in the property card editor for each laminate (not the laminate card) and is a checkbox titled Offset Bottom Surface. This check box should be selected and the offset set to zero.

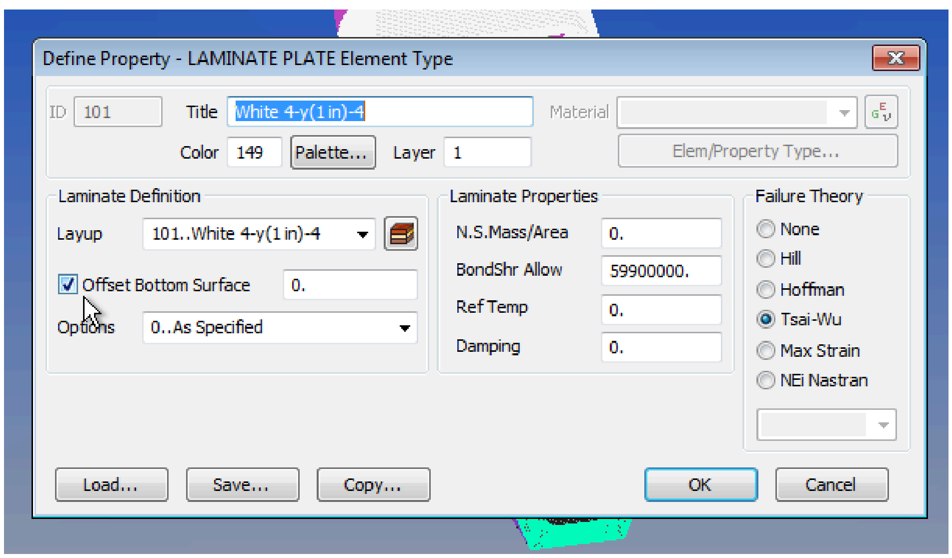

Figure 32: Turn on “Offset Bottom Surface” and set the offset to zero

5.3 Roll Hoop Connection Points

An area of special concern for the model is the connection between the steel roll hoops and the composite chassis. Many of the AF load cases specify constraining the model at these points and significant deflections and stress risers can result if the appropriate modeling techniques are not employed. In the same way that the suspension geometry is attached to the exterior of the chassis, the roll hoops are attached using an array of RBE2s connecting a single node on the roll hoop to the chassis nodes located over the mounting hardpoint. This construction can be seen in the figure below.

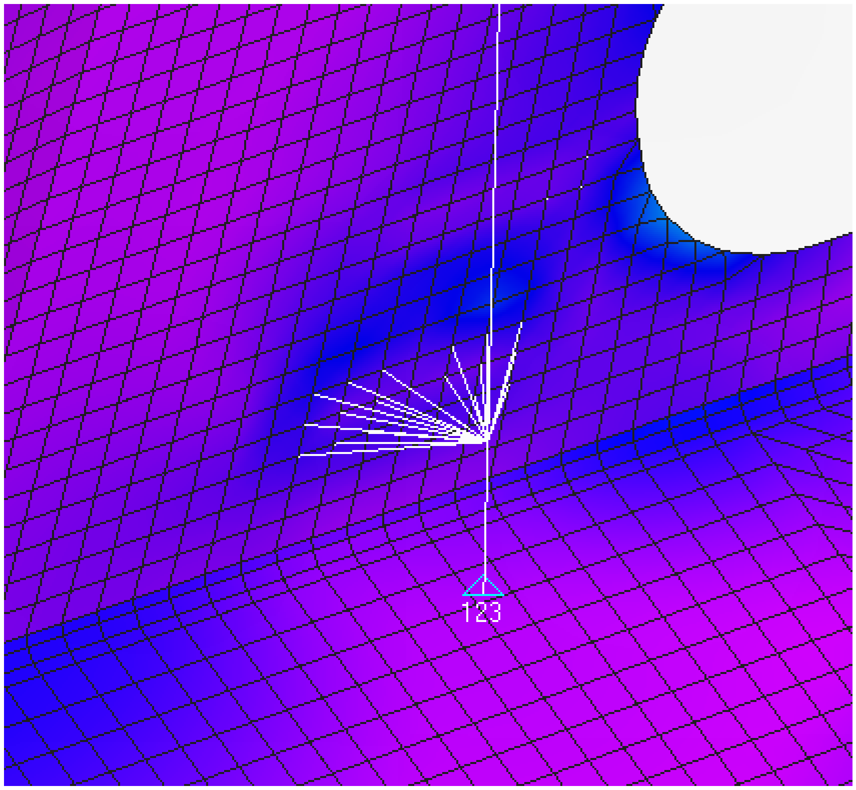

Figure 33: RBE2 connection between chassis and roll hoop

The RBE2 should be built with the master node on the roll hoop and the dependents on the chassis wall. The rigid bodies must be fixed in translation but not rotation.

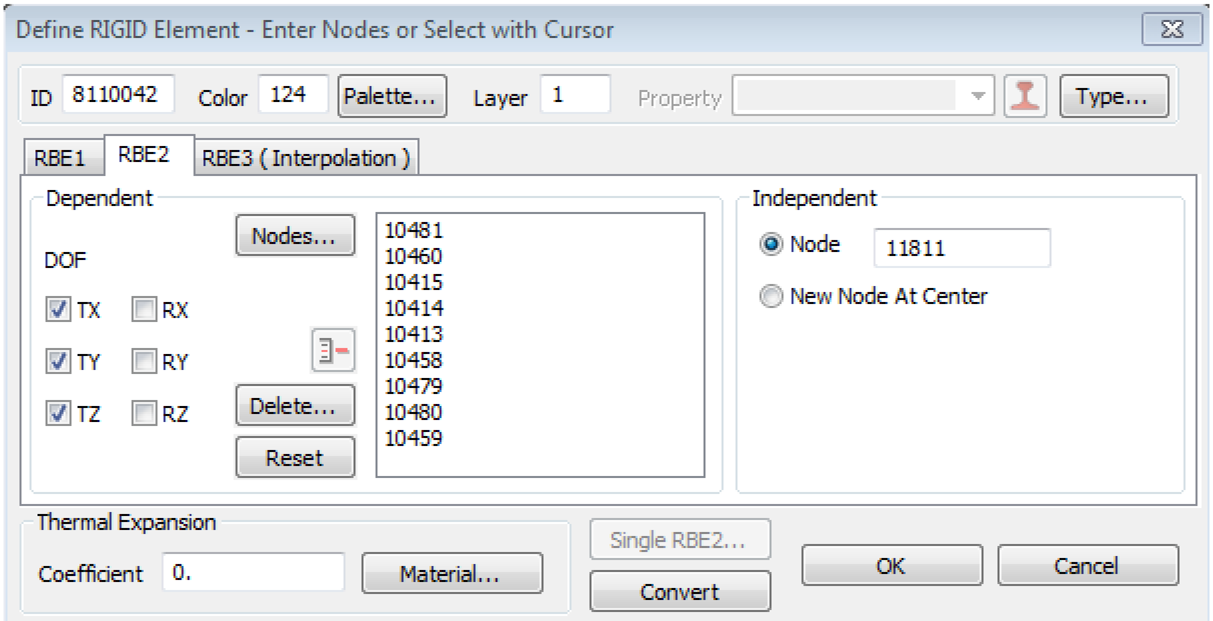

Figure 34: RBE2 Definition for roll hoop connection points

Future revisions of the chassis mesh should investigate more representative ways of modeling this connection and the hardpoints. One potential solution could be the creation of specific harpoint layups applied to the elements in the hardpoint area which use an aluminum core rather than a honeycomb. The roll hoop at this point could also be modeled with shell elements to gain a better understanding of the stresses in the tube at this point. To make a connection between a beam element and a shell element tube, RBE3s should be utilized. To effectively model this connection, specify the dependent node as the end of the beam element and specify the independent nodes as the terminal nodes of the shell elements around the entire circumference of the tube. The dependent node should be fixed in translation and rotation and the independent nodes fixed in translation only.

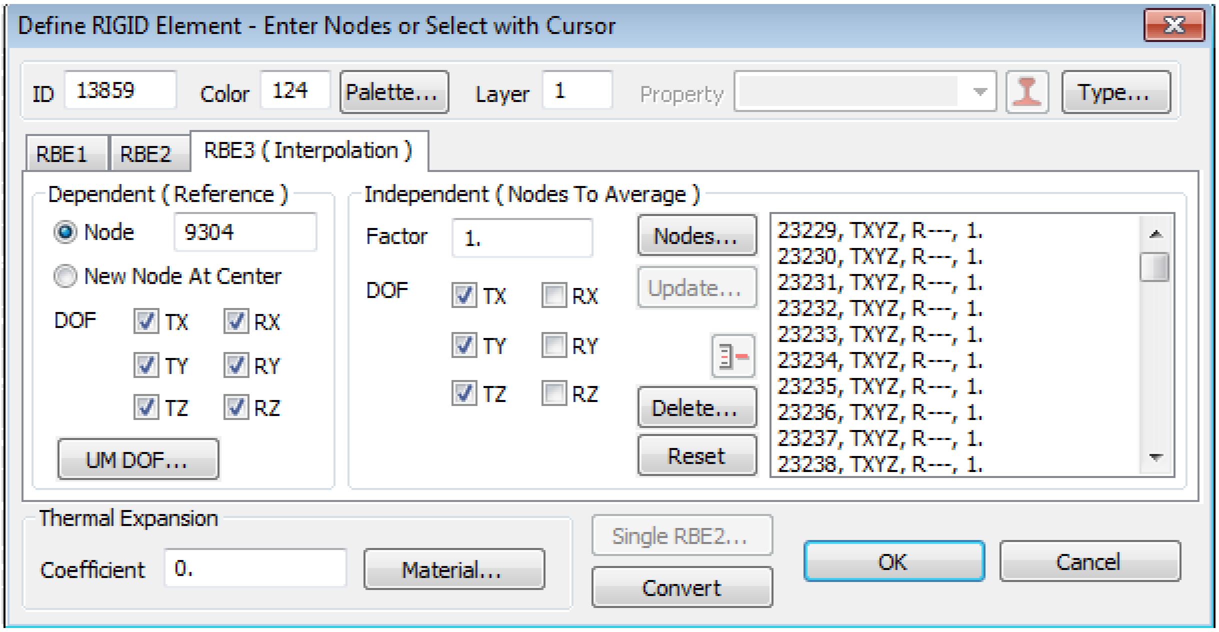

Figure 35: RBE3 definition for connection between beam elements and shell elements

This same RBE3 connection method will be used to apply loads to the chassis in some of the AF rules load cases.

5.4 Material Failure

The AF rules demand that material failure cannot occur under any of the load cases. To determine material failure, the appropriate failure criteria must be used for each part of the chassis. While this list is open to amendment, the most useful material failure indices are Max Strain and Tsai-Wu for composites, and VonMises Stress and buckling for the roll hoops. It should be noted that Femap prints out stresses in the fiber and material directions and not the element coordinate system.

5.4.1 Max Strain

To specify a failure criteria for the composite laminates, select the appropriate bubble in the property card for each laminate. Again, this must be done for every laminate and it is recommended that the same failure criteria be used in all locations for a single analysis run.

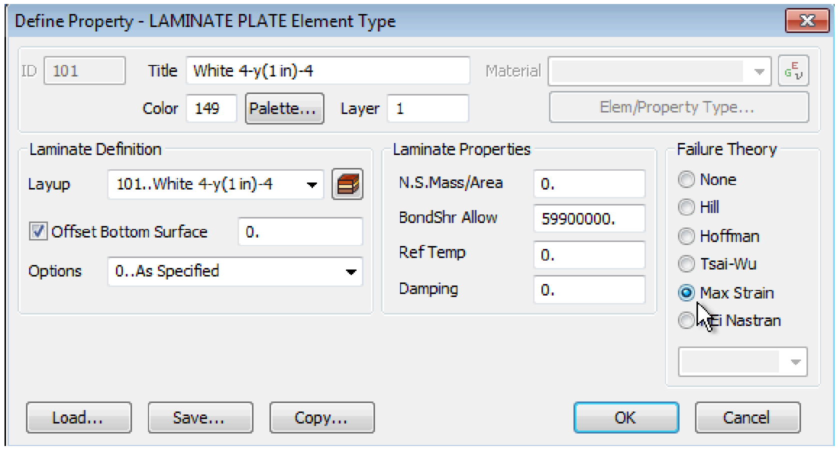

Figure 36: Laminate Property Card

The screenshot of the laminate property card above also shows the Bond Shear Allowable term just left of the Failure Theories. This term must be filled out for any of the composite failure theories to produce results in analysis.

5.4.2 Tsai-Wu

The Tsai-Wu criterion is somewhat more complicated than Maximum Strain but is commonly used in industry as it provides a conservative assessment and accounts for the difference in tensile and compressive failure strengths of composites. Bill Riley, head of the AF Rules Committee, recommends focusing on Tsai-Wu. The Tsai-Wu Failure criteria equation is shown below.

$$f_1 \cdot \sigma_1 + f_2 \cdot \sigma_2 + f_{11} \cdot \sigma_1^2 + f_{22} \cdot \sigma_2^2 + f_{66} \cdot \tau_{12}^2 + 2 \cdot f_{12} \cdot \sigma_1 \cdot \sigma_2 \geq 1$$

Tsai-Wu also accounts for the interaction of normal stresses with the Tsai-Wu Interaction coefficient f_{12}. While this value should be determined through biaxial testing, according to Daniel, Ishai [1] in section 6.6, a reasonable approximation can be made with the equation:

$$f_{12} = \; - 1/2(f_{11} f_{22})^{½}$$

At this point, the Tsai-Wu Failure criterion should not be trusted as simple benchmark models return erroneous failure index results when compared with hand calculations and Abaqus analysis. It is not currently clear why the Tsai-Wu Failure criterion is not behaving as expected. It is very possible that a material failure property has been improperly assigned. There is also the chance that an analysis option has not been properly selected. One surprising behavior of FEMAP is that it rounds the Tsai-Wu Interaction Coefficient (discussed above) to zero. The coefficient is a relatively small number but does make a contribution to the failure index. Tutorials, literature, and colleagues have been queried regarding this problem but to no avail. The next step will be to contact GFR's analysis sponsor, Predictive Engineering. A detailed report covering the benchmark models used validating the failure criteria was created.

5.4.4 Von Mises Stress Criteria

To determine material failure of isotropic materials, such as the steel roll hoops, the recommended failure criteria is Von Mises. This can be easily accomplished by comparing the Von Mises output to the maximum stress of the material being examined.

5.4.4 Buckling

A potential failure mode of the roll hoops is buckling under load. Buckling is a slightly more complicated failure mode to analyze and requires a special analysis with altered conditions. When creating the new analysis the analysis type must be set to Buckling.

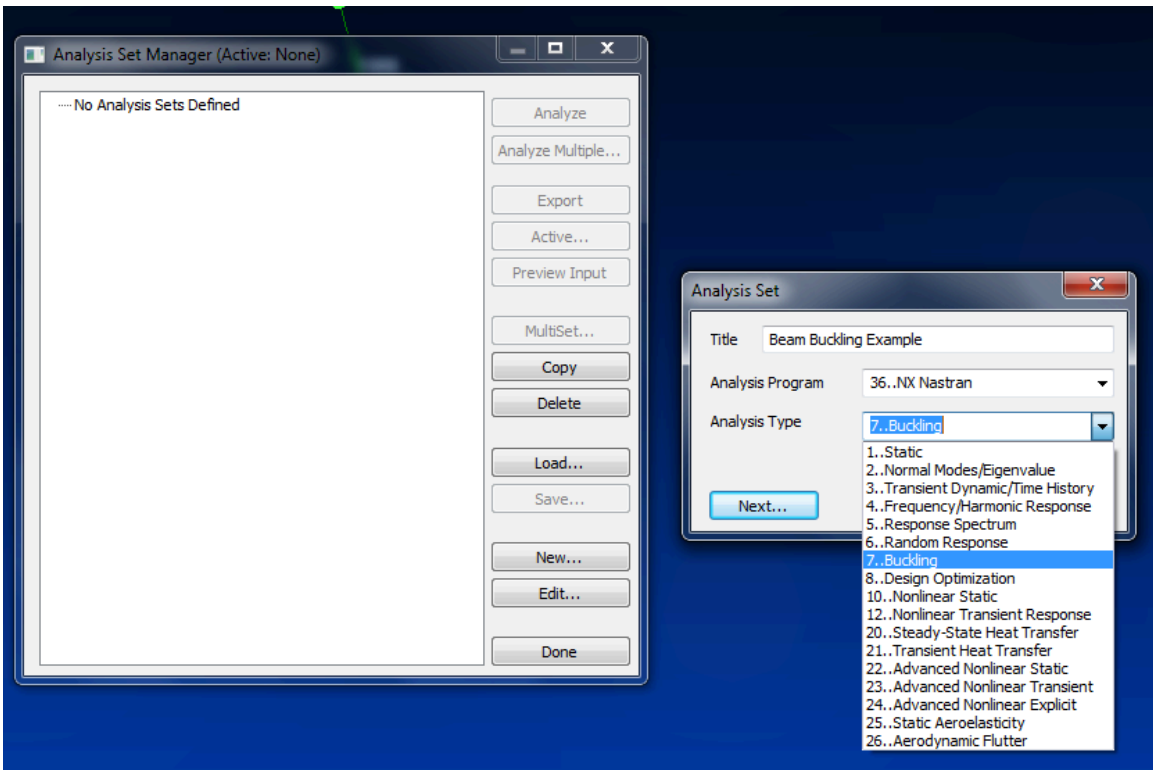

Figure 37: Analysis Set definition for buckling

Once the analysis has been run, the name of the results file is the most valuable output as it contains the eigenvalue for the buckling model. This is essentially the buckling factor of safety.

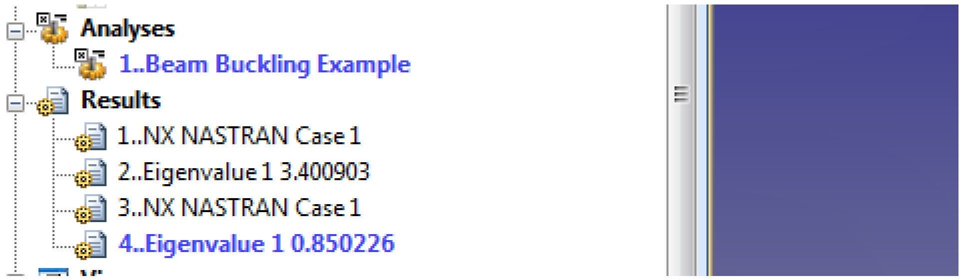

Figure 38: Eigenvalue buckling results output

In this case, it can be seen that the structure demonstrated an eigenvalue buckling factor of 0.85 indicating that buckling would occur and the structure would fail. If the buckling result was 4, this would indicate that the load is only one fourth of the buckling load. It should be noted that a separate analysis must be run as none of the other stress or deflection results from a buckling analysis are correct as they are all normalized to 1.

6. Application to the Alternate Frame Rules

With the model built, analysis of the AF load cases must be completed. This section details the geometries created to apply the loads and to constrain the chassis for each load case. To avoid misrepresenting the rules in the case of a rule change, the specific values of load cases will not be discussed in depth and should be pulled from the most current FSAE Rules document.

6.1 AF 4.1 Main Roll Hoop, Bracing and Bracing Supports

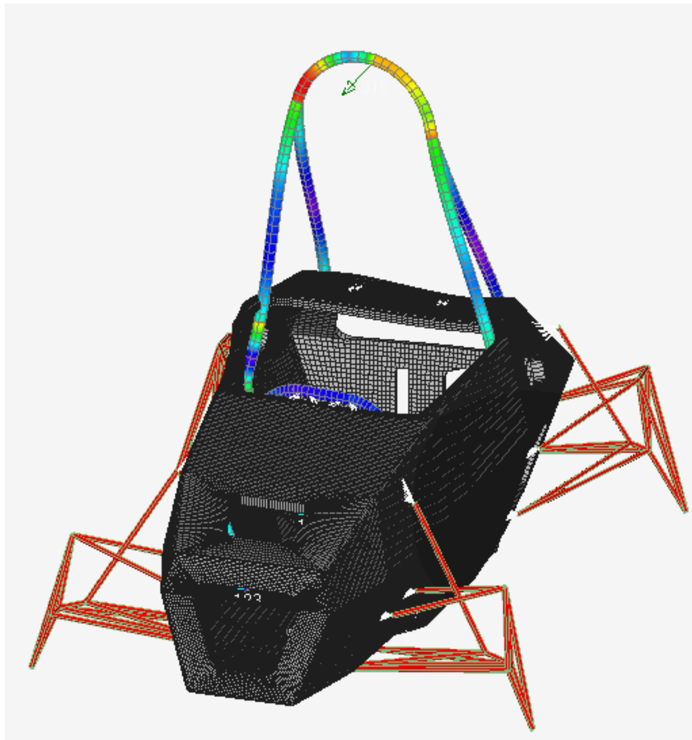

Figure 39: AF 4.1

Load:
The load was applied to a single node at the middle of the main roll hoop.

Constraints:
Nodes at the lower connection point of both roll hoops were fixed in translation but not rotation.

Results Analysis:
Buckling is of significant concern for this test and must be analyzed appropriately. The majority of the deflection occurs within the roll hoop although the chassis to roll hoop connection points did show high stress.

6.2 AF 4.2 Front Roll Hoop

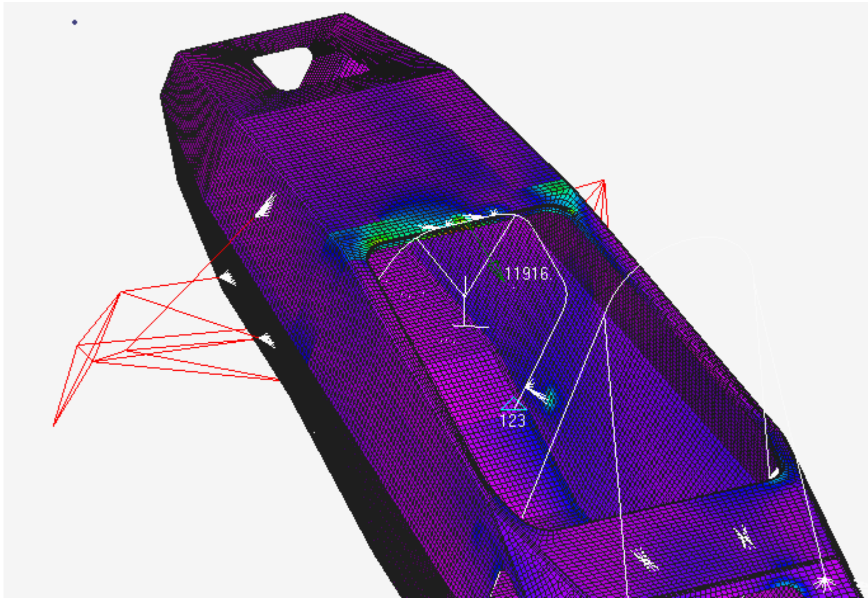

Figure 40: AF 4.2 Max composite strain

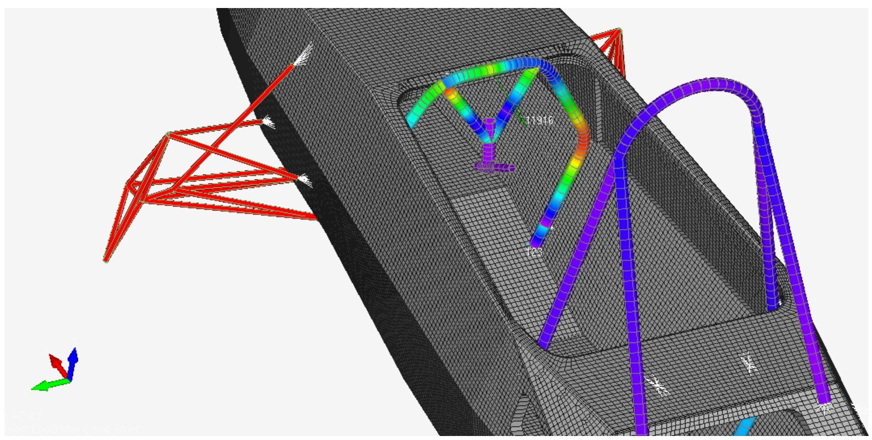

Figure 41: AF 4.2 Max beam stress

Load:
The load was applied to a single node at the middle of the main roll hoop.

Constraints:
Nodes at the lower connection point of both roll hoops were fixed in translation but not rotation.

Results Analysis:
Stress in the front roll hoop must be considered as well as buckling. Composite failure is only a moderate concern.

6.3 AF 4.3 Side Impact

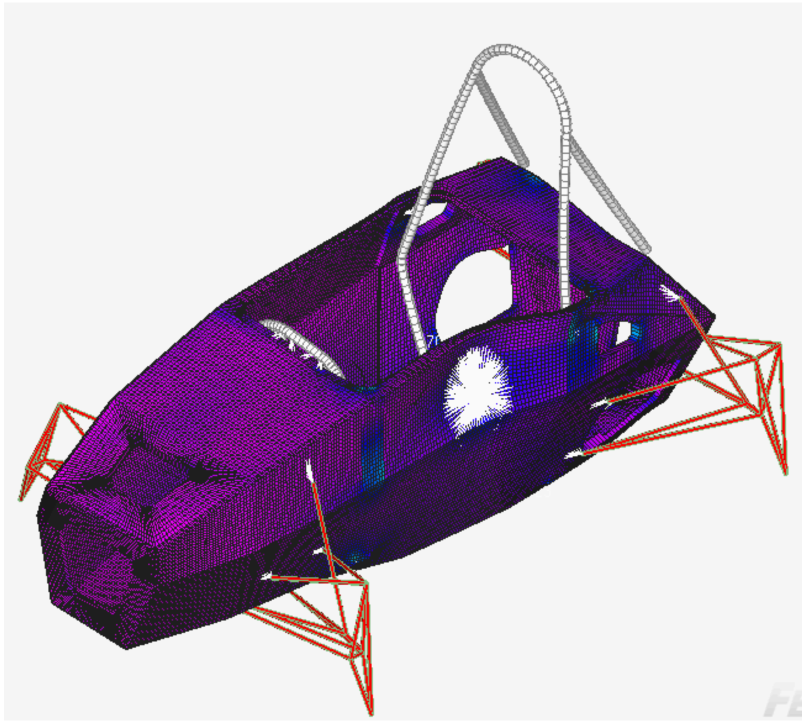

Figure 43: AF4.3 side impact RBE3 load application

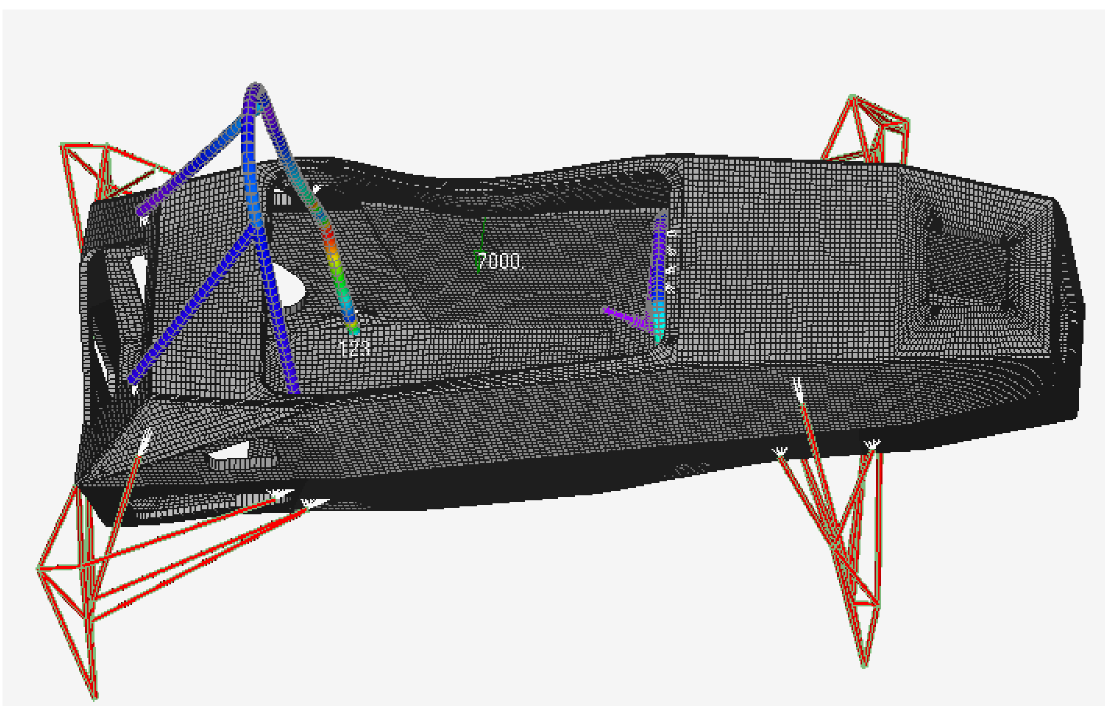

Figure 44: AF4.2 Max beam stress

Load:
As recommended in the rules, a load was applied to an array of RBE3s. This method is described in detail in section 5.3 of this report.

Constraints:
Nodes at the lower connection point of both roll hoops were fixed in translation but not rotation.

Results Analysis:
This load needs to be applied in a variety of locations to establish the worst case scenario. Laminate stress in the corners of the rear access holes are significantly higher than the rest of the chassis.

6.4 AF 4.4 Front Bulkhead & Bulkhead Support

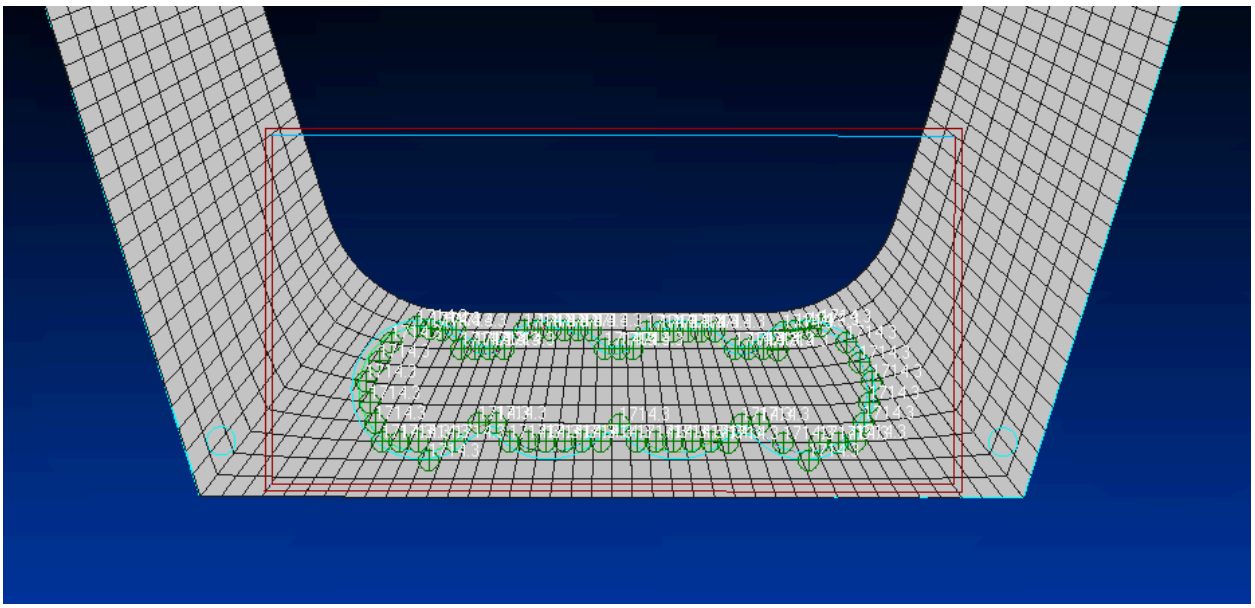

Figure 45: AF4.4 Loads applied to front bulkhead

Load:
To more accurately represent the load application and dispersion of forces, an aluminum bulkhead plate was added to the model. Loads were then applied to the nodes directly under the impact attenuator tubes where the load would be transferred in the event of a front impact. This is an overly complicated load application method and could be simplified with the use of an RBE3 array covering the entire impact attenuator area similar to AF4.3.

Constraints:
Nodes at the upper and lower connection points of the main roll hoop were fixed in translation but not rotation.

Results Analysis:

This load case produces extremely high stresses in the composite behind the front bulkhead as well as the connection points of the main roll hoops. This load case may necessitate a thicker ply in this location unless a more lenient load application method is developed.

6.5 AF 4.5 Shoulder Harness Attachment

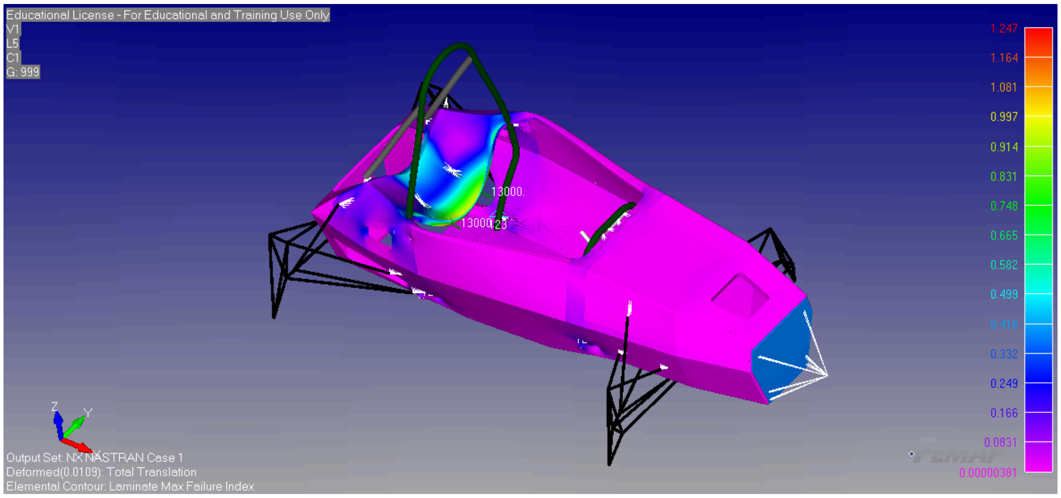

Figure 46: AF4.5 Exaggerated deformation, contour plot of Tsai-Wu

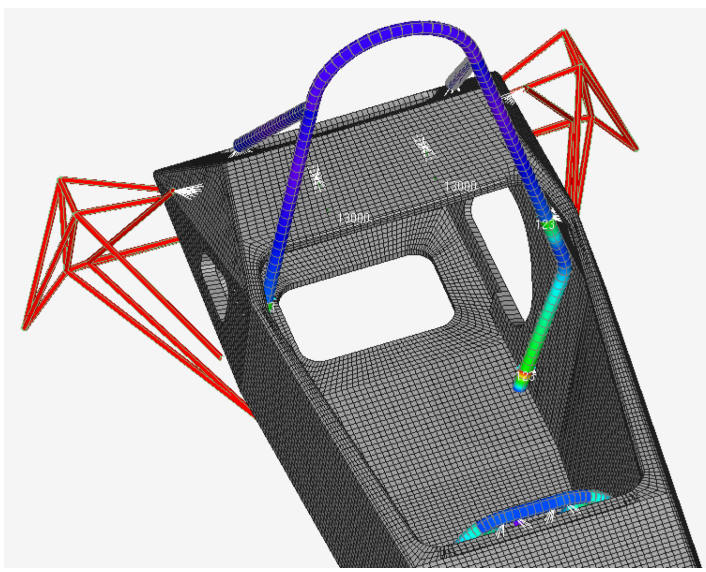

Figure 47: AF4.5 Max beam stress

Load:
To apply the load at the shoulder harness connection points, an array of RBE3s was created over the nodes where the hard point is located. The load must be applied in multiple directions within a specified range of angles (FSAE Rule T5.4.4) to determine the worst case scenario.

Constraints:
Nodes at the upper and lower connection points of the main roll hoop were fixed in translation but not rotation.

Results Analysis:
For the 2015c chassis, the highest stresses resulted by applying the load at the angle furthest

below horizontal and stresses indicating material failure were detected in the chassis.

6.6 AF 4.6 Lap & Anti-Submarine AF Harness Attachment

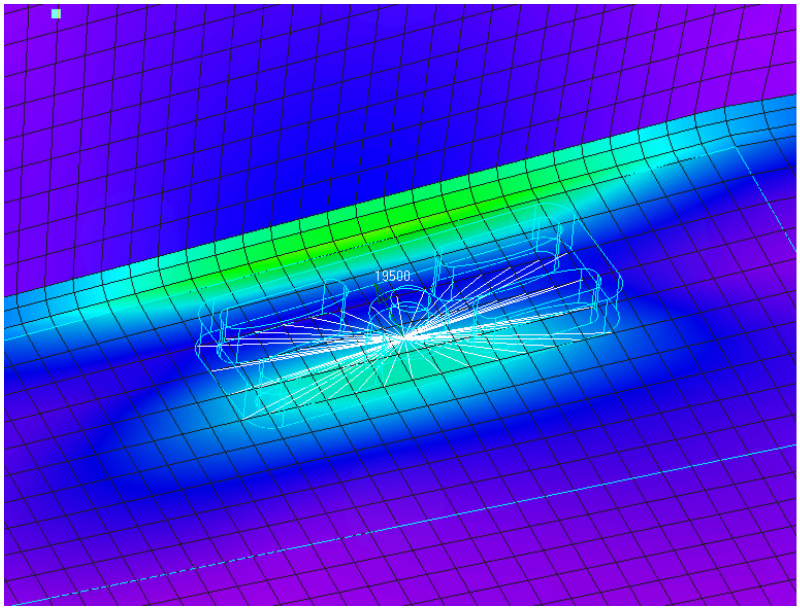

Figure 48: AF4.6 RBE3s placed over lap belt hardpoint

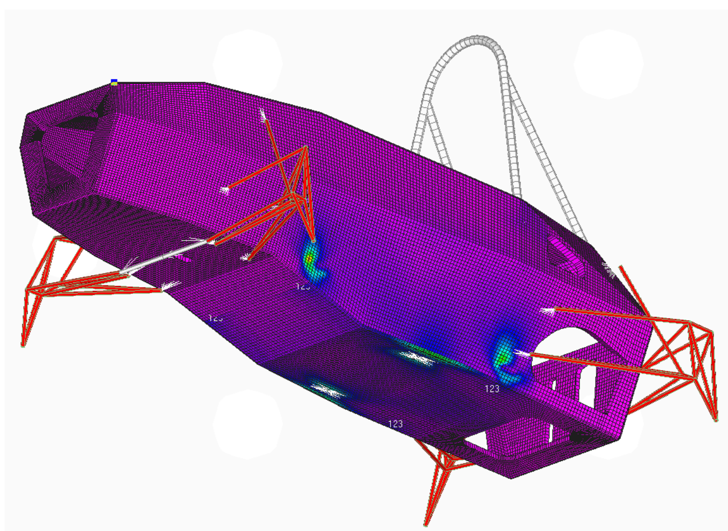

Figure 49: AF4.6 Max Strain

Load:

Load application for AF4.6 is done in a very similar way to AF4.5, however, the load and RBE3s are moved to the lap belt connection point hardpoints. The load must be applied over a range of angles (T5.3.5) and worst case scenario reported.

Constraints:
Nodes at the lower connection point of both roll hoops were fixed in translation but not rotation.

Results Analysis:
The worst case scenario was found when the load was applied at 45° above horizontal. This may be a false result that is a function of using 2D elements to model the chassis. High stresses below the failure point were detected where the floor of the chassis meets the wall.

6.7 AF 4.7 Front Bulkhead & Bulkhead Support Off Axis

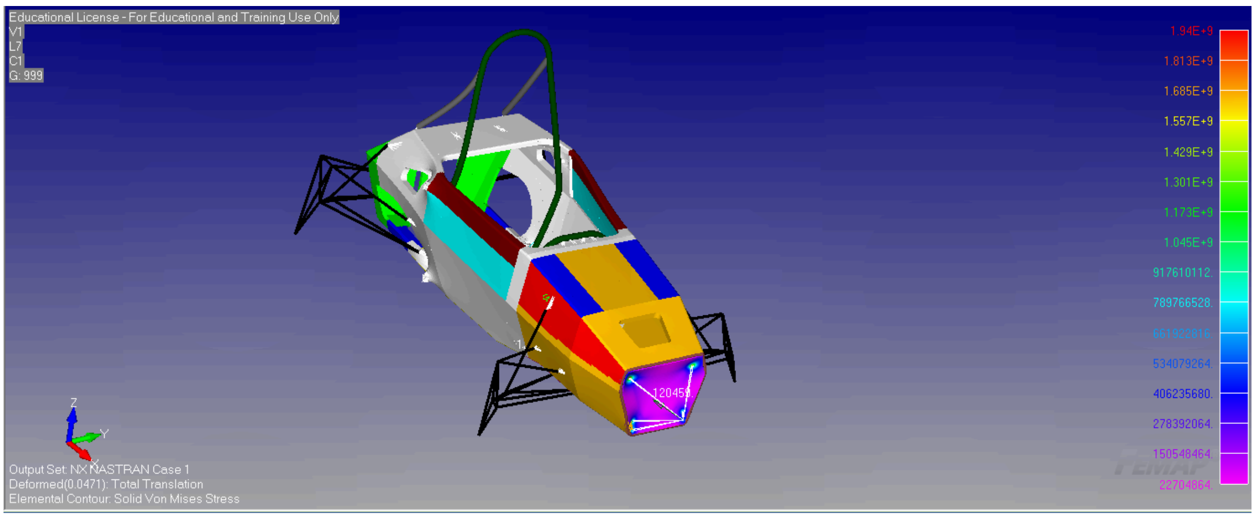

Figure 50: AF4.7 Von Mises stress in aluminum bulkhead

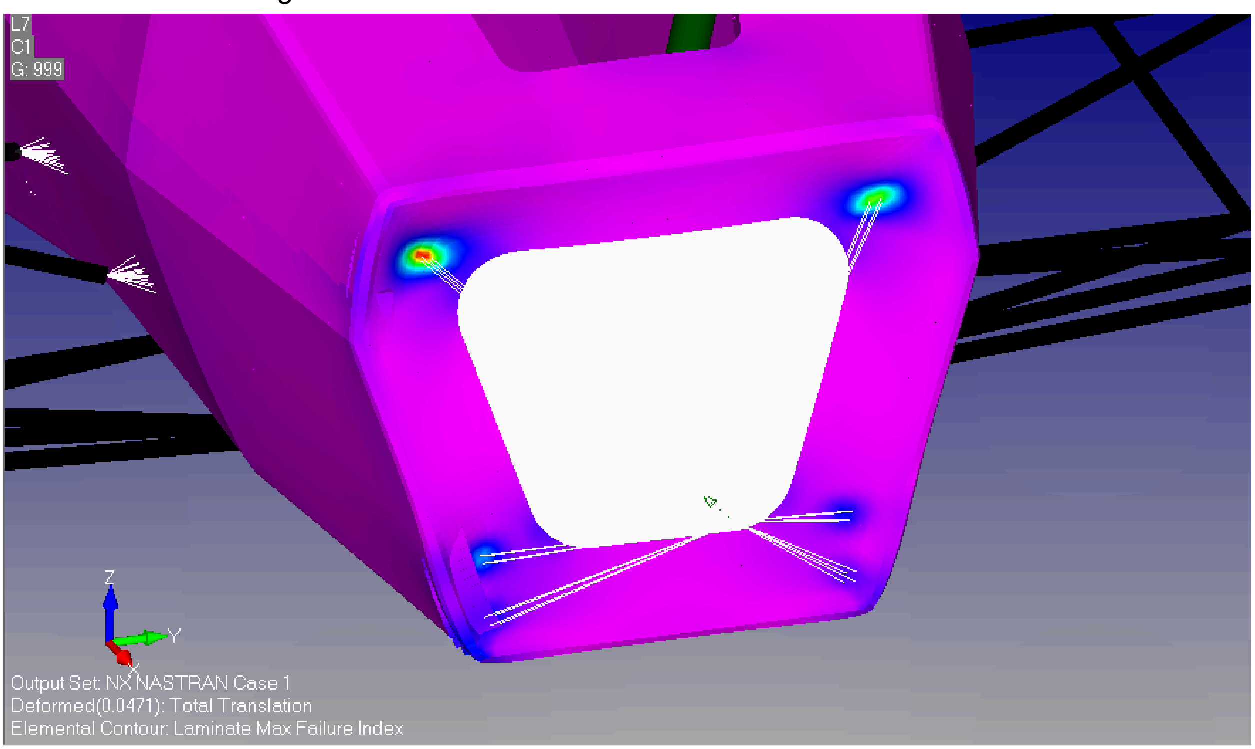

Figure 51: AF4.7 Max Tsai-Wu

Load:
An array of RBE3s was created connecting a middle node where the load is applied to the attachment points of the front bulkhead and the impact attenuator. This load application should be reconsidered as it does not appropriately represent the way a real load would occur.

Constraints:
Nodes at the upper and lower connection points of the main roll hoop were fixed in translation but not rotation.

Results Analysis:
This load application method generates local areas of extremely high stress in a way that is not realistic.

7. Conclusion

The intent of this study was to develop a finite element model and process using NX NASTRAN Femap that could be used to fulfill the requirements of the FSAE Alternate Frame Rules in future years. This required building my own and the team's understanding of composite material behavior, finite element theory, and the specifics of using Femap. A representative model was constructed and all load cases were applied to the 2015c chassis. This model was presented to the head of the FSAE Alternate Frame Rules Committee and was well received.

While adherence to the Alternate Frame rules is a goal for the analysis program and establishes a clear benchmark for the team's capabilities, pursuit of improved analysis contains inherent benefits. Finite element models are expected for the design of most other manufactured components in GFR, however, the current state of the team's composite FEA is at a very low level. While the current methods have been proven to produce winning designs, this is not a reason to stop pushing the envelope. The work completed in this study has developed a useful tool that can help improve the design of future chassis by letting designers compare a variety of layups at zero material cost and marginal labor cost. It was especially beneficial to build this finite element model this year as the new chassis form will likely stay the same for a few seasons. The geometry and elements established are immediately ready for the analysis of new layup designs.

7.1 Future Work

While a strong foundation has been laid, there is still work left to be done before a complete Alternate Frame Rules submission can be completed. Future analysts can look to this list for inspiration and direction.

- Change laminate definition in hardpoint areas to contain a solid aluminum core.
- Use shell elements to model the roll hoops and roll hoop connectors.
- Improved documentation and file storage of material properties.
- Improved understanding and documentation of the capabilities and outputs of each element type to maximize information gained from each analysis.
- Develop a tool that can combine the results from multiple analyses to determine which areas of the laminate are overbuilt and which areas carry the highest stress. This method could be equated to structural topology optimization.
- Investigate the alternative method of constraining the chassis with an inertial constraint. This would reduce the high stresses around the roll hoop connection points and could potentially allow for thinner laminates.
- Determine effective way to share files from Femap Student, with users who have the complete version of Femap. Currently there is a compatibility glitch in the software that inhibits file sharing and an update fixing this has been promised for July 2015. It is doubtful that GFR will get this update without some effort.

7.2 Personal Reflections

The most valuable part of a senior project with the Formula team is the chance to dive into a high level technical problem that is aligned with your interests. My project was a perfect example of that as composites FEA was something I have been interested in and feel will be a useful skill to progress my career. I am very happy with the knowledge I have gained from my project and feel that it has made a positive contribution to the team.

Unfortunately, I was not able to commit as much time to the team as some others as I have also been completing my honors thesis. I do appreciate the flexibility from the TAs regarding deadlines and commitment and their willingness to work with me to find an appropriate balance of work. Gabe's feedback on my reports was well placed and insightful. I also feel that the weekly presentations are a valuable part of the process.

Recognizing my limited time and individual nature of my project, the full team meetings did not feel like the most productive use of my time, however, I understand their purpose and recognize they are crucial for the function of the team. The material in those meetings should be reduced to only critical information. While it is somewhat surprising, I did not use Catia at all during my project and wish I could have used the time committed to the Catia tutorials to work on my project.

I appreciate the opportunity to work with GFR and wish future teams all the best.

8. Works Cited

[1] SAE Composite Chassis Manufacturing Process, 2009 Chassis Team

[2] "01_FEA Basics (FEMAP, Solid-Edge).pdf." Google Docs. Web. 23 Jan. 2015. <https://docs.google.com/a/ba-racing-team.de/file/d/0B3W_IYqhsqARcTEtUG5CMVdoYXM/edit>.

[3] FSAE. "2015 Formula SAE® Rules." (n.d.): n. pag. SAE.org. FSAE. Web. <http://students.sae.org/cds/formulaseries/rules/2015-16_fsae_rules.pdf>.

"[4] FSAE. "2015 FSAE Alternative Frame Rules Sample Frame" SAE.org. FSAE. Web. <http://www.fsaeonline.com/content/2015%20AFR_Notice_of_Intent_Sample_Frame.xls>"

[5] "Design of composite sandwich panels for a Formula SAE monocoque chassis", Robert Story

[6] "Sandwich Panel - 12." Protech Composites. Web. 23 Jan. 2015. <http://www.protechcomposites.com/sandwich-panel-12-x-12-5-nomex-core/>.

[7] "Downloads." SAE.org. FSAE. Web. 23 Jan. 2015. <http://www.fsaeonline.com/page.aspx?pageid=5ade9b01-8903-4ae1-89e1-489a8a4f08d9>.

[8] Daniel, Isaac M., and Ori Ishai. *Engineering Mechanics of Composite Materials*. New York: Oxford UP, 1994. Print.

[9] "Lamina Coordinate Transformation for Arbitrary Orientation." *Lamina Coordinate Transformation for Arbitrary Orientation*. Web. 23 Jan. 2015. <http://www.efunda.com/formulae/solid_mechanics/composites/comp_lamina_arbitrary.cfm>.

[10] "Competition Results" Global Formula Racing. Web. <https://sites.google.com/a/ba-racing-team.de/global-formula-racing-general /wike/competition-events/4_competition-results#TOC-2014>

[11] ASTM International. C393 / c393m standard test method for core shear properties of sandwich constructions by beam flexure. ASTM 15.03, October 2011.

[12] Bheemreddy,Venkata and Adrian Jensen. "Composite Laminate Modeling." Technical Online

Seminars. 2014. Predictive Engineering. 2/15/2015

<http://www.appliedcax.com/support-and-training/technical-online-seminars/seminars/composite-laminate-modeling/Composite%20Modeling%20White%20Paper%202014%20Rev-0.pdf>

[13] Jensen, Adrian. "Composite Modeling." Personal interview. 13 Feb. 2015.

[14] “OSU Hardpoints and Core Report”, GFR, Simon Bettscheider, 2015.

[15] “SES Tutorial (Up to 2014)”, GFR, 2014

[16] "George E. P. Box." - *Wikiquote*. N.p., n.d. Web. 23 Apr. 2015.

http://en.wikiquote.org/wiki/George_E._P._Box