Spring 2020

Mini Baja Frame Analysis

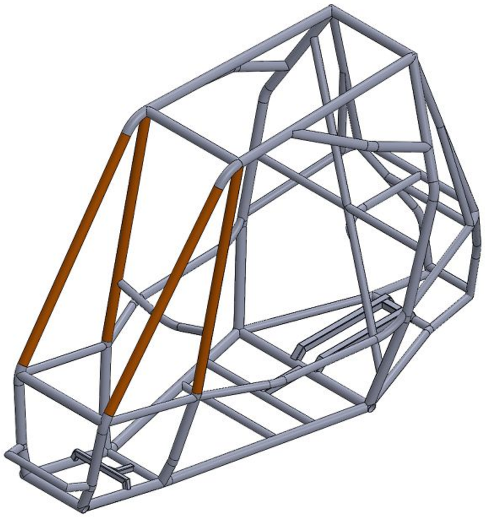
Jacob Swanson
jts136@zips.uakron.edu

Recommended Citation
Swanson, Jacob, "Mini Baja Frame Analysis" (2020). *Williams Honors College, Honors Research Projects*. 1085.
https://ideaexchange.uakron.edu/honors_research_projects/1085

SAE Mini Baja Frame Analysis

Anthony Bodde
Nicole Fletterick
Alexander Stanik
Jacob Swanson

Dr. Christopher Daniels
Department of Mechanical Engineering
The University of Akron
Spring Semester 2020



Abstract

The Zips Baja Off-Road Racing team needs a cost effective, and accurate testing method for future frame designs. The following report exhibits the design cycle performed to correlate the real world testing to CAD testing of the frame design.

Executive Summary

The purpose of this report is based on education. The mathematics, formulation of tests, experiments, and FEA are all done in the pursuit of knowledge. We set out to gain knowledge in the field of impact loads, and examine the results of how applied forces can change the shape of a design. In order to come up with a cost effective and reliable testing method for the Zips Baja Off-Road Racing Team, our senior design group used our research to help ease future teams in their frame confidence. The CAD drawings and tests performed through FEA helped solidify an understanding as to what needs to be improved upon in the future. Overall the results of both real life testing and FEA showed that the current frame design is the best fit for the frame team. The weight loss in removing two of the beams is not worth the compromise in structural integrity. Our results in this paper will reflect our findings in this summary and help to confirm the hypothesis in our original proposal.

Acknowledgements

Special thanks to our faculty advisor Dr. Chris Daniels, Steve Gerbetz, Dave McVaney, and Yesteryear Farms for their guidance throughout the design and testing process. As well as The Zips Baja Team for support throughout the project.

Table of Contacts

Chapter 1: Introduction

1.1 Background

The Society of Automotive Engineers Collegiate Design Series offers several types of competitions constructed to allow the majority of engineering disciplines to test their understanding in semi-real world situations. The types of competitions vary from Supermileage, Aero Design, all the way to Clean Snowmobile challenges. Narrowing to the Baja SAE rules, they state, “ The objective of the competition is to simulate real-world engineering design projects and their related challenges. Each team is competing to have its design accepted for manufacture by a fictitious firm. The students must function as a team to design, engineer, build, test, promote and compete with a vehicle within the limits of the rules. They must also generate financial support for their project and manage their educational priorities”(1). For the Baja SAE competitions, the students are asked to design, and manufacture an all-terrain vehicle with the aspiration of being a market-leading product. Each competition is split into two types of events. First there are static events which include a cost event, sales presentation, and lastly a design presentation. The design presentation allows each team to talk about and defend their design decisions. Second there are the dynamic events. These include an acceleration run, a maneuverability course, a suspension course, a hill climb or tractor pull, depending on the location of the event, and lastly a 4 hour long head to head endurance race.

With the release of the 2020 rule book, the Baja governing body also announced a proposed rule change for 2022. The proposed rule change would make an “All-Wheel Drive (AWD) or Four-Wheel Drive” vehicle mandatory for the 2022 season. Meaning in order to compete in 2022 the Zips Baja Off-Road Racing Team will have to develop an AWD vehicle. Weight is one of the largest driving factors to the baja vehicle design. Walking the fine line between lightweight and durability is something every top team is familiar with. The addition of the weight from the AWD system makes optimizing the rest of the vehicle to lose weight, where feasible, essential to the team’s success. Although the ZB19’s weight is comparable to the field's top teams, finding a cost effective way to test many optimized iterations is a necessity. The solution cannot only be cost effective but it must also be accurate, allowing the team to use the data found during this report for future use.

1.2 Literature Research

One of the concepts we had to understand before diving into this project was finite element analysis, or FEA. FEA is the process of taking a 3D rendered drawing and then applying some kind of strain, stress, or force to it. In our case we wanted to use FEA for an applied load scenario. In most cases, using FEA is just accepted as the truth and questions are not asked. The color of the drawing pops up, the results seem error free, and the final indication is accepted as what would happen in real life. Understanding the uncertainty in FEA helps to use it as a valid source in the scientific method. Reading on the topic of FEA, it has become an accepted notion that many uses of the technology can perform the tasks required, but do not fully grasp the theory as to how the technology works. This is where the differing views on the accuracy of FEA lie. An accuracy of 20 percent error is considered to be very high functioning and this reflects the overall uncertainty that can arise in a mesh that is inconsistent with raw real data.

Having repeatability of an experiment is something that defines success in the scientific world. The scope of this project needed to be validated by the repeatability of the testing process. The testing of FEA is something that can be done over and over, so this was not the issue. Researching the best testing method for simulating an impact is what needed to be discovered. The most repeatable test on the frame of any vehicle to simulate that of real life impact is applied loads. The testing that we went through based on literary research was that of an applied load. Applied loads can be safe, easy to replicate, cheap, and have an overall proof of concept mathematics. These factors helped in the final decision n how to move our testing forward

1.3 Project Objective

The primary objective is to establish a reliable and cost effective test procedure, with a secondary goal of being able to recommend a frame with reduced weight. Figure 1 shows the Solidworks model for the Zips Baja 2019 frame design, the frame tubes highlighted in red will be the focus of this analysis. All tested frame designs were designed within Baja SAE rules, and with subsystem requirements in mind.

The frame is one of, if not the most complex designs on the vehicle, therefore it gets a lot of attention. Being that it has the most rules to regulate it, it is the heaviest part of the vehicle, as well as being the main mounting point for every other subsystem on the vehicle. The frame must be ergonomically designed to fit the 95th percentile male down to the 5th percentile female. However, the main requirement of the frame is to maintain a minimum space around the driver while sustaining its integrity during normal operation, collisions, or roll overs. It is also the most expensive and has the longest lead time on the vehicle, with its fabrication process being very labor intensive.

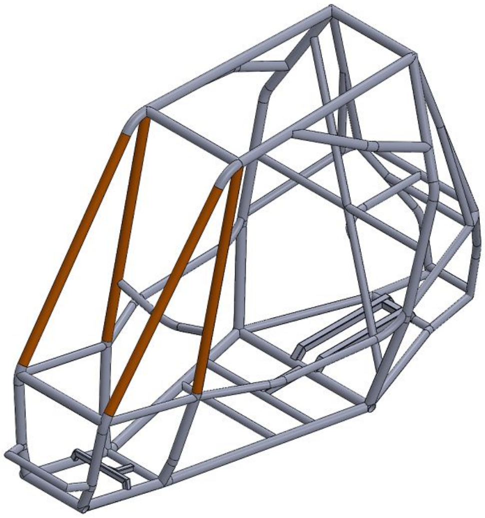


Figure 1: 2019 Full Frame Design

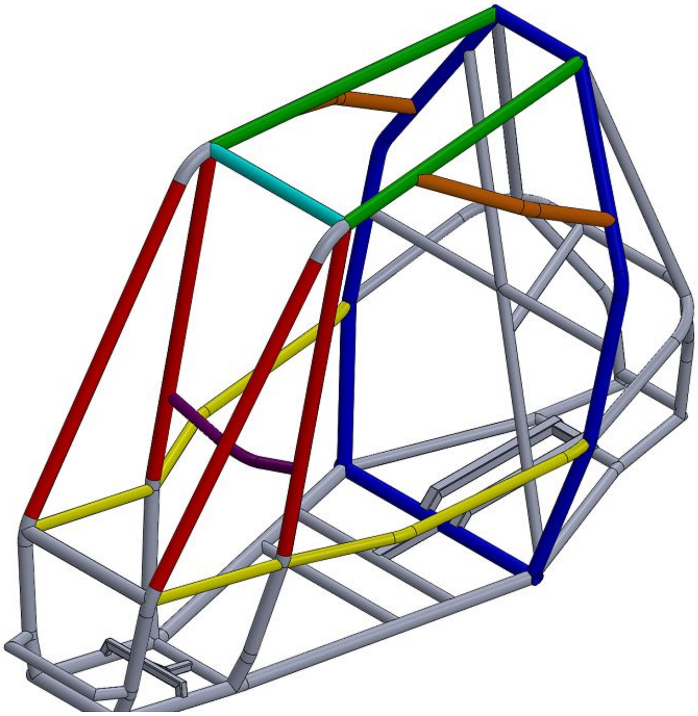


Figure 2: 2020 Frame with colored tube indicators

Tube	Color Indication
Roll Hoop Overhead (RHO)	Green
Side Impact Members (SIM)	Yellow
Front Bracing Member (FBMup)	Red
Gusset	Orange
Steering Column Mount	Purple
Firewall	Blue
Lateral Cross Member (CLC)	Teal

Chapter 2: Original Design

2.1 Product Definition

The original design for the Baja frame consists of two bracing members at the front. These members are highlighted in red in Figure 2 above. In the current design the front brace is encompassed in the Roll Hoop Overhead (RHO). The secondary bracing member is referred to as the Front Bracing Member Up in the rulebook (FBMup). This original design is very robust and safe; however both tubes are not required by the rules and the additional brace is heavy and adds up to an additional five pounds of weight to the vehicle. This vehicle chassis was optimized for safety and its mounting points of various components. As well as allowing for a quick, and rules compliant fix in case of a rollover. This frame will be the baseline for the tests and will ultimately be used as an indication for if the team can reduce the overall amount of tube that creates the chassis for further years.

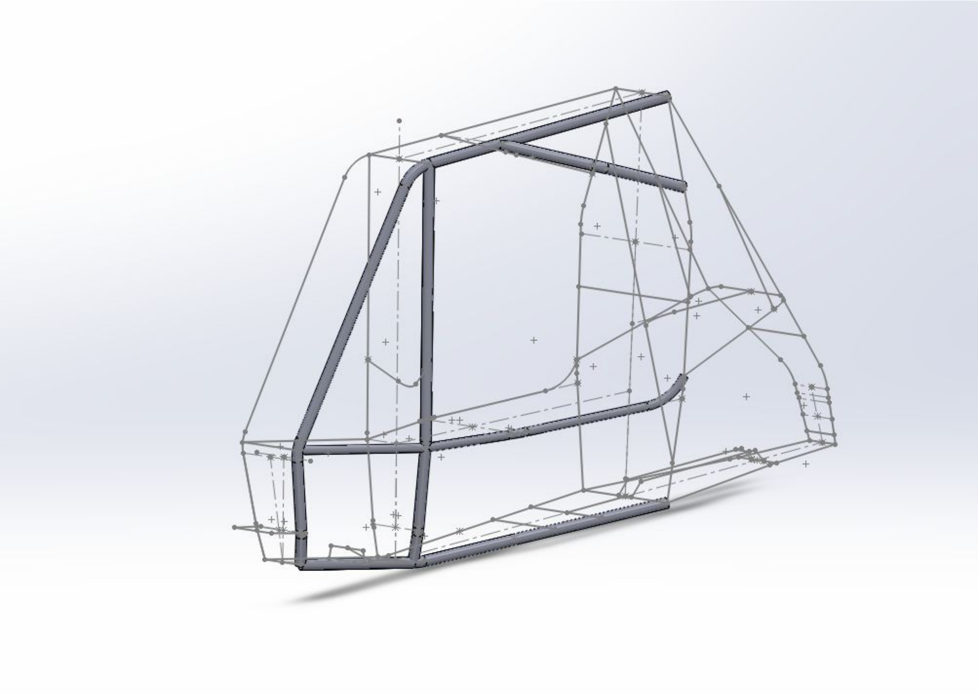
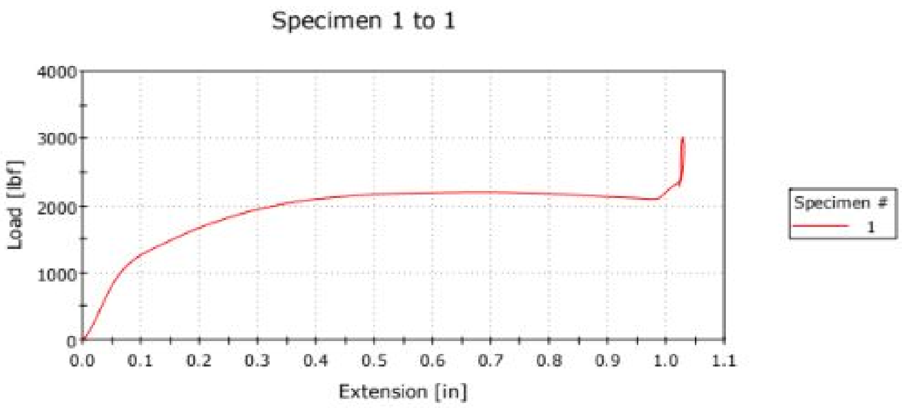


Figure 3: Original Tube Frame Design

2.2 Testing Validation

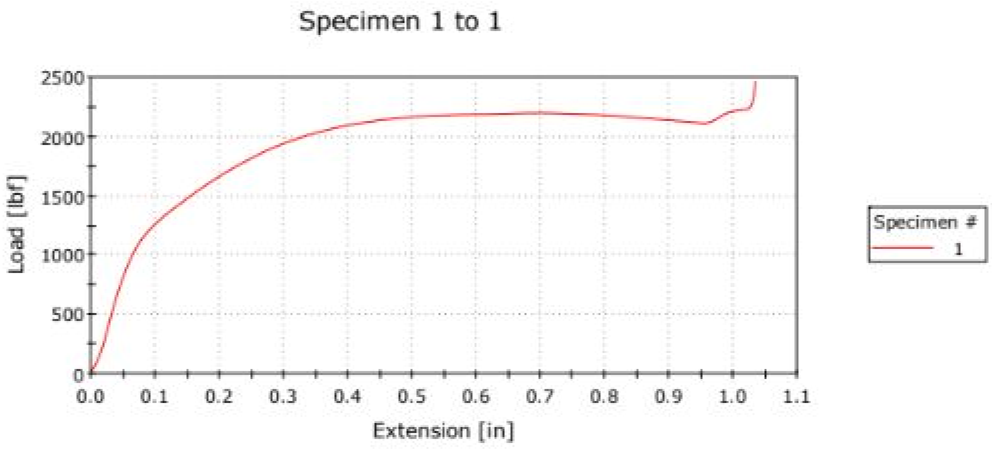
The first concept of real-life testing after our mathematical calculations came in the form of applied load tests on small bumper frames. There were four small bumper frames welded and then tested through an applied load machine. The four bumpers were set up in the instron machine and then slowly applied the load at different rates to determine if the overall mathematics of load forces works the same as the time of the load is reduced.



	Compressive strain (Extension) gauge length [in]	Time at removal [s]
1	3.93701	-----

Figure 4:

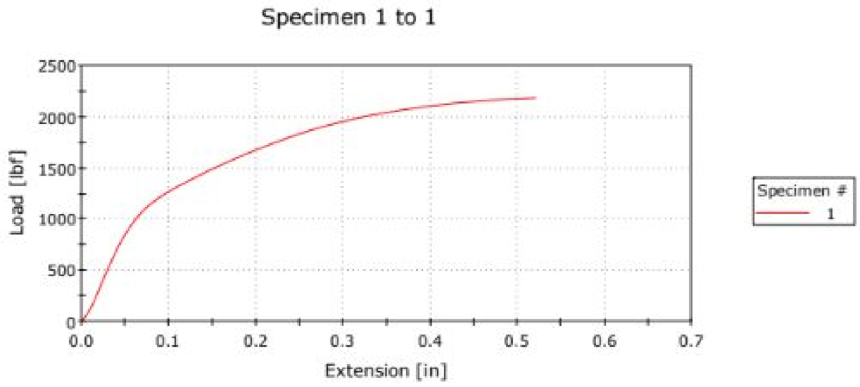
In the above Figure 4, the rate of load was applied at 250 lbf/minute. This rate was a low base line so we could tell the reaction of the applied force on the beams. The results of the graph show that right around when the 2000 lbf is applied, the bumper started to have plastic deformation and give way.



	Compressive strain (Extension) gauge length [in]	Time at removal [s]
1	3.93701	-----

Figure 5:

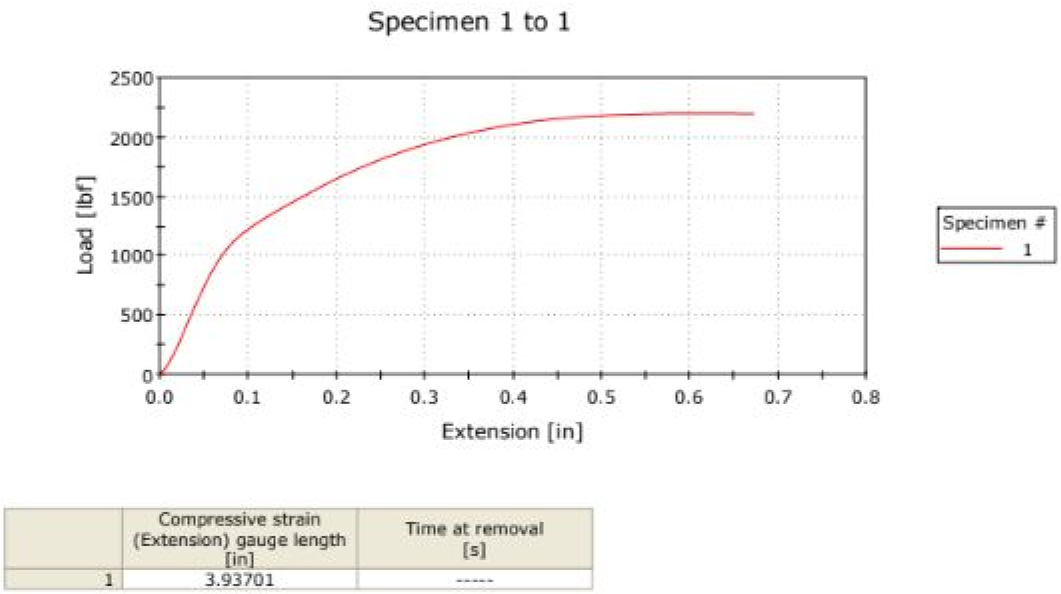
In the above Figure 5, the rate of load was applied at 1000 lbf/minute. The slow nature of the first test made us want to see how the bumper beam would react with a much higher load rate. The results of the load extension graph reflect that the load capacity was around 2000 lbf before plastic deformation occured.



	Compressive strain (Extension) gauge length [in]	Time at removal [s]
1	3.93701	-----

Figure 6:

In the above Figure 6, the rate of load was applied at 5000 lbf/minute. The last two tests showed the applied force mimicking much closer to that of an actual impact. The force held was once again around 2000 lbf.



	Compressive strain (Extension) gauge length [in]	Time at removal [s]
1	3.93701	-----

Figure 7:

In the above Figure 7, the rate of load applied was at 10,000 lbf/minute. The final bumper test showed that the 2000 lbf was consistent with all of our tests and reflected the mathematics behind our applied load tests. The results of bumper tests really validated our initial findings and helped us feel confident moving forward with our testing.

Chapter 3: Frame Concepts and Testing

3.1 Design Descriptions

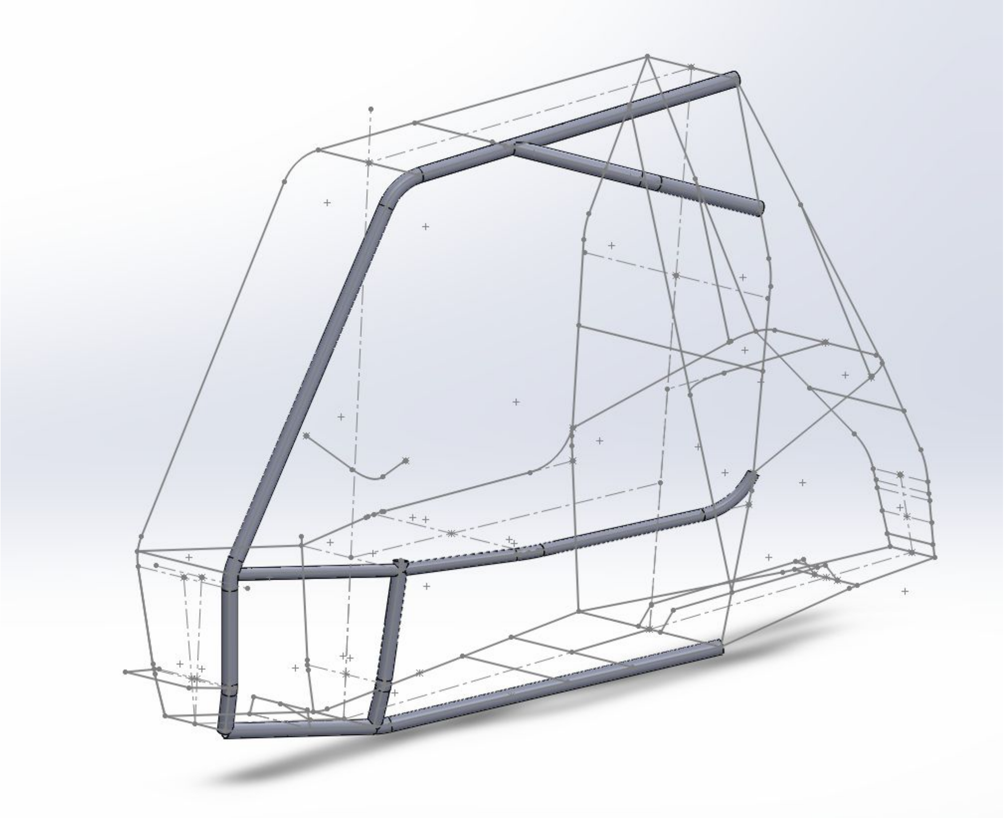


Figure 8: Front Tube Frame Design

This is the first of three iterations being tested. The above iteration depicts the maximum possible angle that the FBMup and RHO could be at consisting of a single front brace. There are a few benefits to having the bracing at the maximum angle, these include easier egress, more clearance for the drivers feet, and more protection for components mounted in the front box. Egress is the test done during technical inspection that requires a driver in full gear and safety harnesses to completely exit the plane of the cockpit in under five seconds. With the second bracing member removed there is more room for the driver to be able to exit the vehicle which would improve their egress times and ensure a safe exit.

One of the rules provided by SAE is that the driver's feet must be enclosed by the tubes that makeup the front box. The maximum angle provides a slight advantage by lateral crossmember at the front of the box being able to be raised slightly for extra clearance. The final benefit of the maximum angle is there is a reduced risk of breaking the various components that are mounted to the SIMs in the front box. In the event of a rollover the tube will protect all of the components in the front such as brake reservoirs. If one of the other iterations is selected there is an increased risk of an object hitting and puncturing the brake reservoirs or damaging other components. The maximum angle iteration also has some downsides that are associated with it. One of the major downsides would be that the steering column mount would have to be completely redesigned since the second FABup member is not there to be mounted to. Another downside is during wheel to wheel racing there is a higher chance that an opponent's tire can enter into the cockpit and affect the driver. There is also a higher possibility that an object will make contact with the driver in the event of the vehicle tipping over.

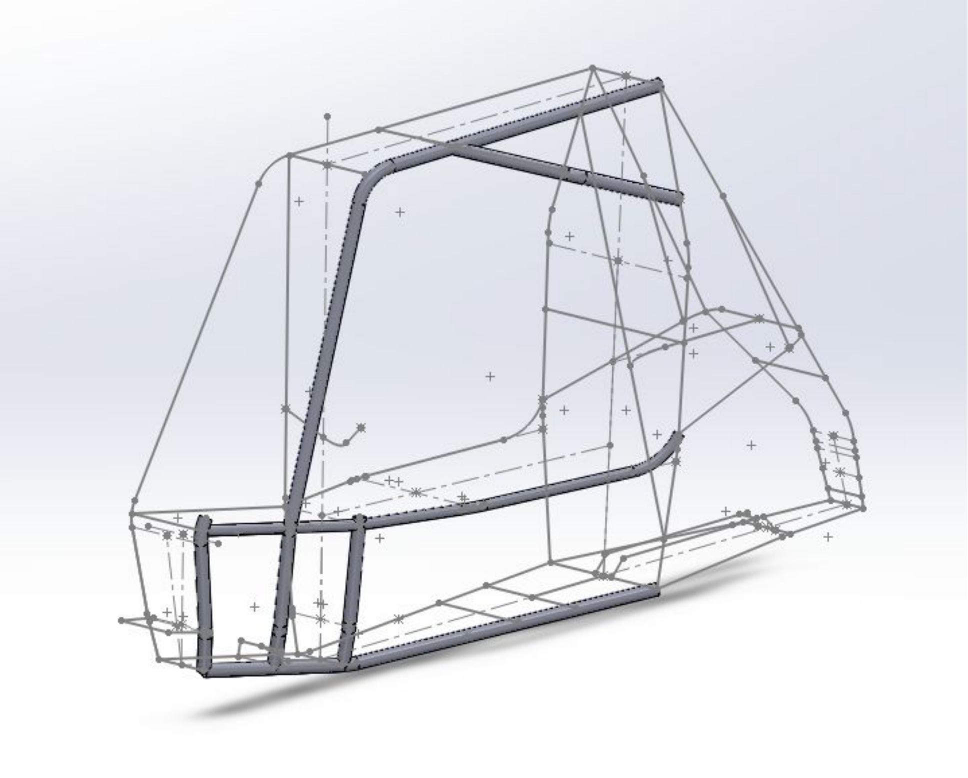


Figure 9: Medium Tube Frame Design

The second iteration would provide data for the angle that is in the middle of the maximum and minimum possible angles. One of the benefits of this bracing is the torsional rigidity of the vehicle will increase. With more tubes added as bracing in the front box a larger force will be needed to produce the same amount of deflection in the front. The additional tube also provides more mounting points for various aspects of the vehicle. One such aspect would be an improved shock mount. The FBMup would provide more vertical placement options to the suspension designer to optimize the compression in the shocks which would improve overall vehicle performance. Similar to the maximum angle iteration, this iteration also provides more clearance for egress.

With the benefits of this change there are also a few negative aspects. There is potential that the new tube in the plane of the front box will interfere with the steering pinion. The CLC placement also has to be taken into account. According to rule B.3.2.8 (Roll Hoop Overhead members) the CLC is required to be at least 12 inches in front of a line that is 4 inches from the back of the seat. This is due to wanting ample head clearance in case of roll over so there is no contact between a foreign object and the driver's head. This iteration also causes the team to create a new steering column mount design. However this iteration provides more mounting options and simpler design considerations than the maximum angled design. The second iteration is also the heaviest option of the three single tube designs. The additional tube weighs 0.75 pounds which is still lighter than the original dual braced chassis.

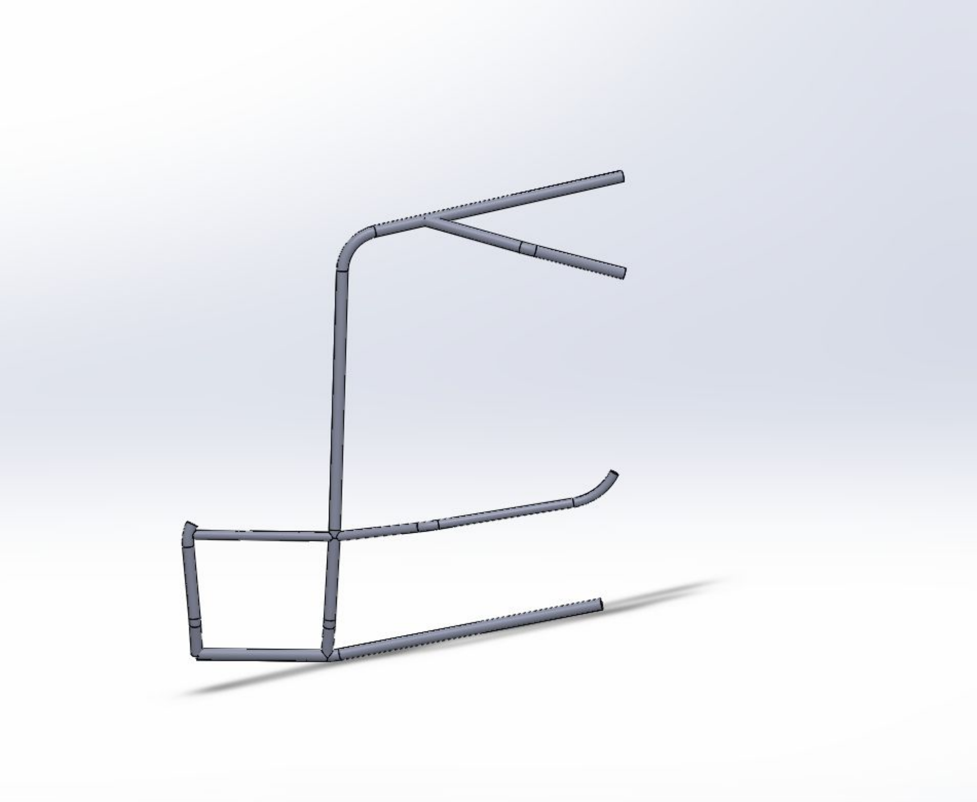


Figure 10: Minimum Tube Frame Design

The third and final iteration that was designed to be tested was the minimum possible angle. This iteration is known as a nose car in the Baja design series.

The nose car is the most compact and lightweight vehicle chassis design out of the previously mentioned iterations. This iteration also has the least impact on the other subsystems. There are no clearance issues similar to the pinion contact in the second iteration. There is also no need to completely redesign the steering column mount since the same mounting points are still in place. The mount will just have to be adjusted to each new year's geometry. Another benefit is this design limits the possibility of another driver's tire entering the cockpit and affecting the driver. In previous years drivers have had issues with a tire entering the cockpit while racing wheel to wheel with another driver and making contact. This iteration limits the space that a tire can enter while still providing ample space to egress in the required time. A drawback for the minimum bend angle is that there is less protection for the front box in the event of a roll over. The brake reservoirs and other components would have a higher probability of being impacted with this iteration due to there not being a bracing member to protect the components.

3.2 Real-Life Testing

In order to test the four samples of the frame members in the most accurate way possible an accurate force needed to be set which would be similar to what the car would see in a race environment. A common deformation that is seen on the car, especially after an accident, is on the front bumper on the car. Four samples of the front bumper on the car were fabricated and put under an Instron machine seen below in Figure 11. These test pieces were subjected to a force until they yielded. The average force the bumper samples were able to hold up to was 2000lbf this was then assigned to be applied to the four frame samples.

Figure 11: Instron 5569 11k Machine Located in the Civil Engineering Labs of ASEC

The 2000lbf was applied from an overhead hydraulic cylinder that utilized a load cell and an electron position sensor. This made it simple to plot the deflection vs force applied. The four samples were TIG welded under the same conditions that the complete car frame would be welded under. A fixture was fabricated to hold the frame samples at the same angle used in the FEA testing. This fixture held the samples inline with the hydraulic cylinder used to apply the force to the samples. The Instron machines in ASEC labs were large enough to fit the sample frames. The large 300Kip press in the turbine build was utilized to fit the four samples in the test area. The press is shown below in Figure 12.

Figure 12: The 300Kip Press in the Gas Turbine Testing Facility

Each sample was tested to its max capacity and deflection that they could withstand which was recorded through the load cell and position sensor on the cylinder. The frame samples were then measured after deflection and compared to the FEA results. A measuring jig was set up to compare the before and after measurements of each sample in order to gain a deflection value after the force was applied. Pictured below are the four samples after the testing was completed. The deflection is difficult to see, but it shows the set up used to test each sample.

Figure 13: Original Tube Frame After Test

Figure 14: Front Tube Frame After Test

Figure 15: Medium Tube Frame After Test

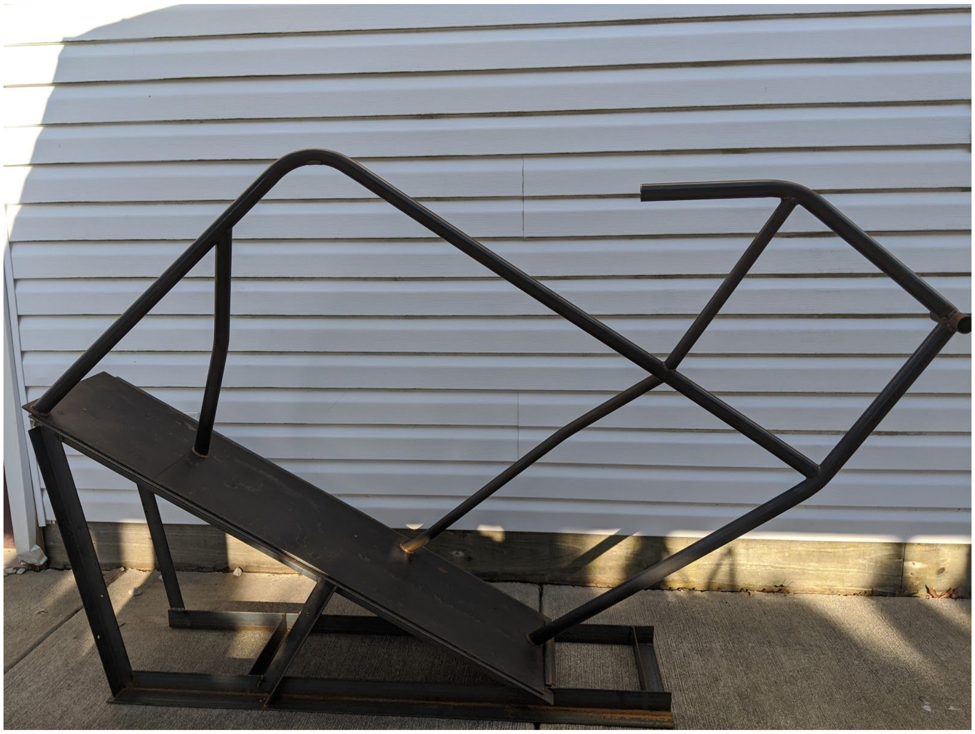


Figure 16: Minimum Tube Frame After

Figure 17: Test Setup Performed in Gas Turbine Testing Facility

With more time and available resources, a set of sensors and equipment that could have mapped the deflection of the frames on a digital layout would have resulted in more accurate measurements. This would have made for a more accurate relationship to FEA testing in Solidworks. Due to the current condition the samples were measured and tested in the most efficient and accurate way that possible.

3.3 Finite Element Analysis

All of the frames were designed as half frames, to not only lessen the cost and lead time of the prototypes but also so assumptions could be made that allowed the Finite Element Analysis (FEA) to be as close to the real-life testing as possible. Some of those assumptions are that the firewall can be considered rigidly fixed therefore no frame members behind the firewall need to be considered for this project. As well as half of the frame, before the firewall, would act similarly to the full frame while being tested. Finally, the tube at the radius of the FBMup and the RHO was split so the force applied would closely match the impact seen when testing in real-life.

From there the Solidworks simulation could be set up for the original tube frame. First, the fixed points were set, shown in Figure 18 by the green arrows. The fixed members were the SIM, gusset, RHO, and at the tubes at the bottom of the frame. Then the force determined with the tube testing, 2000lbf, was applied to the radius of the FBMup. Shown in Figure 18 by purple arrows. After that a mesh can be added to the tubes, finer mesh for a more accurate result.

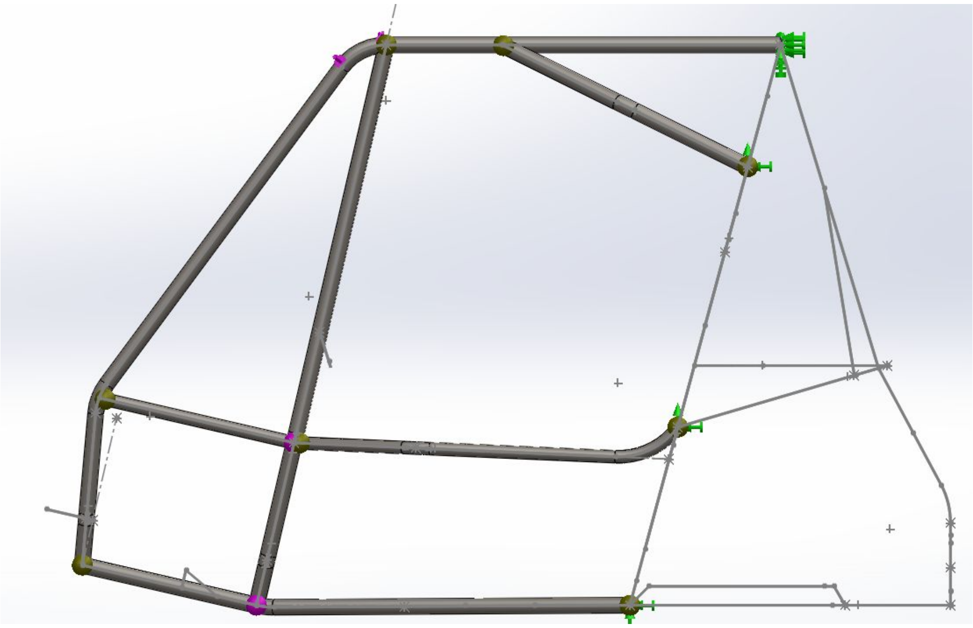


Figure 18: Original Tube Frame Simulation Set-up

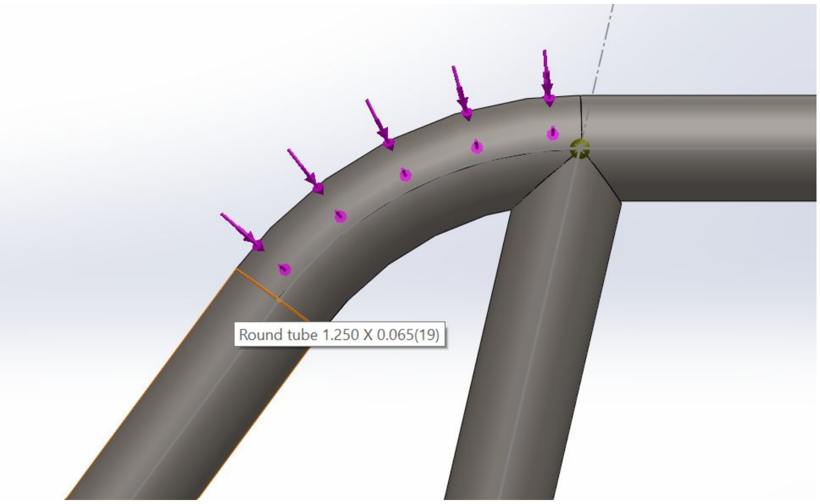


Figure 19: Applied Force Location on Original Tube Frame

The results of the FEA simulation can be seen in Figure 20 below. The dramatized deflection was chosen to show how the entire frame moves during the impact. The scale to the right showing red as the furthest deflection to blue being the least. The overall resulting deflection for the Original Tube Frame was 0.7694in.

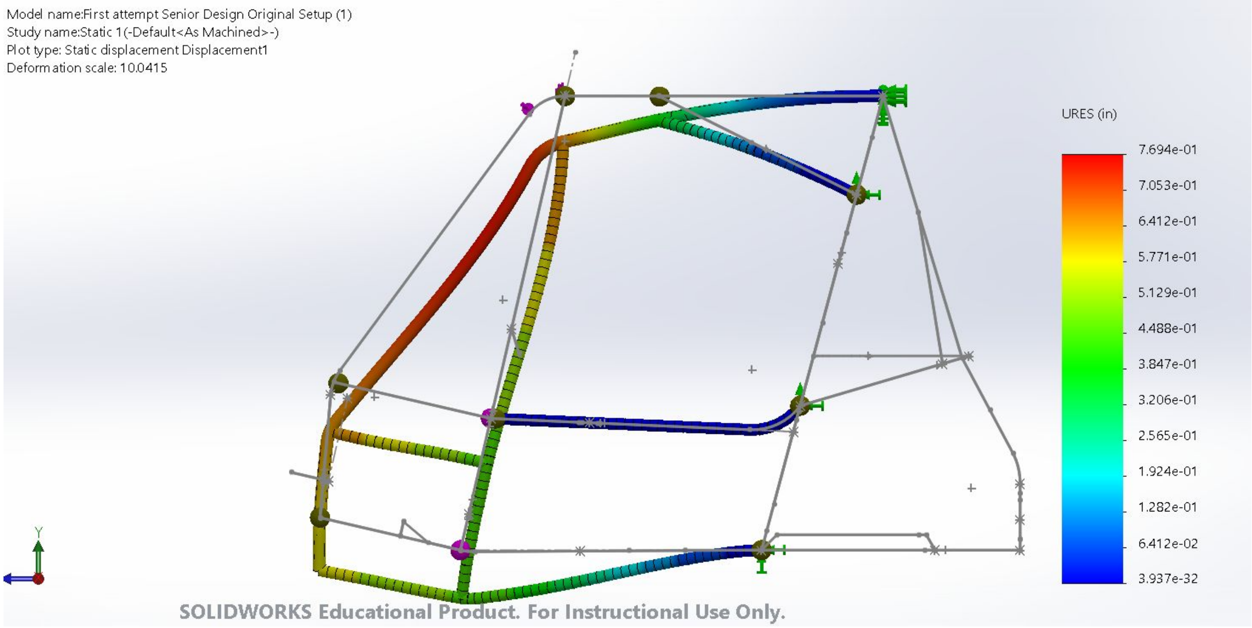


Figure 20: Original Tube Frame FEA

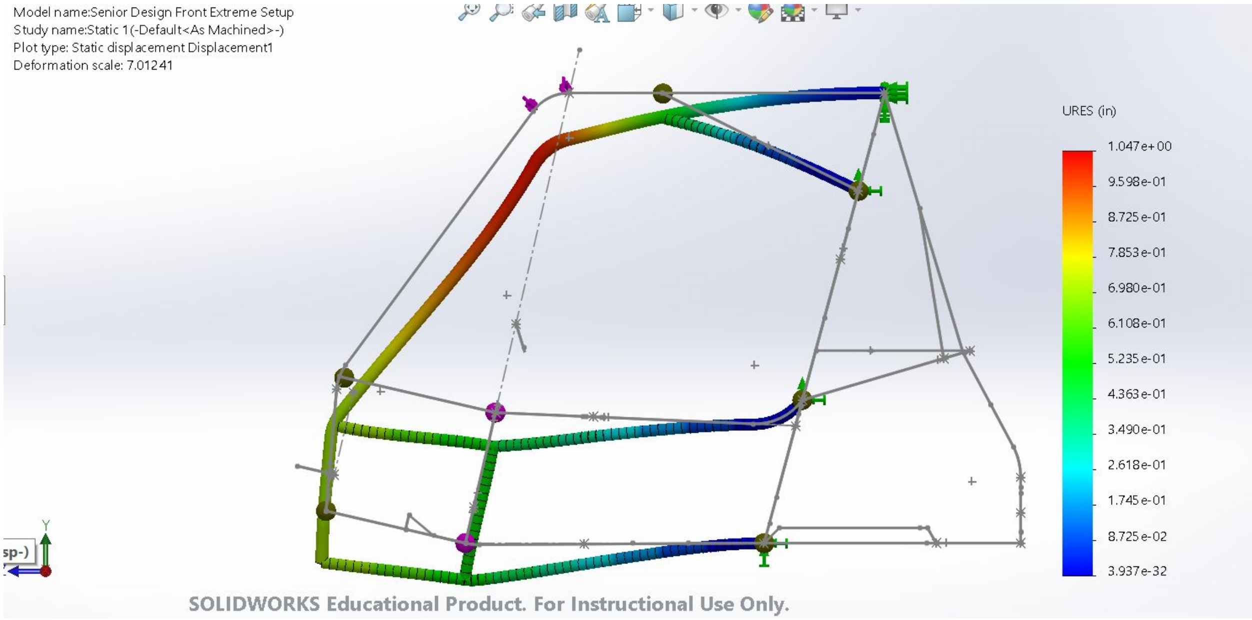
The Solidworks simulation set up for the three trial frame designs was the same as the Original Tube Frame. All frames were fixed at the same points, and loaded in the same manner. After real-life testing showed that the trail frames would not withstand the full 2000lbf adjustments were made to the FEA. Using the assumptions previously made, and the forces found in the real-life testing of the trail frames resulted in the following deflections seen in Table 1.

Frame Style	Impact Force (lbf)	Overall FEA Deflection (in)
Original Tube	2049.932	0.769
Front Tube	1719.312	1.047
Medium Tube	1650.463	0.819
Minimum Tube	1499.209	0.516

Table 1: Solidworks FEA Deflection of all tube frames

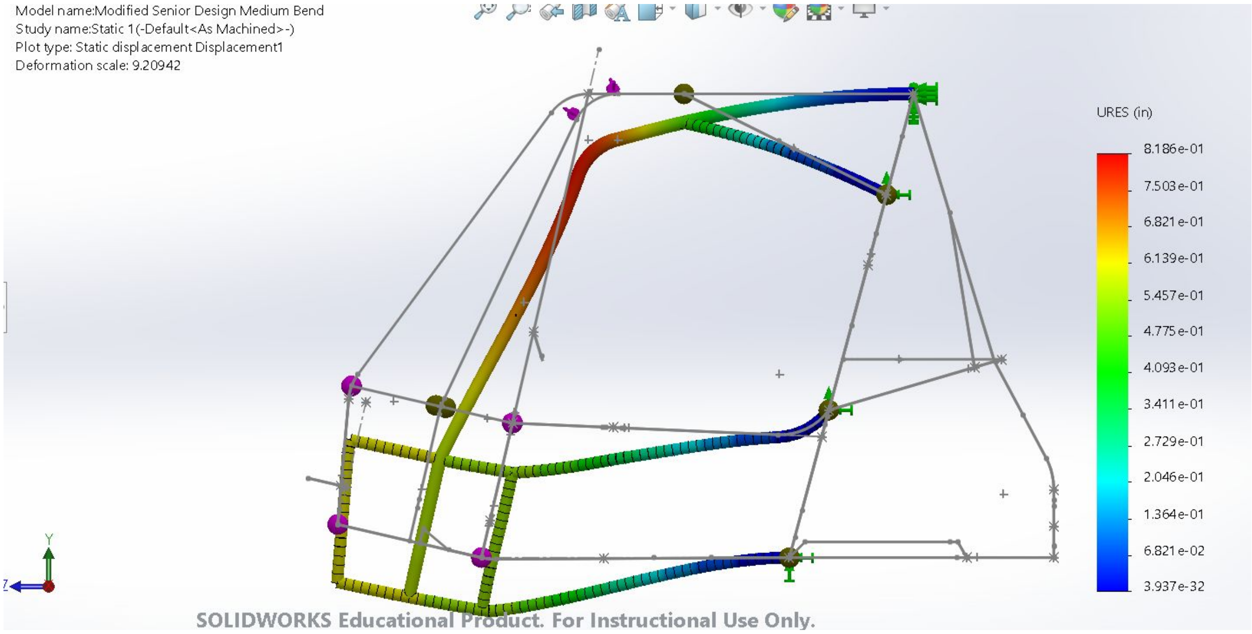
Though loaded with less force, the Front Tube Frame deflected the most between all frames, making it clear that the extra tube in the Original Tube Frame is highly important to the structure of that frame. Being that the Original Tube Frame was able to take the highest load, and deflect the second least. Dramatized frame deflections can be seen in Figures 20 through 23. All trial frame's dramatized FEA models showed deflection in the same manner as the real-life frames.

There were no separations or broken welds in real-life, and the same was seen in the FEA simulations. This is important because during competition in the case of an impact to the vehicle the vehicle's frame is checked, all frames would have been cleared to rejoin competition. It is also important in showing accuracy of the FEA simulation procedure.



Model name:Senior Design Front Extreme Setup
Study name:Static 1(-Default<As Machined>-)
Plot type: Static displacement Displacement1
Deformation scale: 7.01241

Figure 21: Front Tube Frame FEA



Model name:Modified Senior Design Medium Bend
Study name:Static 1(-Default<As Machined>-)
Plot type: Static displacement Displacement1
Deformation scale: 9.20942

Figure 22: Medium Tube Frame FEA

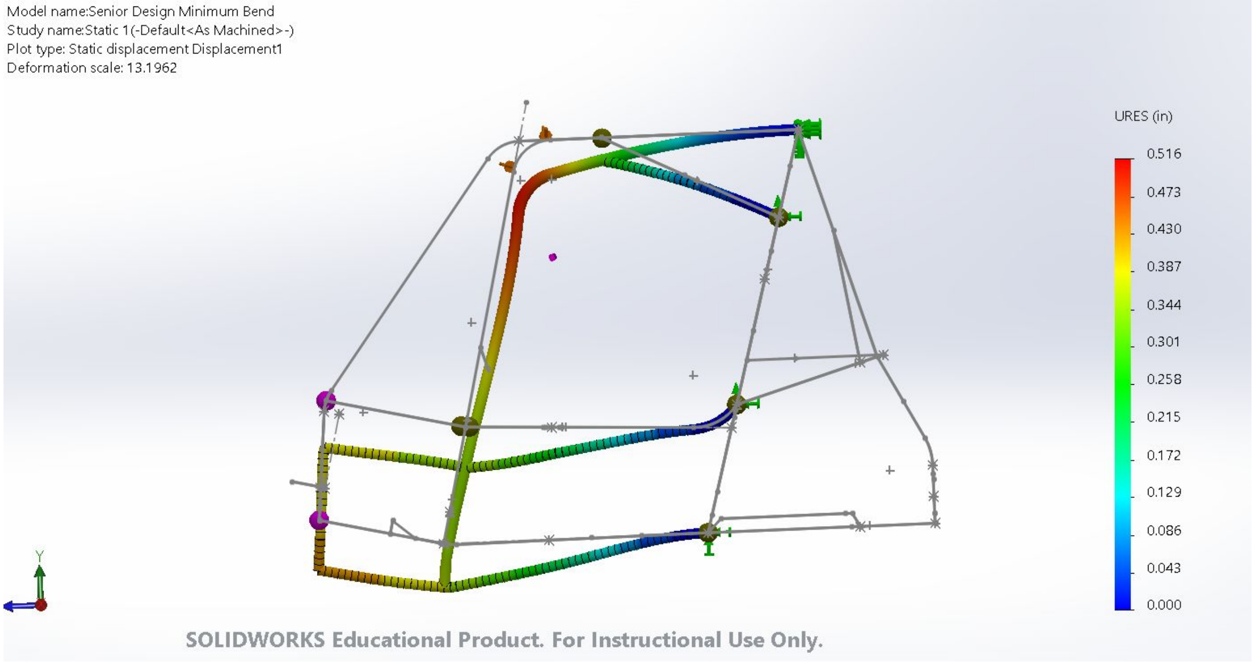


Figure 23: Minimum Tube Frame FEA

Chapter 4: Conclusions

4.1 Error Analysis

The average percent error when comparing the real-life deflection to the Solidworks FEA for all four frames was 33%. The breakdown of the percent error for each frame can be seen in Table 2. It is important to note that real-life testing and FEA simulation will never be exact. The majority of the discrepancies between the real-life and the FEA comes down to the manner in which the real-life deflection was measured. The inability to place sensors on the frames when impacted made an apparatus to measure the deflection after testing a necessity. Though the measured deflection was done in an accurate way, it was only able to measure in a two dimensional plane and the Solidworks simulation is measuring the overall 3 dimensional deflection.

Frame Style	Real-life Deflection (in)	Overall FEA Deflection (in)	Percent Error (%)
Original Tube	1.023	0.769	24
Front Tube	1.525	1.047	31
Medium Tube	1.225	0.819	33
Minimum Tube	0.925	0.516	44

Table 2: Percent Error Analysis for All Trial Frames

4.2 Conclusion

After the bumper validation, real-life testing, and Solidworks FEA simulation the average 33% error between the real-life frames and the FEA simulation frames is seen as validating the process. The process began with validating the mathematics with bumper testing to confirm the force that would be applied in the real-life testing and the FEA. Then the frames were designed and manufactured based on assumptions made. From there, real-life testing and FEA could be performed. It is believed that the Solidworks FEA simulation setup and procedure is accurate enough to use the simulations to test future trial frames.This will allow multiple frames to be tested as fast as they can be designed.

Frame Style	Overall FEA Deflection (in)
Original Tube	0.769
Front Tube	1.218
Medium Tube	0.992
Minimum Tube	0.689

Table 3: Overall FEA Deflection of All Four Frames Impacted with 2000lbf

For future recommendations all frames were subjected to the FEA simulation process at 2000 lbf. Table 3 shows the FEA deflection of all three trial frames. It is seen that the Minimum Tube Frame would have the least overall deflection, and it is recommended that further simulations are run with close variations of the Minimum Tube Frame. With increased time and testing possibilities, the FEA to real life tests could show even better results. Through the data gathered and the deflection shown in the Minimum Tube Frame, it can be shown that future tests should be performed using this Frame. This variation gives the best viable option for future frame capacities. The safety of the team and weight reduction can be possible with more real life tests performed on the Minimum Tube Frame variation. As previously stated, the safest frame so far is the current design. If the team would perform more future testing, the tube durability that is sacrificed can be justified through the reassurance of safety, shown in future test results.

References

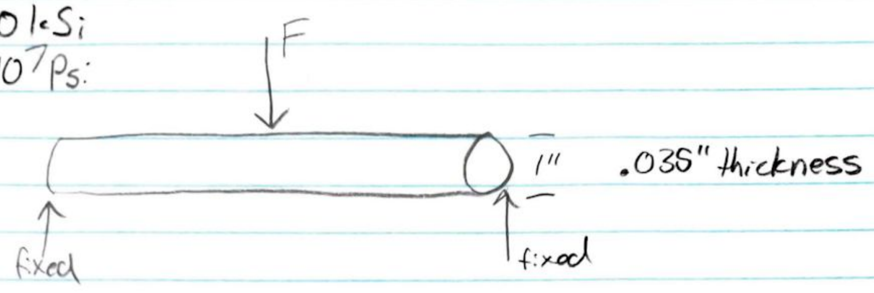
1. Baja SAE 2020 Collegiate Design Series Baja SAE Rules. (2019, September 8). Retrieved September 9, 2019, from https://www.bajasae.net/cdsweb/gen/DocumentResources.aspx
2. Hoffpauir, D. (2016, December 29). FEA - Not All Beautiful Color Plots are Precise or Accurate. Retrieved from https://www.nasa.gov/offices/nesc/articles/finite-element-analyses
3. Further References for design work, concept designs, testing, and data collection are commented and thanked in our acknowledgement section.

Appendices

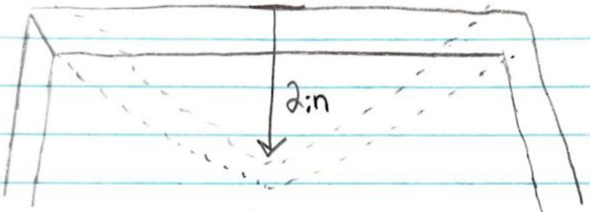
Appendix A: Hand Calculations

Force to yield a 1in tube .035 thickness by 2in

$E = 29700\ \text{ksi}$
$= 2.97 \times 10^7\ \text{Psi}$



$F = MA$



$$MI = \frac{(\pi(OD^4 - ID^4))}{64} \qquad \delta = \frac{L^3 F}{3\,E\,MI}$$

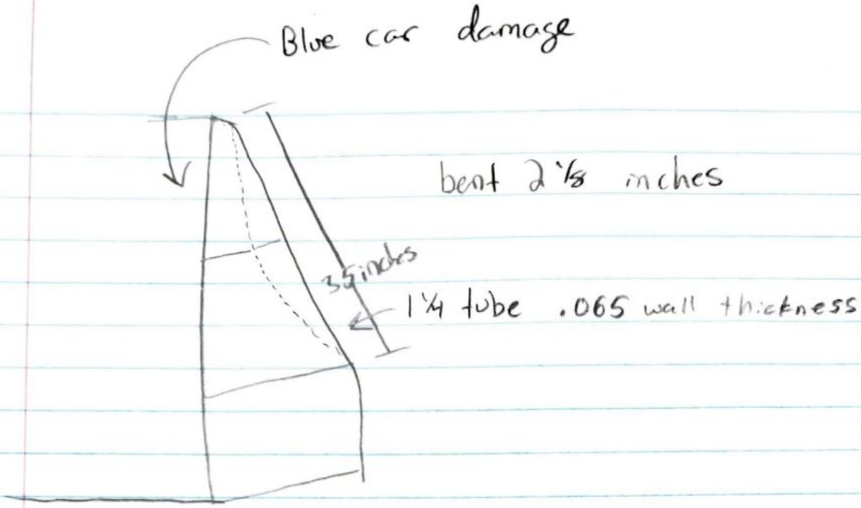
$$\text{Bending Stress} = \frac{(FL)}{\left(\frac{MI}{.5H}\right)}$$

$$MI = \frac{(\pi(1^4 - .93^4))}{64} = .012367468$$

$$\delta = \frac{L^3 F}{3E\,MI} \rightarrow \frac{3EMI(\delta)}{L^3} = F$$

$$F = \frac{3(2.97 \times 10^7\ \text{Psi})(.012367468)(2.25\text{in})}{(14\text{in})^3}$$

$$\boxed{F = 903\ \text{lbs}}$$



$$MI = \frac{(\pi(OD^4 - ID^4))}{64} \qquad \delta = \frac{L^3 F}{3 E MI}$$

$$MI = \frac{(\pi(1.25^4 - 1.12^4))}{64} = .0426022981 \text{ in}^4$$

$$F = \frac{3 E MI \delta}{L^3} \rightarrow \frac{3(29700 \text{ ksi})(1000)(.0426022981)(2.125)}{(34 \text{ in}^3)}$$

$$F = 205 \text{ lb}$$

Impact force

$$F = \frac{mv^2}{2d}$$

$V = 15mph$

$= 6.7056 \, m/s$

$m = 145.15 \, kg$

$d = 10cm$

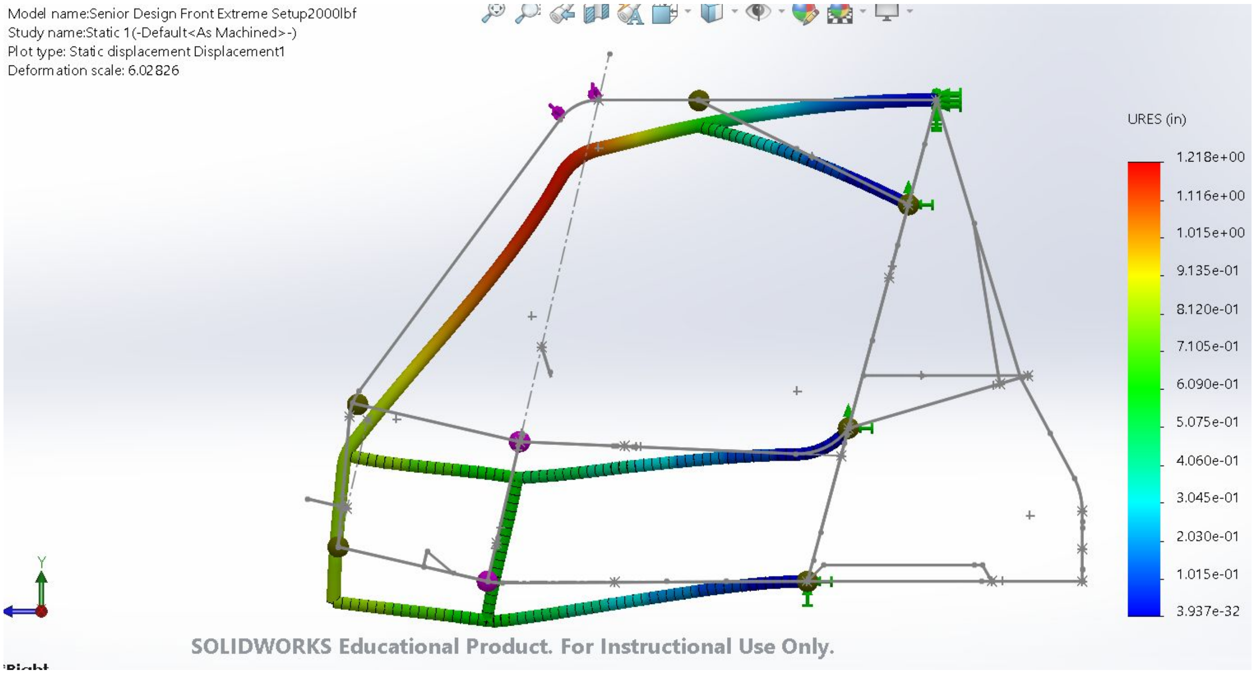
$= .10m$

$$= \frac{(145.15 \, kg)(6.7056 \, m/s)^2}{2(.10m)}$$

$$\frac{(145.15 \, kg)(44.965 \, m^2/s^2)}{.2m} \rightarrow kg \, m/s^2 = 32599$$

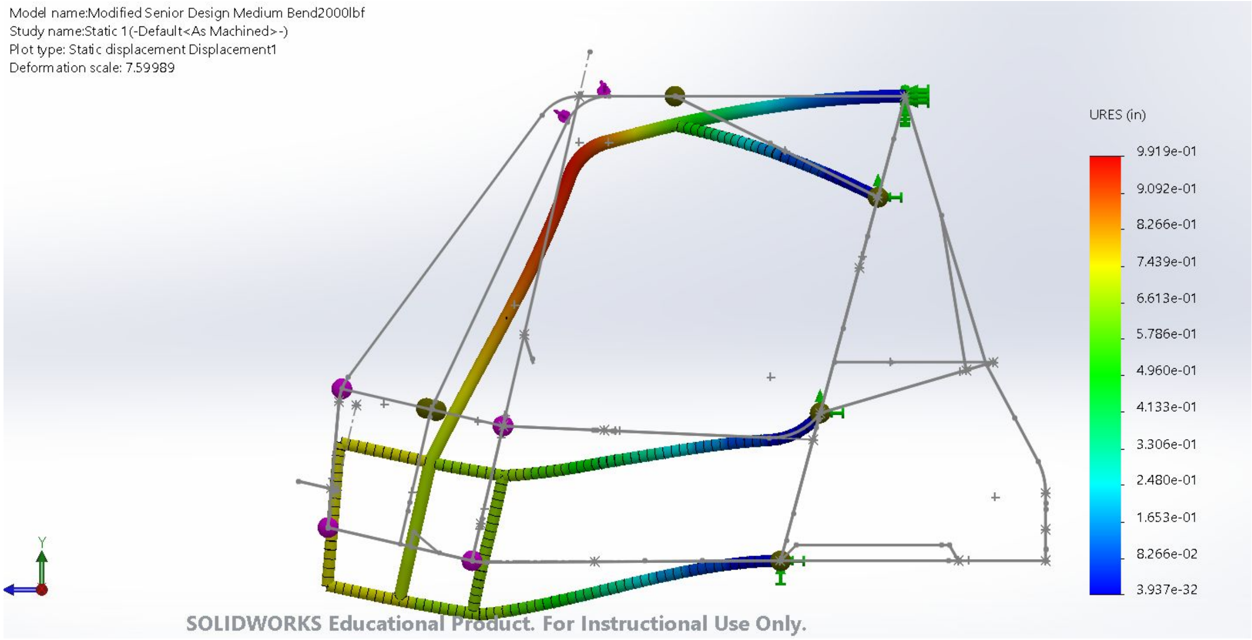
$32599 \, kg \, m/s^2$

Appendix B: FEA Simulation Results



Model name:Senior Design Front Extreme Setup2000lbf
Study name:Static 1(-Default<As Machined>-)
Plot type: Static displacement Displacement1
Deformation scale: 6.02826

Figure 24: Front Tube Frame FEA 2000lbf Load



Model name:Modified Senior Design Medium Bend2000lbf
Study name:Static 1(-Default<As Machined>-)
Plot type: Static displacement Displacement1
Deformation scale: 7.59989

Figure 25: Medium Tube Frame FEA 2000lbf Load

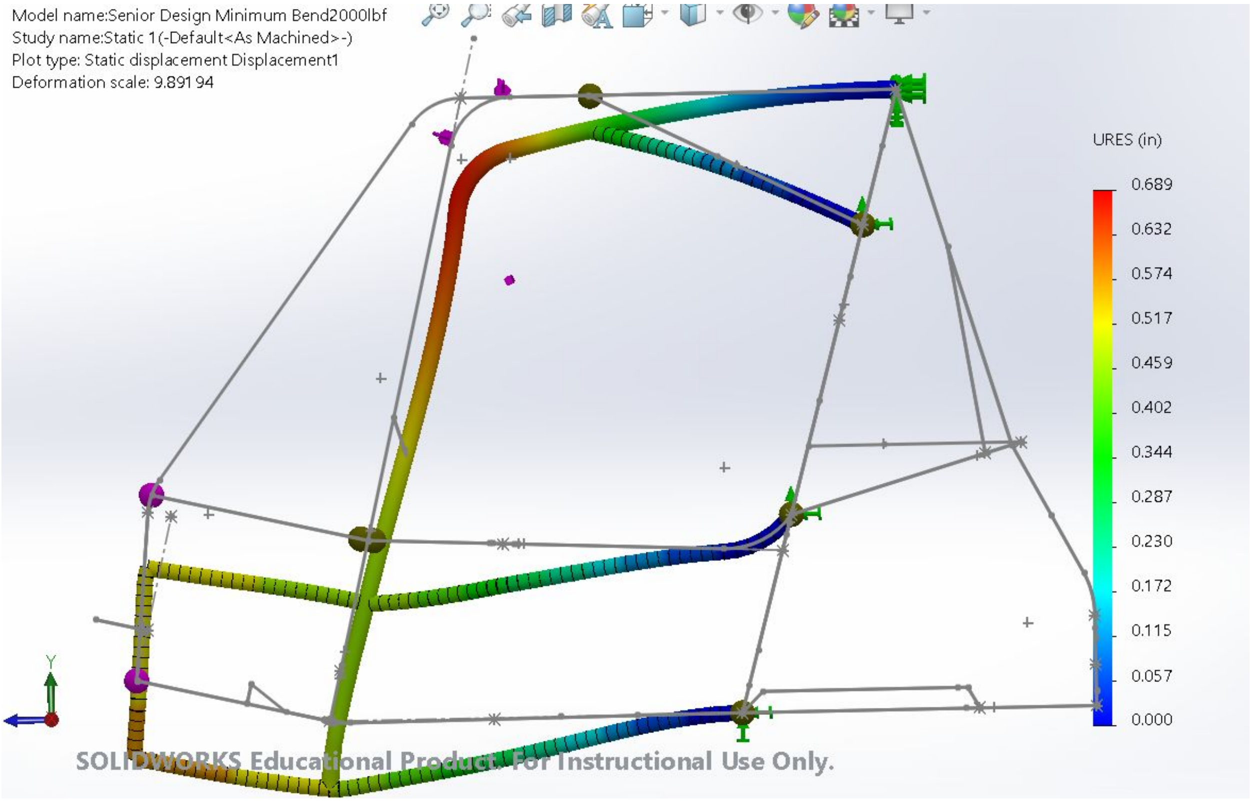


Figure 26: Minimum Tube Frame FEA 2000lbf Load

Appendix C: Group Member Participation Breakdown

	Subpart 1	Subpart 2	Subpart 3	Subpart 4	Subpart 5
Anthony Bodde	25	0	10	100	20
Nicole Fletterick	25	100	10	0	30
Alex Stanik	25	0	60	0	10
Jacob Swanson	25	0	20	0	40

Subpart 1: Conceptual design and overall brainstorming of project
Subpart 2: Finite Element Analysis
Subpart 3: Welding and Testing
Subpart 4: Solidworks rendering of 4 frame structures
Subpart 5: Report and Compiling of Final Data