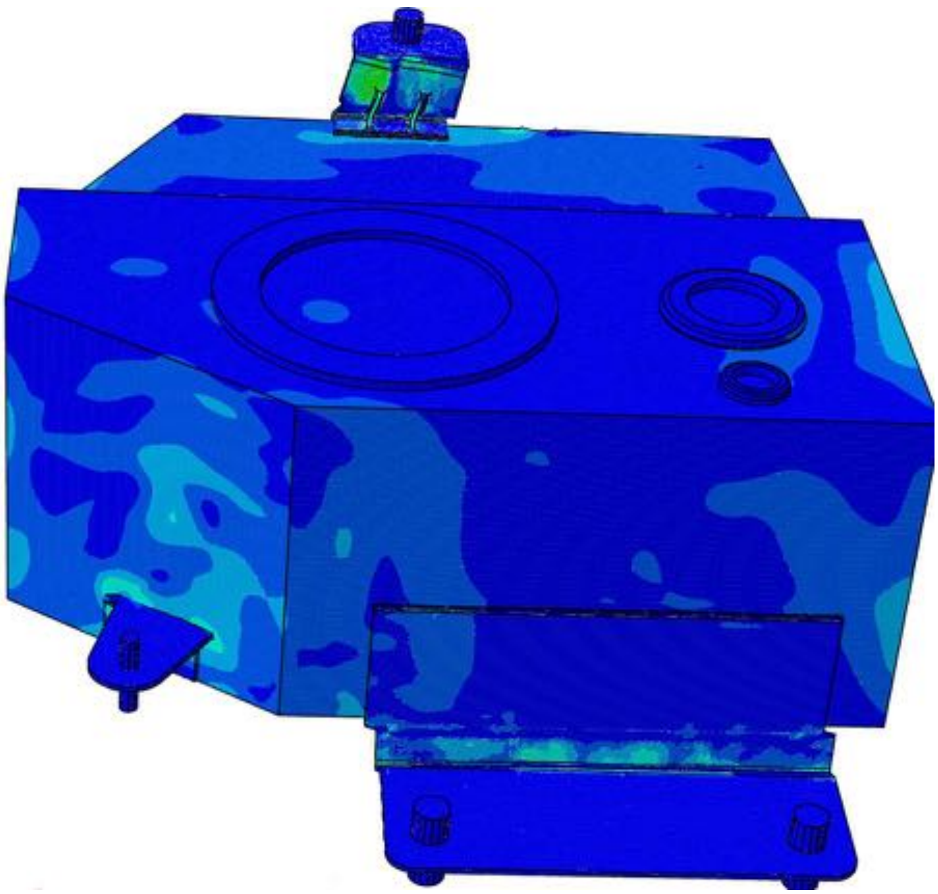
ME 461:
Finite Element
Analysis

Spring | 2016



The Development and Analysis of an FEA Simulation on a Vehicle Fuel Tank

Group Members:
Nathan Wallingford, Lance Clemmer

PennState
College of Engineering

Department of Mechanical and Nuclear Engineering, University Park, PA

Table of Contents

Table of Contents 2
Executive Summary 3
Acknowledgements 4
List of Figures 5
Section 1: Background and Project Plan 6
Section 2: Development and Description of the CAD Geometry 8
Section 3: Development of Finite Element Meshes 9
Section 4: Development and Description of the Model Assembly and Boundary Conditions . 11
Section 5: Development and Description of Model Interactions 13
Section 6: Analysis of Finite Element Model 14
Section 7: Summary of Major Findings 16
Section 8: Works Cited 18
Appendix A: Other Tests 19
Appendix B: Special Figures 20

Executive Summary

In modern vehicles, the fuel tank must not experience a catastrophic failure during a collision in order to ensure the safety of all passengers. In the past, fuel tanks have leaked causing fires and even explosions due to the highly combustible nature of fuel. To contain the risk, fuel tank mounts must be designed with extreme care by examining the stresses throughout the supports during extreme circumstances.

This report reflects the findings and approach to a Finite Element Analysis (FEA) of a student designed and fabricated fuel tank for the Penn State Advanced Vehicle Team (AVT). The AVT is involved with a national competition known as EcoCar, which aims to convert a 2016 Chevrolet Camaro into a hybrid electric vehicle. The competition organizers set strict rules and guidelines that all student built components have to abide by; the fuel tank for the car explicitly needs to be analyzed using FEA in order to validate its structural integrity.

The results of the FEA showed that the tank experienced stresses and deformation that were well beyond the yield stress of the materials used. In order to meet the loading requirements, the mounts must be redesigned in a way that reduces the stress to a suitable range. A different material with better properties as well as increasing the mount thickness would be sufficient in meeting the loading conditions.

Acknowledgements

The team would like to thank Dr. Reuben Kraft for assisting with Abaqus errors as well as helpful suggestions to improve loading conditions, constraints, steps and meshing.

List of Figures

Figure 1: Geometry of the hybrid fuel tank designed. 6
Figure 2: Fuel tank geometry with the direction of acceleration due to different collisions. 7
Figure 3: Fuel tank meshed with tri-type elements. 9
Figure 4: Front mount meshed with tetrahedral elements. 9
Figure 5: Close up of tank meshing. 10
Figure 6: Rear mount meshed attached to the meshed tank. 10
Figure 7: Boundary conditions for the tank mounts. 11
Figure 8: Acceleration due to gravity on the mass of the tank. 11
Figure 9: Acceleration due to rear-end collision. 12
Figure 10: Pressure due to impact from fuel. 12
Figure 11: Tying the brackets to the tank. 13
Figure 12: Spikes in stresses due to constraints. 14
Figure 13: Passenger side collision simulation. 15
Figure 14: Von Mises stress 15
Figure 15: Rear-end collision simulation. 15
Figure 16: Rear bracket during passenger-side collision 16
Figure 17: Rear bracket during a rear-end collision 16
Figure 18: Visualizing the displacement of the tank during passenger-side collision 17

Section 1: Background and Project Plan

The primary objective is to analyze the stresses of the supports that hold the fuel tank during the course of extreme changes in acceleration. The supports must be able to withstand significant loads to ensure that the mounting structures do not yield and create a possible hazard. Identifying the locations of the maximum stresses and the magnitude of those stresses will determine whether the design and the material is appropriate to handle the high changes of acceleration experienced during collisions.

The rules for the fuel tank state that in different collision situations, the material must not enter into the plastic region of deformation with a factor of safety accounted for. The horizontal loading is 20 G-force while the vertical loading is 8 G-force. All loading conditions must pass the FEA in order for the tank to be fabricated and used in the vehicle.The goal was not only to build a strong tank, but the team's intentions were to also make it lightweight so 5052-h32 aluminum was chosen over a stainless steel or other material. Figure 1 illustrates the geometry of the fuel tank, while Figure 2 shows the orientation of the tank in the vehicle as well as the impact situations.

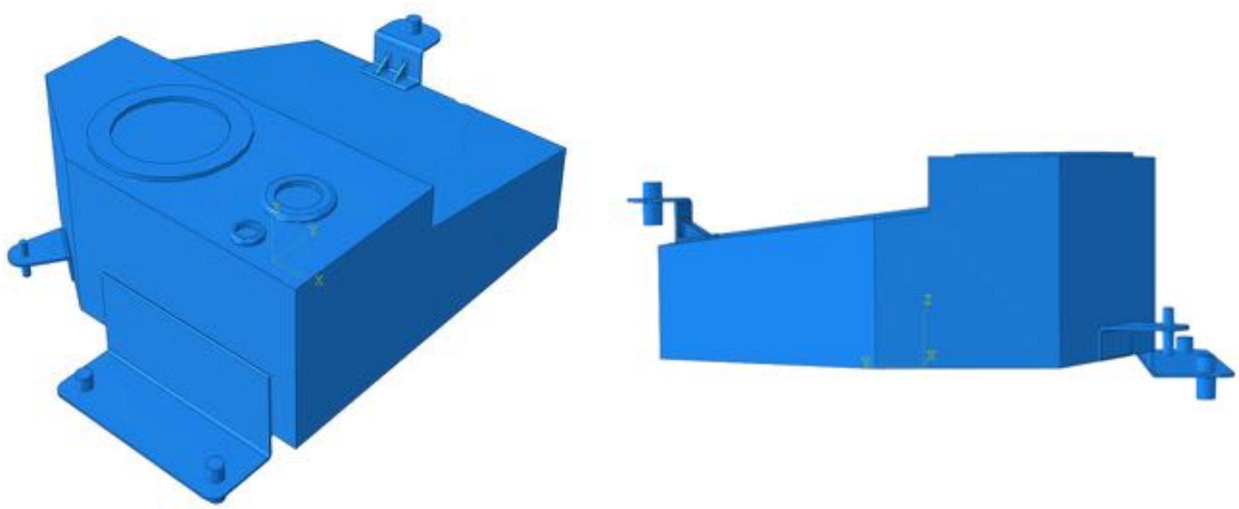


Figure 1: Geometry of the hybrid fuel tank designed.

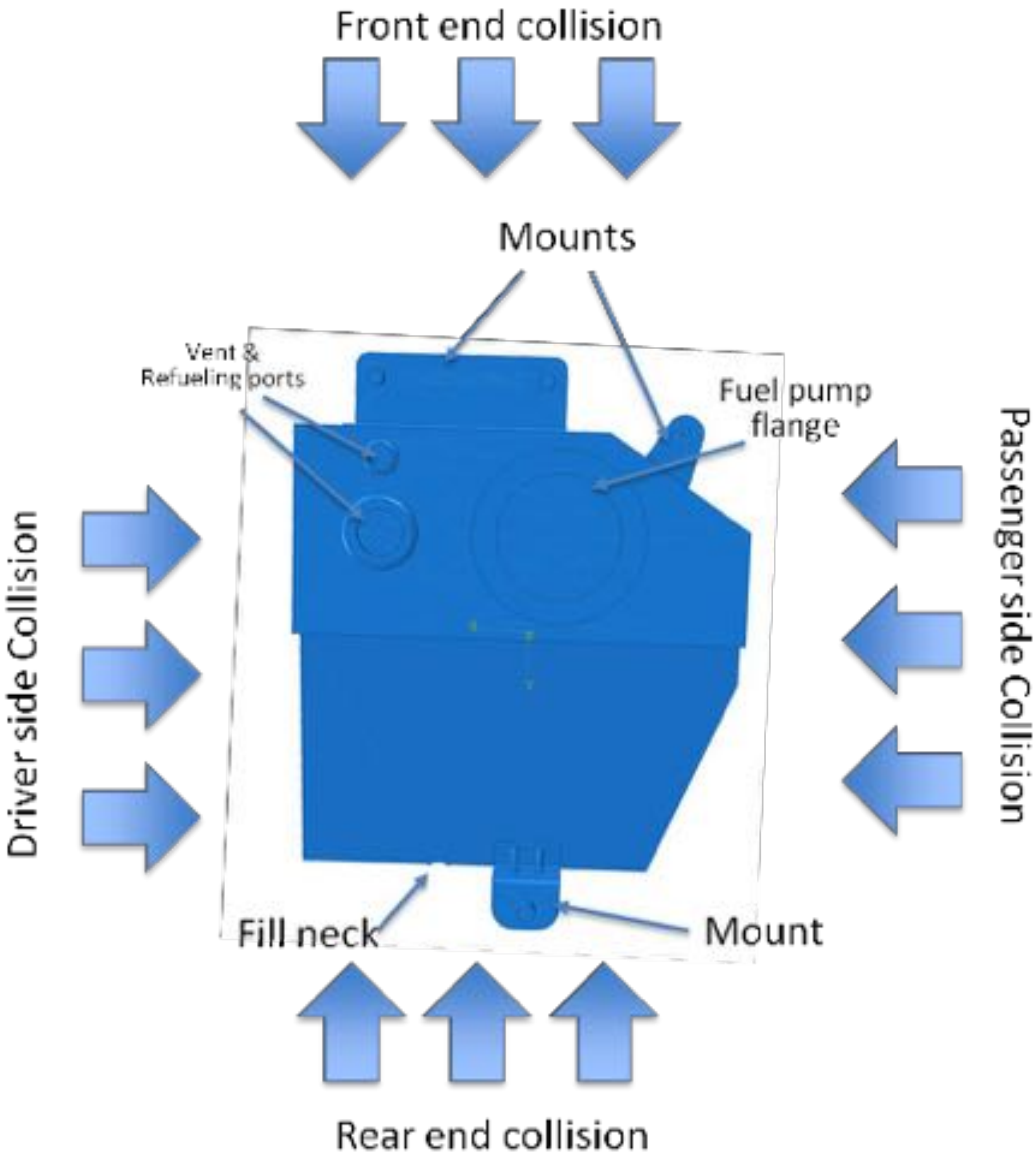


Figure 2: Fuel tank geometry with the direction of acceleration due to different collisions.

Section 2: Development and Description of the CAD Geometry

The tank was designed using Siemens NX, and then exported as an .IGS file which could be imported into the FEA software, Abaqus. The primary analysis of the simulation was focused on the stresses in the mounts and not the tank. The material properties of the 5052 aluminum used on the tank are from the Aero Space Metals website which can be seen in Table 1.

Table 1 Material Properties of 5052-H32 Aluminuum

Property	Value
Modulus of Elasticity	70.3 GPa
Density	2680 kg/m^3
Poisson's Ratio	0.33
Yield Strength	193 MPa
Ultimate Tensile Strength	228 MPa

Additionally, both plasticity and damage have been inputed into the material properties of the model. For the plasticity, 193 MPa is used as the onset of plastic strain and 228 MPa is used as the stress when plastic strain is 0.3. As for the damage, 0.3 strain is set as the condition for the elimination of elements.

The thickness of the tank and mounts were 3 mm and the approximate size of the tank was 540 x 490 mm. By using simple geometry to estimate the volume inside the tank as well as using the properties for the E85 gasoline used in the tank, the mass of the fuel and tank could be calculated and used for the loading conditions.

Section 3: Development of Finite Element Meshes

In order to create a quality mesh, the geometry of the entire tank was examined to identify the most efficient meshes for bulk parts. The automatic meshing options of abaqus were utilized in order to create meshes for large areas instead of manual creating meshes. Approximately 130,000 elements were used to create the mesh.

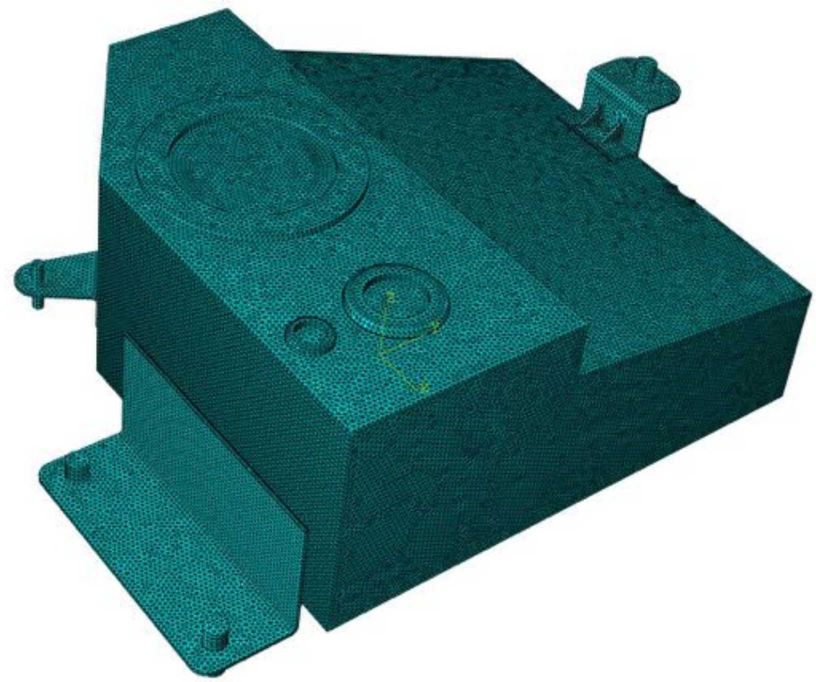


Figure 3: Fuel tank meshed with tri-type elements.

Originally, the tank was meshed using rectangular blocks, or a quad type, since the mesh best suited the geometry. This type of mesh proved to be problematic when tying the mounts to the tank because the mesh geometry didn't agree. To resolve this issue, a tri-type mesh was used with a global seed size of 4 as seen in Figure 3. The tank took the most elements to create its mesh – almost 95,000 elements.

The brackets had some more complicated geometry so the decision was made to use tetrahedral elements to accommodate a wide variety of geometries. All three brackets were originally seeded with a global size of 10 as shown in Figure 4. The largest bracket, on the right, required 26,000 elements. As you can see, the mesh fits the geometry precisely even around the bolts and bent joints.

Figure 4: Front mount meshed with tetrahedral elements.

Further refinement of the mesh brought the mounts global size to 4, the same as the tank. Figures 5 and 6 illustrate the mesh on both the tank and mount. By reducing the mesh size, the accuracy of the results was increased because the stresses were not averaged over a large area but rather a small element. This small mesh also proved to indicate high stress concentrations in only one element. The mesh appears to capture the circular geometries on the top of the tank extremely well.

An additional view of the rear mount can be seen in Figure 6. The rear mount experiences the most stress and the most plastic strain, so it is important that the mesh is fine. The mesh is able to accurately represent the two braces and the creases, which are the areas causing the most stress concentration.

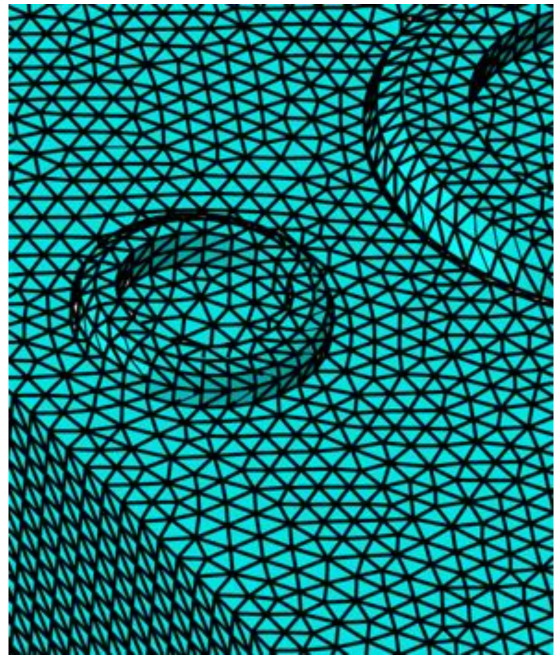


Figure 5: Close up of tank meshing.

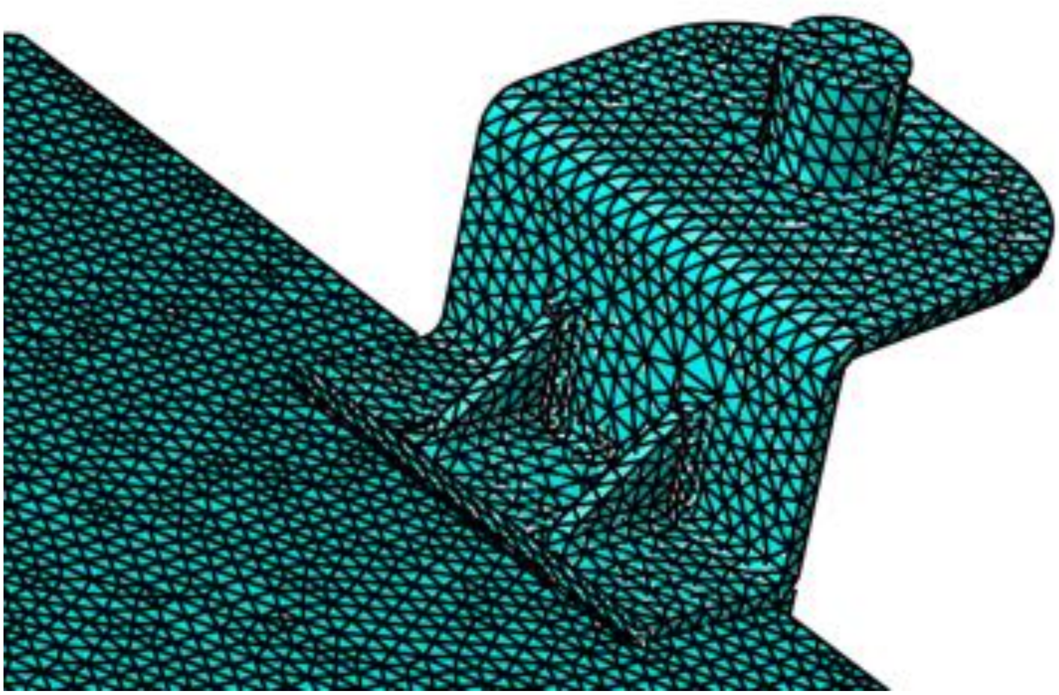


Figure 6: Rear mount meshed attached to the meshed tank.

Section 4: Development and Description of the Model Assembly and Boundary Conditions

As illustrated in Figures 7-10, the conditions to be simulated require the three supports to be held rigidly and various forces to simulate both the body forces and the forces due to the rapid changes in acceleration during a collision. Several different techniques will be utilized in order to adequately simulate the previously stated conditions.

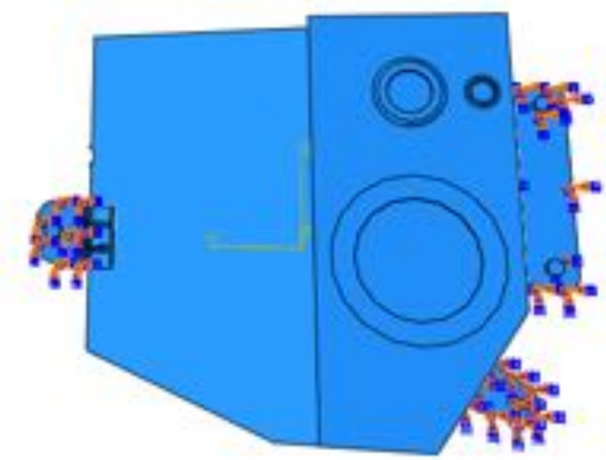


Figure 7: Boundary conditions for the tank mounts.

The boundary conditions applied to the three supports are supposed to imitate the rigid connection directly to the frame of the vehicle. Therefore, the boundary condition is a mechanical displacement/rotation with a zero displacement for all directions as illustrated in Figure 7. Having the mounts rigidly connected will allow the analysis of stresses throughout the tank and the mount structures but not the bolts or the tops of the mounts.

The weight of the tank must be considered as an additional load so a gravitational acceleration of $9.81\ \frac{m}{s^2}$ was applied to the bulk of the tank in the negative z direction. As illustrated in Figure 8, the yellow arrows indicate the direction of the gravitational acceleration. Abaqus will automatically create the force asscociated with the mass of the tank multiplied by the indicated acceleration. This gravitational load will be applied using a ramp input where the initial vaue is 1 and the final value is 1 to ensure the load is constant throughout the simulation.

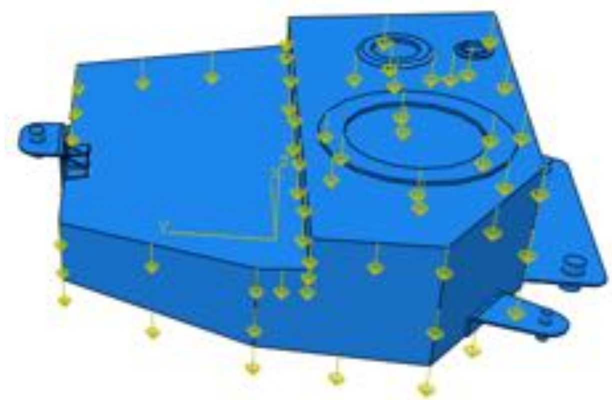


Figure 8: Acceleration due to gravity on the mass of the tank.

The collisions will be simulated using the gravitational load command just like weight simulation above. However, the “gravitational” acceleration is ramped so that the initial acceleration is 0 and the final acceleration is 1 or maximum. The maximum acceleration that will be applied to all four sides is approximately 20 g’s or $20 * 9.81\ \frac{m}{s^2} \approx 200\ \frac{m}{s^2}$ to simulate a severe accident. As illustrated below in Figure 9, the yellow arrows point in the direction that the acceleration will occur during the simulation.

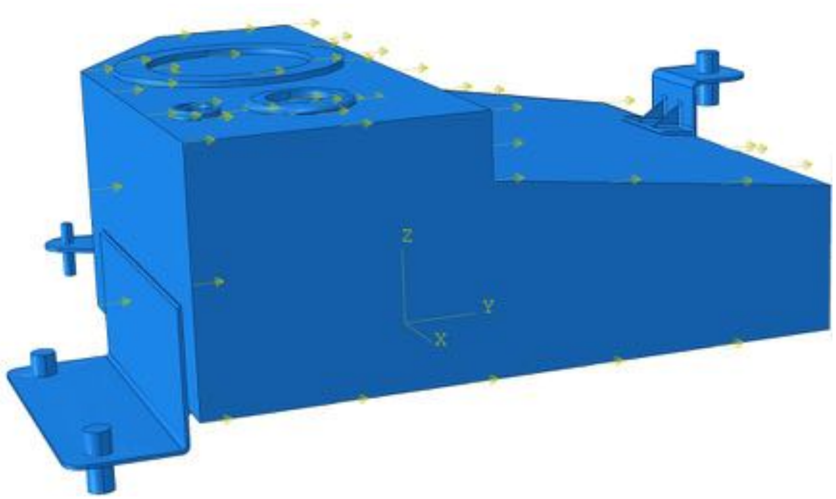


Figure 9: Acceleration due to rear-end collision.

In addition to the mass of the tank being accelerated, the fuel gets accelerated in the same direction and hits one face on the inside of the tank. As illustrated in the cut-view of the tank in Figure 10 below, a pressure is assigned to act on one face. The pressure is ramped during the step, so initially the pressure is zero and the final pressure is maximum. The pressure was calculated by the equations below with 25 kg of fuel accelerated at $200\ \frac{m}{s^2}$:

$$\text{Force} = \text{Mass} * \text{Acceleration} \qquad 25\ (\text{kg}) * 200\ \left(\frac{m}{s^2}\right) = 5{,}000\ N$$

$$\text{Pressure} = \frac{Force}{Area} \qquad \text{Area} = 390\ (\text{mm}) * 112\ (\text{mm}) = 43{,}680\ mm^2$$

$$Pressure = \frac{5{,}000\ (N)}{43{,}680\ (mm^2)} = 115{,}000\ Pa$$

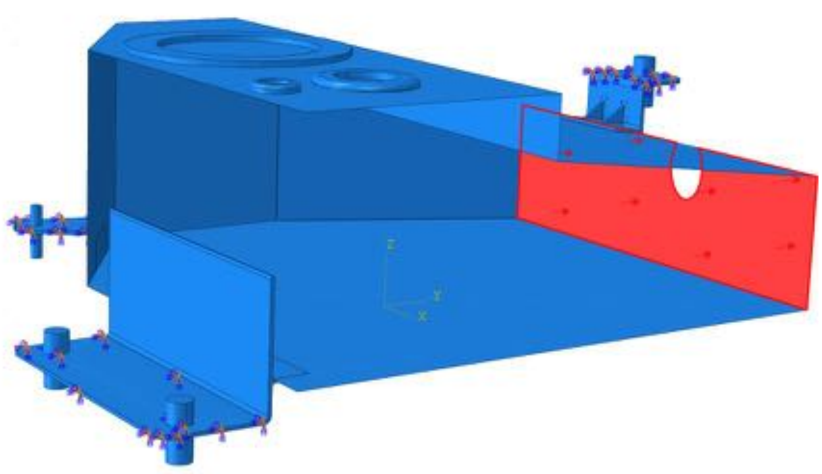


Figure 10: Pressure due to impact from fuel.

Section 5: Development and Description of Model Interactions

The fuel tank is one large shell with the supporting brackets welded to the sides. In order to simulate welded surface to surface contact, the tie command was utilized. The tie command is the best type of interaction to use if a rigid connection between parts is desired. The assumption here is that the welds are done properly and that the brackets themselves will fail prior to the welds. As illustrated in Figure 9 below, the bracket surface has been tied directly to the tank shell using the tank as the master and the bracket as the slave. The tie is a surface-to-surface type and it excludes the element shell thickness since the shell thickness was causing problems with the tie.

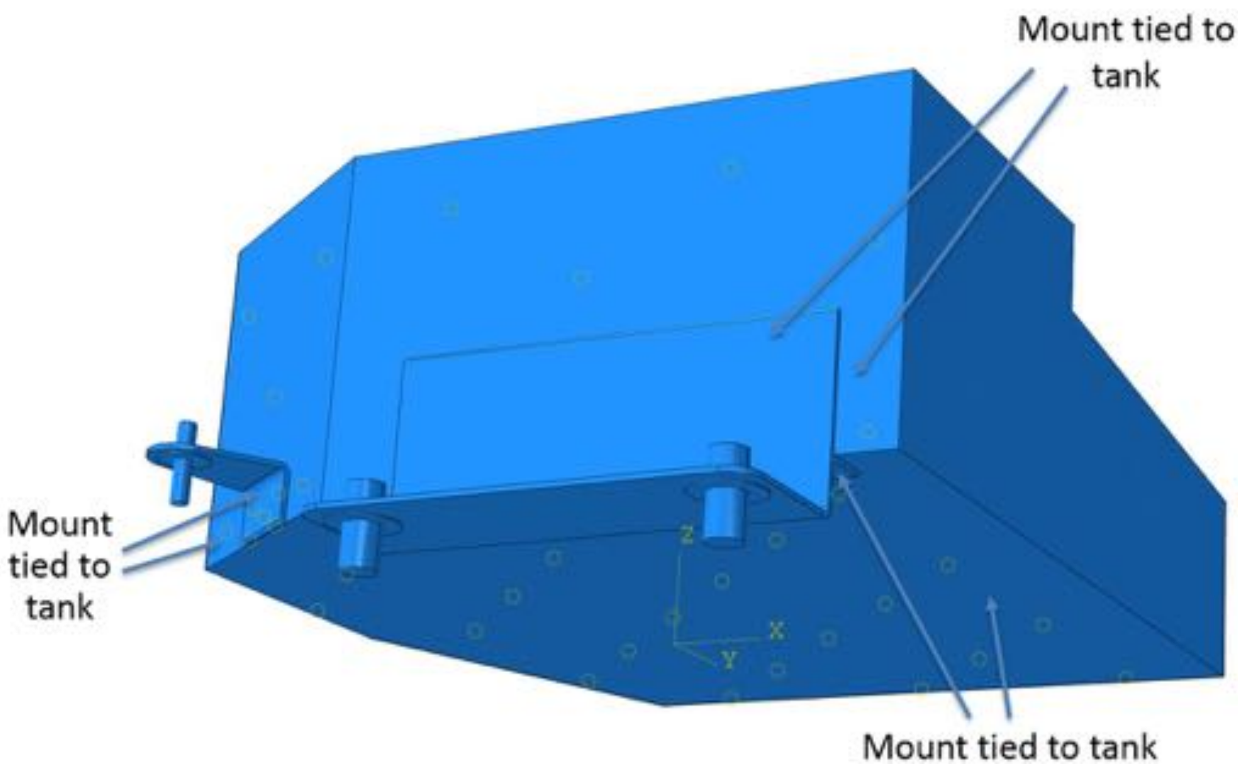


Figure 11: Tying the brackets to the tank.

In addition to the bracket interactions, element mechanical tangential friction was applied to the model in case the elements deformed and started interacting with each other. Specifically a penalty of .1 was added for the coefficient of friction and all-with-self was specified to ensure that the simulation runs even if the elements deform into self contact.

Section 6: Analysis of Finite Element Model

The model was run using Explicit Dynamic steps due to the significant motion and small time step of the process. The simulation consists of two separate Explicit Dynamic steps the first for 0.002 seconds and the second for 0.01 seconds. Using lionxg the jobs were submitted and ran for approximately two hours utilizing only one node or processor. A separate job was submitted with a gloabal mesh size of 2, which ran for twelve hours and was only half-way finished. The team decided to use a mesh size of 4 in order to reduce the computational time.

The reasoning for the first Explicit Dynamic step is due to the mount constraints and ties creating preload conditions, which were undesired. As seen in Figure 12, the stress spikes during the very beginning of the step where the forces are not even significant.

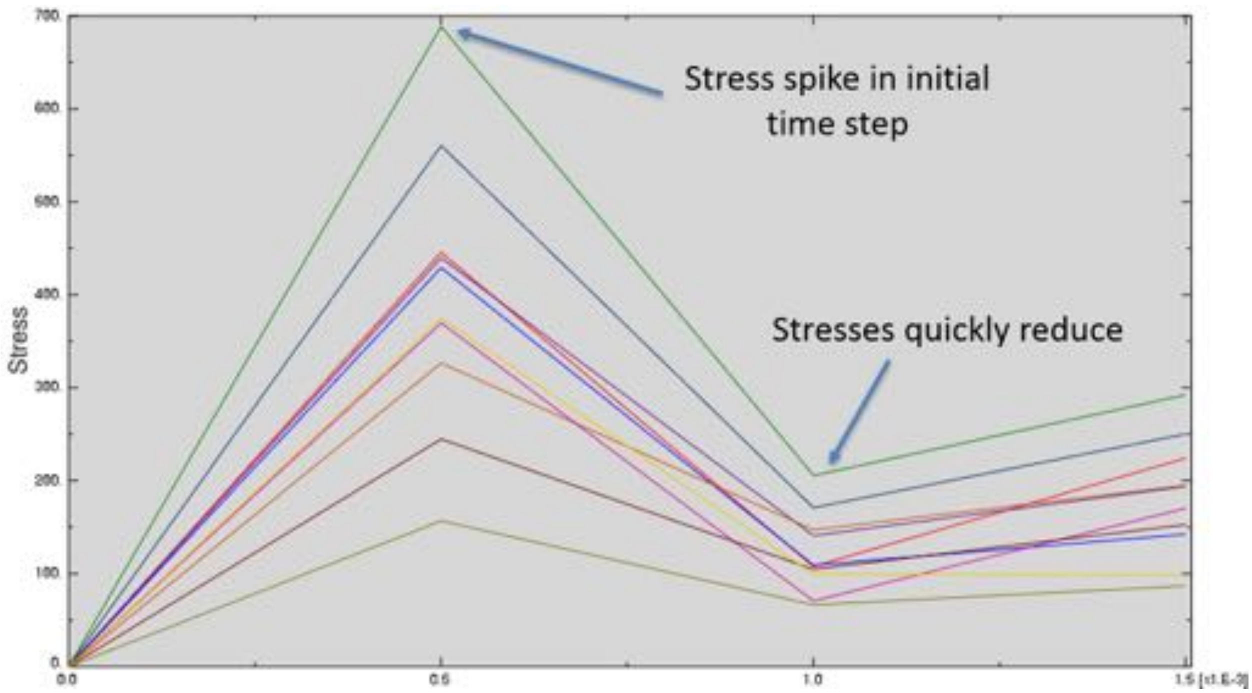


Figure 12: Spikes in stresses due to constraints.

The stress reaches 700 MPa at only 0.0005 seconds and then reduces quickly down to 250 MPa. In order to fix the problem, Dr. Kraft recommended to add another time step that goes for 0.002 seconds to neutralize the problem. After the second step was added, the stresses in the elements behaved much more noramally.

Once all of the steps were generated, several jobs were created to simulate all four collision directions. Each direction resulted in unique stresses and deformations due to the orientation of the mounts.

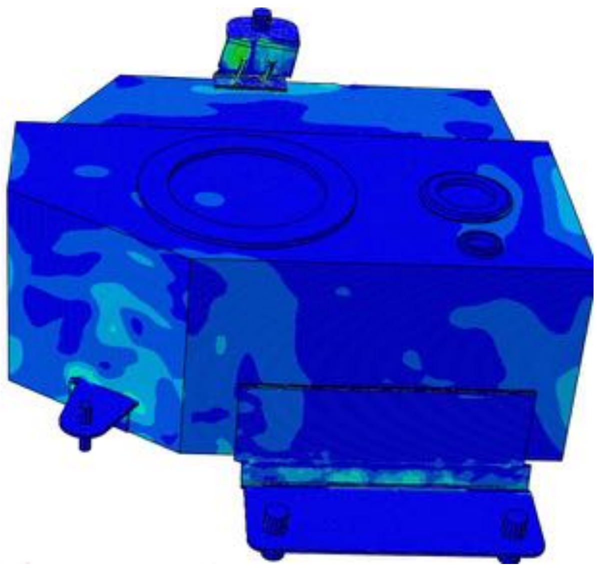


Figure 13: Passenger side collision simulation.

The simulations revealed that the tank experienced significant stresses and deformations. As illustrated in Figure 13, the passenger side collision causes a shearing stress to all the brackets. The visible deformation on the rear bracket indicates that the bracket has failed. As you can see by the stress Von Mises stress bar, the stresses experienced in the rear bracket are mostly around 400 MPa according to the colors. The yield point for the material is 193 MPa and 228 for the ultimate tensile stress. Therefore, the significant deformation in the bracket is understandable. The rest of the tank stays closer to 200 MPa, which indicates there is significantly less deformation occurring in the tank than on the brackets.

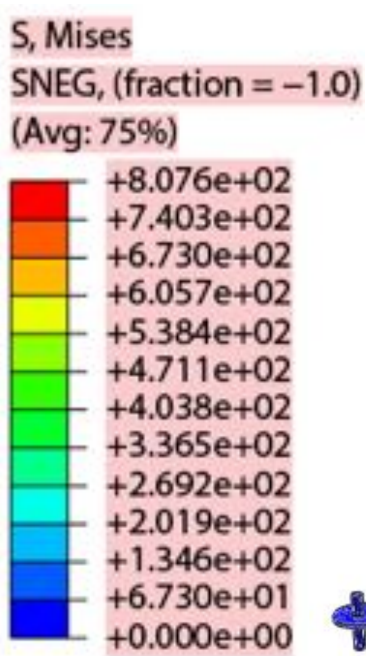


Figure 14: Von Mises stress

The stress bar, illustrated in Figure 14, shows significantly higher stresses than the majority of the elements are experiencing. The high stresses are due to a few of the elements deforming significantly, resulting in stress flares as showing in Figure 1 of Appendix B. In the rear-end collision simulation, illustrated in Figure 15, the brackets bend in the perpendicular direction. The brackets each have supports, which are designed to prevent such bending, but the brackets bend at the 90 degree angle due to the stress concentration.

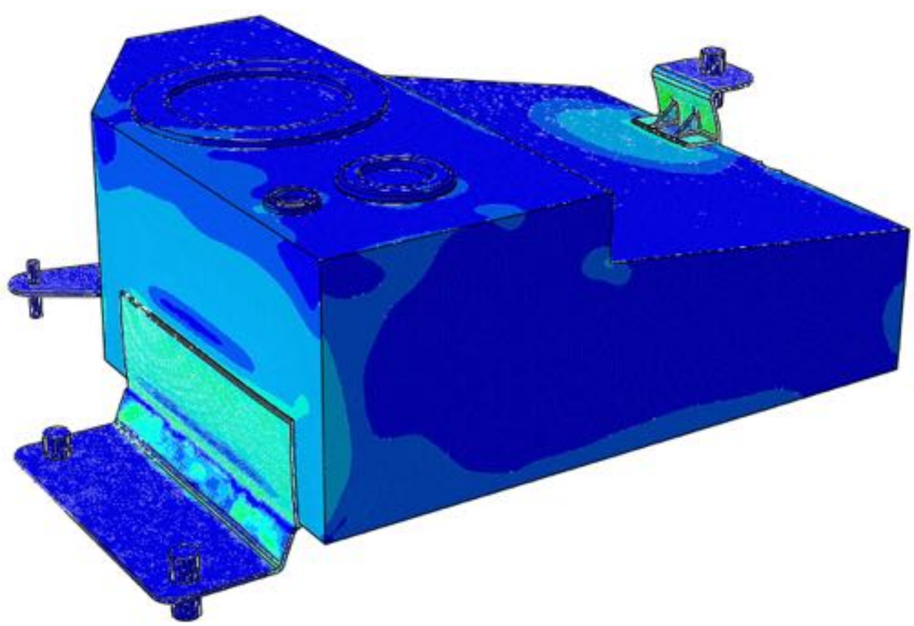


Figure 15: Rear-end collision simulation.

Section 7: Summary of Major Findings

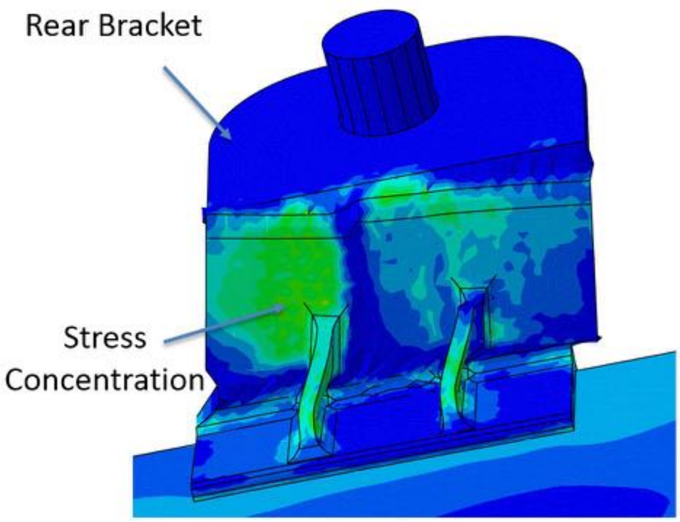


Figure 16: Rear bracket during passenger-side collision

The shearing action along with the stress concentrations are visible in the close-up view of the bracket during a passenger side collision. In Figure 16, the level of deformation is on a 1-1 scale. Clearly the bracket would not be able to withstand a 20 G-force impact from the side. The bracket needs to be longer and the thickness of all the parts of the bracket needs to be increased.

In contrast, the close-up of the bracket during a rear-end collision as illustrated in Figure 17, shows more bending than shearing. The preliminary bends in the bracket cause stress concentrations, which make it easier to bend even further. The supports are there to help prevent that type of bending. In order to reduce this type of deformation, the supports should be longer. Additional supports could also be added to reduce the load.

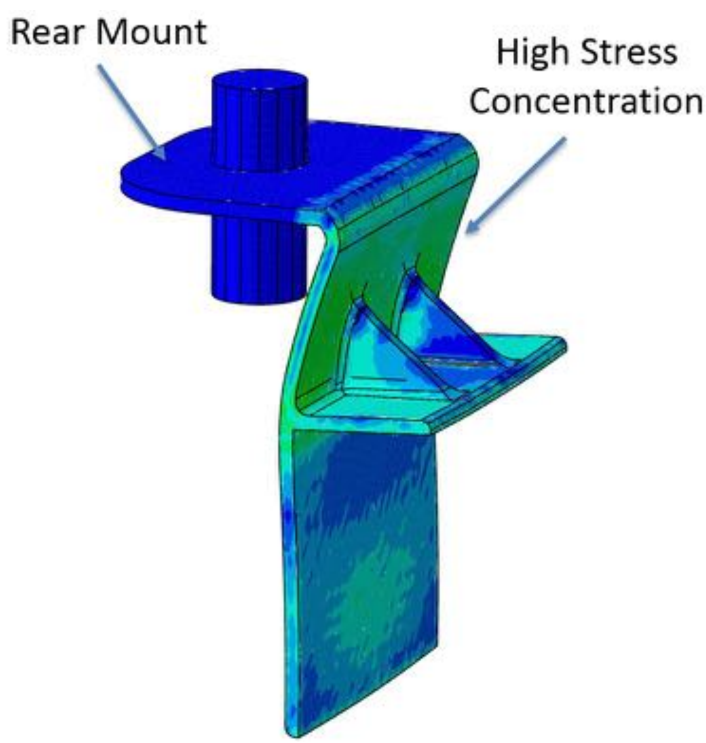


Figure 17: Rear bracket during a rear-end collision

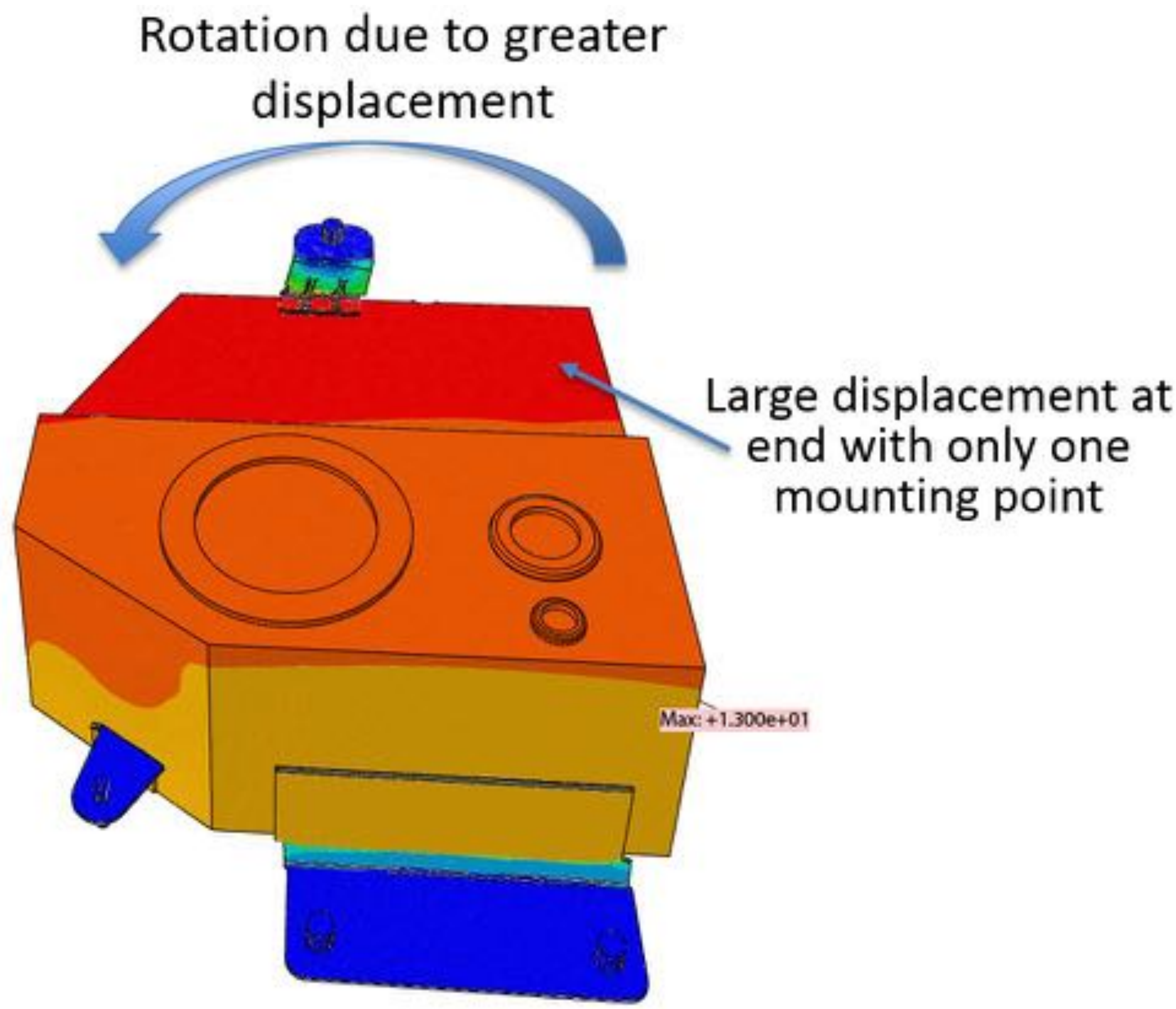


Figure 18: Visualizing the displacement of the tank during passenger-side collision

The team decided to visualize the displacement of the model of the passenger side collision as seen in Figure 18. The red color corresponds to a significant displacement while the yellow corresponds to approximately half the displacement of the red. The visualization is set to a 1-1 scale, so all visual displacement is the actual displacement of the model during the collision. As you can see, the tank moves significantly more on the rear due to the single bracket. The two front brackets pin the tank much more, which actually causes the tank to rotate slightly. The rear bracket is clearly the most deformed, which is understandable after seeing the displacement visual. Even though the rear bracket is the most deformed, the other two brackets were not able to completely restrain the tank causing them to deform as well.

Overall the brackets are not fit to sustain a collision that will cause the fuel tank to experience $200\ \frac{m}{s^2}$ of acceleration. The team recommends remodeling the brackets to make them larger and thicker to help sustain the required loading. After the bracket is remodeled, further FEA should be done to confirm that the bracket can withstand the load.

Section 8: Works Cited

1. Allain, Rhett. 2011. "The Physics of a High-Speed Crash: 70 MPH vs. 85 MPH." *wired.com.* 04 12. Accessed 2016. http://www.wired.com/2011/04/crashing-into-wall/.

2. MatWeb. LLC. n.d. "Alumninum 5052-H32." *asm.matweb.com.* Accessed 2016. http://asm.matweb.com/search/SpecificMaterial.asp?bassnum=MA5052H32.

Appendix A: Other Tests

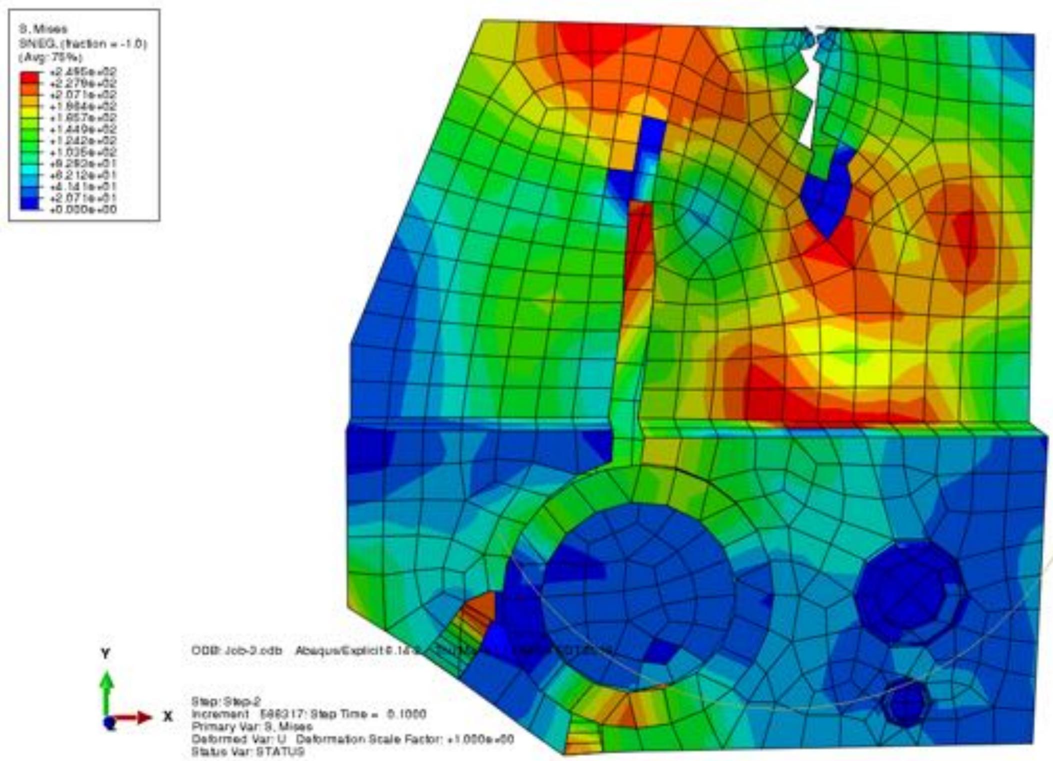


Figure A 1: Tank without brackets, damage from 50 mm displacement

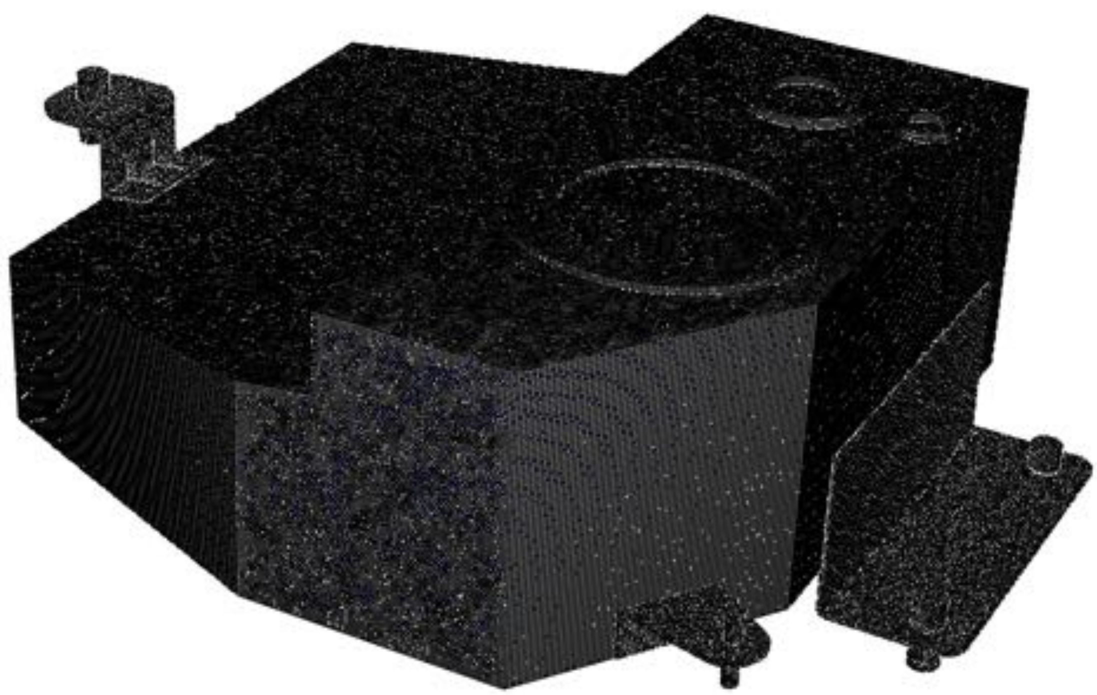


Figure A 2: Fine mesh job; 12 hours for half the step

Appendix B: Special Figures

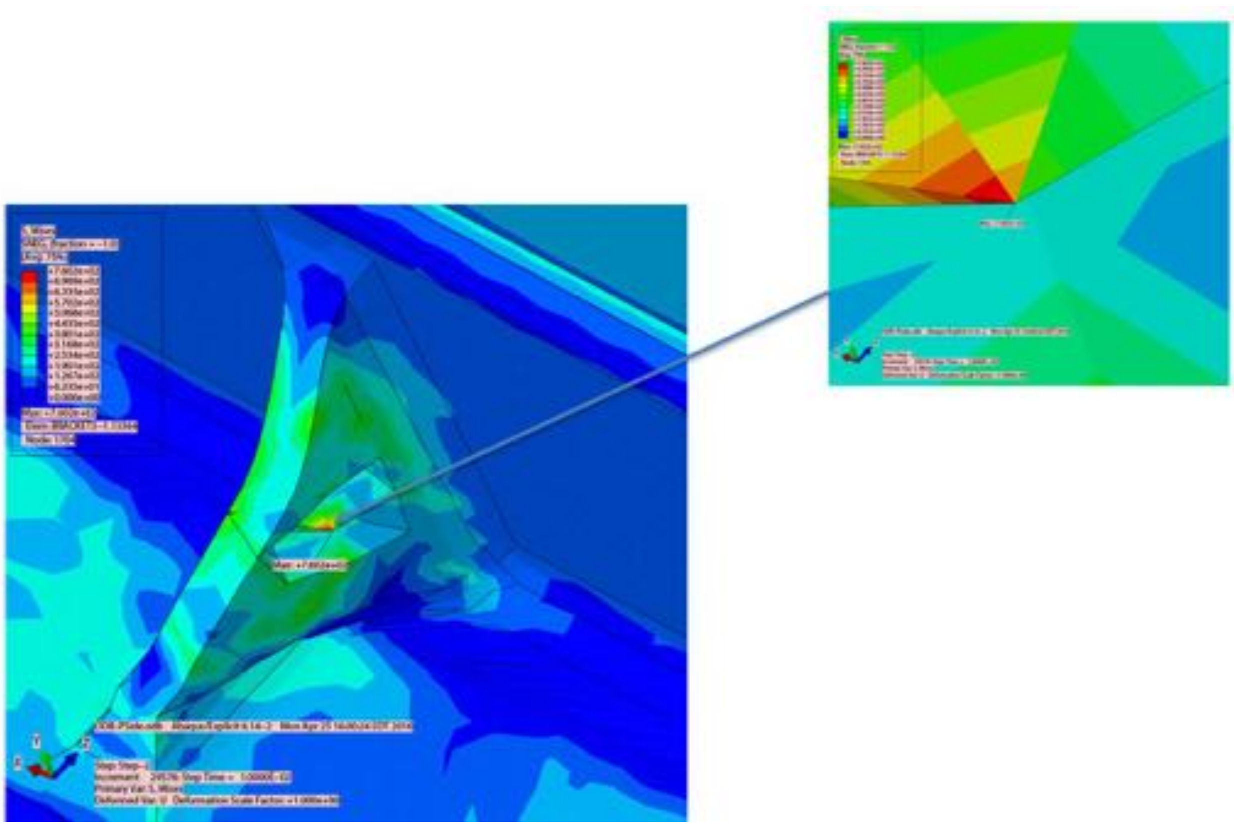


Figure B 1: High stress element within supports