Redesign Platforms for Mounting of Fire Suppression Equipment

A Major Qualifying Project

Submitted to the Faculty of

WORCESTER POLYTECHNIC INSTITUTE

In partial fulfillment of the requirements for the

Degree of Bachelor of Science by:

____________________________ ____________________________

Matthew Tatro (ME) Weitai Hu (ME)

In Collaboration with Huazhong University of Science and Technology (HUST)

HUST Partners:

Peng Zhan Xinqian Xiang

Date: May 2014

Professor Yiming (Kevin) Rong, Major Advisor, ME

Abstract

The sponsor from Caterpillar Inc. required the team to evaluate the structural stability of the access platform of the current 972H wheel loader. They were unsure of what effect would result after having fire suppression facilities installed on the platform. The team worked and achieved results in two areas- evaluating stress and deformation in key locations by conducting finite element analysis and redesigning the current platform.

Acknowledgements

The team would first like to thank our project sponsor Mr. LaForest Bradley from Caterpillar Inc. for granting us the opportunity to complete our Major Qualifying Project with a project partnered with Caterpillar Inc. His expectations motivated us throughout the project, and we appreciate his kind instructions and guidance.

The team would also like to thank Professor Kevin Yiming Rong, who gave the team tremendous encouragement and motivation. The project would not come to fruition without his care and support.

The team expresses gratitude to Mr. Lingsong He, Mr.Yanhua Sun, from Huazhong University of Science and Technology (HUST) and to Mr. Dany Dong and Mr. Bo Qi from Caterpillar's division in Suzhou. The team certainly appreciates the support in both setting up and executing the project.

Lastly, the team would like thank WPI and HUST for granting us the use of resources and facilities.

Table of Contents

Table of Figures

Table of Tables

1.0 Introduction

Heavy weight wheel loaders are popularly applied in mining, steelmaking, and construction. They have been reported to be able to withstand high temperature environment, where fire is a risk. In the operations of Caterpillar Inc., the customers reported a request; they want Caterpillar's 972H wheel loader to be able to have fire suppression facilities installed. This would be likely in the form of a large canister attachment to supply the fire suppression delivery. The company sponsor expressed doubts about the structural stability of the platform as a result of the additional load, and asked the team to perform engineering analysis of it.

The sponsor made an assumption to have one fire canister installed on the access platform and asked the team to investigate the stress and deformation of the platform under such canister and human load. Due to the safety requirements and design, Caterpillar Inc. found this to be the most suitable location for the canister. The sponsor also expected a redesign of the platform that would withstand such assumed loadings. These designs would be reasonable and within manufacturing requirements.

The team understood that these two goals, if accomplished, would provide advices for Caterpillar on whether or not there is a need to change its current platform and if so, what could be done to strengthen the design for those customers who requests the specially modified platform that can withstand the additional load of the fire suppression canister load..

2.0 Background

Preliminary information before the project's main start date was researched individually by each team member. After gathering the required information, it was then formulated into a presentation with the partner from the same university. This was then presented to the advisors to gain feedback about the plan and ensure a solid understanding of the project requirements and process. The following is a collection of the background information researched for the project.

An additional component of the background research was to prepare a series of questions to ask the sponsor, to further help the understanding of the project's specific requirements and needs. The answers from the questions helped guide the team to develop the objectives of the project, as well as clarify any misunderstandings in the project specifications.

2.1 Company Profile

Caterpillar Inc. was founded in 1925 and its headquarters are located in Peoria, Illinois. It is considered the largest manufacturer of construction and mining equipment, diesel and natural gas engines and industrial gas turbines in the world. In the last eighty years, Caterpillar has built a global infrastructure and cooperated with agents all over the world.

As the industry leader, Caterpillar will face more and more tough problems as time goes by, and management should be the team prepared to address the problems that emerge. Agents of Caterpillar are present in over 200 countries in the world where they supply equipment, service, and financial business for their customers. In addition, 1500 branches all over the world provide rental service.

Caterpillar regards honesty and quality as guiding principles for their company. As the industry leader, they know that they shoulder the compelling obligation to rebuild the positive image of American company. Their products are design to last long, and retain greater value. While this can increase the price of some of its products, it can be more desirable for clients because the investment last longer.

2.2 Structure Briefing

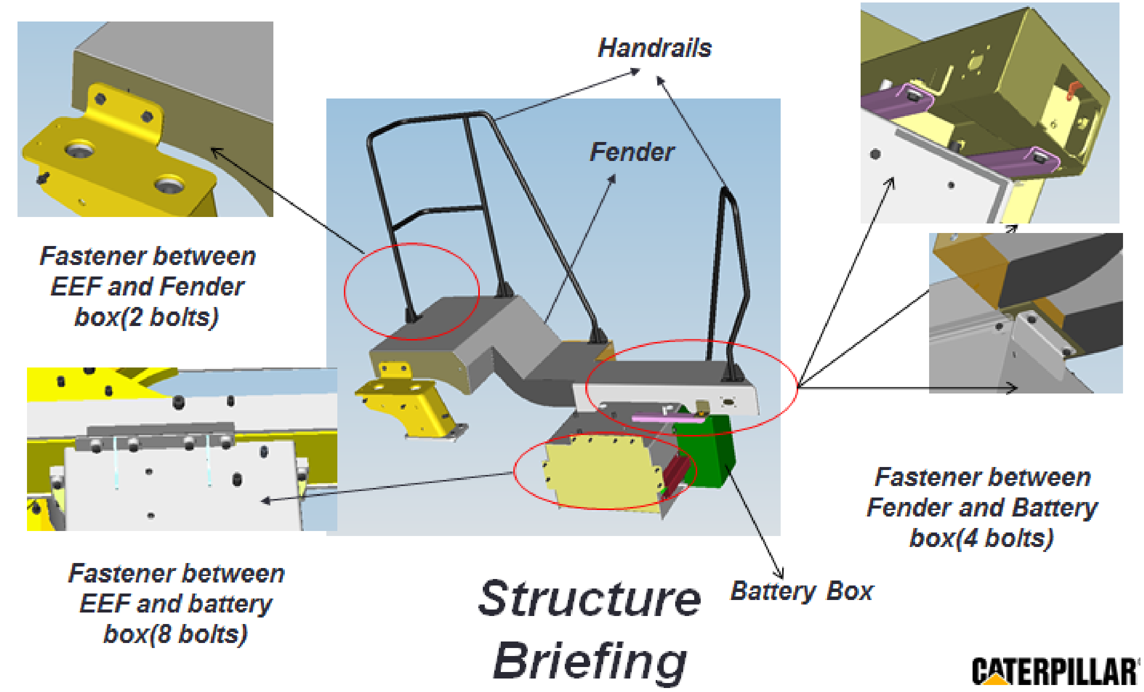


Figure 1: Structure Briefing

The image presented in Figure 1, sent from the sponsor, presents the structure of the current access platform for the 972H front wheel loader. The three red-circled areas are key locations to be investigated for stress concentration and deformation. Additionally, the group decided to investigate the stresses at all the fastener anchor points.

2.3 Regulations

Caterpillar Inc. provided data and necessary regulations for the designs. They required that all designs comply with their ISO standards. The regulation standard is: ISO 2867: 2011 Earth-moving machinery – Access systems. The following are the components and requirements of the ISO standard that apply to the design project:

1. Any surface, including steps, used for walking; crawling etc. shall be able to withstand the forces given with elastic deformation less than 10mm. [ISO 2867:2011(E), 10]
2. A foot barrier shall be provided wherever a foot could slip from the edge of a walkway or platform. [ISO 2867:2011(E), 10]

These standards will be used later during the design goals and requirements for the project.

2.4 Questions and Answers

This section contains a summary of the questions prepared for the sponsor and the answers. These questions and answers were done prior to the majority of the project to assist in developing a sound understanding of the project and its requirements.

The team had interest in knowing any industry standards and any standards that Caterpillar Inc. has for their designs. For the development of the designs, all ISO 2867:2011 requirements must be met.

The team also had interest in more experimental methods for measuring stress concentration, such as photo-elastic stress analysis, brittle coatings, and strain gauges. It was determined that these would not be needed for the project, and would be difficult to implement in the given time. Additionally, our focus should be on factors such as:

1. Supplier Tooling - The new platform must to be able to be easily fixture with the current tooling.
2. Cost – Ideally there should not be much additional cost.
3. Regulations - There may be regulations regarding where this canister can be mounted.
4. Manufacturing - The platform should be able to be lifted, manipulated, and installed easily. Also the platform should fit on the current returnable containers used.
5. Service - Common tools should be used to take the platform on and off, install or uninstall the canister, etc.
6. Design Validation - Full report showing the design.

From the questions answered, the team understood that the main focus of the project was the analysis of the fender under the new load, and that some simplifications could be made during the analysis, so long as the results complied with the standards. We were also given details about the reasons for why the canister was placed there and further why the redesign is expected to be needed.

2.5 ANSYS Equations

Basic background research into how finite element analysis is calculated, specifically that of ANSYS, was compiled to understand the workings behind the program. As well, basic definitions including those important to the post-processing results were also research to ensure the values collected were the values of interest for the project.

- Ɛy= σ_y / E
  - E is Young's Modulus

$$\sigma_e = \left[\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}{2} \right]^{1/2}$$

Figure 2: Equivalent Stress Equation

Equivalent (von Mises):

- Equivalent stress (also called *von Mises stress*) is often used in design work because it allows any arbitrary three-dimensional stress state to be represented as a single positive stress value. Equivalent stress is part of the maximum equivalent stress failure theory used to predict yielding in a ductile material. It is denoted in the above equation by σ_e.

Maximum, Middle, and Minimum Principal:

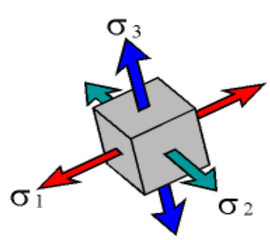


Figure 3: Maximum, Middle, and Minimum Stress

- From elasticity theory, an infinitesimal volume of material at an arbitrary point on or inside the solid body can be rotated such that only normal stresses remain and all shear stresses are zero. The three normal stresses that remain are called the principal stresses:
  - σ1 - Maximum
  - σ2 - Middle

- $\sigma 3$ - Minimum
- The principal stresses are always ordered such that $\sigma 1 > \sigma 2 > \sigma 3$.

The reported stress desired from the ANSYS calculations is the Equivalent stress, giving the magnitude of the overall stress (but not direction). Looking at the directional stress can help determine how to reduce the stresses, but the Equivalent stress determines if the design will yield or not. Furthermore, it is important to note that the results provided by ANSYS are estimations, and have errors from reducing the model to elements and nodes. The estimation calculation, if done properly, can produce results accurate enough to use as an answer.

3.0 Methodology

This chapter contains information about the objectives and methods that the team set and used.

3.1 Objectives

The main objectives of the project from the PQP portion of the project defined as the following:

1. To evaluate structural integrity of 972H Platform Assembly by performing a combination of Canister Load and Human Load.
    a. To find the displacement and stress induced on the platform assembly under the following conditions:
        i. With Human Load at 1-G acceleration.
        ii. With Canister Load at 1-G and 3-G acceleration
        iii. With Canister and Human Load at 1-G acceleration.
    b. Determine the situation with greatest deformation, stress, and strain to use as a benchmark test for redesign. The same load would be applied later to determine if the design is sufficient.
2. To generate and evaluate designs that solve structural issues found in the previous evaluation based on the following design goals:
    a. Ability to withstand load requirement (safety in elastic deformation) using element analysis.
        i. Select best design out of a multiple iterations when applicable.
    b. Discuss and evaluate safety, convenience for supplier tooling and parts, low production cost, and manufacturability.

In order to achieve the objectives within a limit of 7 the weeks, the team decided to utilize two methods simultaneously: Finite Element Analysis and Standard Engineering Design Procedure.

The two methods complement each other where the first method would evaluate the feasibility of the current platform design and the second method would provide results that could become alternatives of the current design. Designs would be made where the product fails to meet expectations and exceeds the design specifications. These new designs would be then placed back into the element analysis to evaluate the integrity of the new designs and ensure they meet all requirements.

To execute the project efficiently, each task was comprised of various steps using standard procedures and prerequisites. The finite element analysis was comprised of the following steps:

- Pre-Processing
    - Define geometric domain
    - Define element types to be used
    - Define geometric properties of elements
    - Define material properties of elements
    - Define element connections
    - Define physical constraints
    - Define loading
    - Define mesh
- Solution Calculation
- Post-Processing
    - Sort element stresses by magnitude.
    - Check Equilibrium.
    - Plot deformed structure.
    - Produce color-coded plots.
- Accuracy and Error Discussion

Once the element analysis was completed, the design portion would be implemented where necessary. In order to completely and reliably produce a successful design, the following steps were used in the design process:

- Design Goals
  - Addresses what problem
  - Specifications
- Design Generation
- Design Analysis
  - Element Analysis
- Design Evaluation
  - Estimation of additional cost, parts, and tooling.
  - Comparison with alternative designs

3.1 Strategy and Purpose - Finite Element Analysis

The team chose to use FEA to conduct the analysis on the fender's load. Simple linear calculations were originally used, to help determine if the load was going to be close to causing larger stresses than allowed. From there, the FEA was used to determine what components were under the stress and how it translated through the system. Other methods the team thought utilizing were stress gages on the surface of the physical fender and other physical testing. This was not a viable solution because the team did not have long enough access to the physical fender, and instead had free access to the CAD models which were critical for using in FEA analysis.

The team was not familiar with any FEA programs. It was decided by recommendation and reviews that work should be conducted with ANSYS Workbench. ANSYS Workbench provides a strong user interface, to assist in the creation and implementation of the element analysis, as well as offering support for debugging the program set up and execution.

3.1.1 Pre-Processing

Define geometric domain

The geometric domain is the area for which the analysis will take place. In the case of using a computer aided simulation such as ANSYS Workbench, the domain is selected automatically upon creation/importation of the CAD model.

Define element types to be used

The element types are also selected by the ANSYS Workbench software, and are not adjusted manually. ANSYS decides how to pick the type of elements based on the geometry and input model. Additionally, the team used the static-model analysis which came with some settings pre-defined. The static-model was chosen because it matched the criteria for the objectives and conditions, and outputs were stress, strain, and deformation.

Define geometric properties of elements

The model geometry is imported in the FEA program, and checked for errors. Figure 9 shows the geometric model with a mesh generated. It's important to note that the mesh sizing varies depending on the neighboring geometry. This is because near bends, holes, and corners there are likely to be more dramatic changes in stresses and deformation in that region, and thus the program uses a finer mesh around the components. The size of the mesh was adjusted to allow for an accurate enough result, while still being solvable within a reasonable amount of time.

The model from ProE Wildfire was saved, and imported in ANSYS Workbench Model editor using the import wizard. From there, the model was checked for errors and assigned a material.

Define material properties of elements

The material properties of an assigned material are defined in a separate section. The model used the CAT Steel specifications that were provided. For some of the values that were not provided, default "construction steel" was used as the base for some characteristic values. The density was set to 7840 kg/m^3, with a Poisson's ratio of 0.3 and a Young's Modulus of 202.5 GPa. These values were used to define the resulting stresses, strain, and deformation of the fender system. After the model is imported and checked, the model was assigned the CAT Steel material.

Define element connections

The element connections traditionally would be defined with the mesh and mesh nodes. However, ANSYS Workbench defines the mesh after all other properties. Instead, the connections were associated with the geometry of the model. When the mesh is generated, the geometry connections would be converted to the correct mesh component connection. This also helps have the mesh sizing be adjusted based on the connections and constrains in the region.

For the model the team worked with fully rigid bonded connections, defining two parts connected as if they were one. In this way it simulates a perfect bond. The company and team decided that this would provide reasonable data, even with the assumption of perfect bonding. Only surfaces that would be welded were bonded together. This was done simply by creating a bond with the two surfaces selected.

Define physical constraints

Physical constrains define the boundary conditions of the system. For the model, the team used static boundary constrains for a majority of the anchor points. These anchor points is where the fender is bolted into the other components, including the frame. The team also experimented with using elastic boundary conditions, simulating the bolted components would also have some compliance in them. The constraints are defined by selecting the surfaces and then selecting the designated boundary type.

Define loading

The loading is defined to analyze the deformation, stress and strain given the properties of the fender. This is done by applying either a pressure or mass over a surface. The team chose to apply a circular mass over a $0.0113m^2$ area; roughly the size of a compact-disc (CD). This specific area was chosen because it seemed like a reasonable size for a fire suppression canister, as well as the size of a single foot pressure area. The loads were chosen to be the recommended 205kg mass (~2000N force) for the human load, as well as the 185kg mass for the canister load.

To input the load properly, a circular area was mapped on the surface of the fender to the right size mentioned above. Then, a mass value was distributed evenly across the defined surface, acting as a pressure with the load over the defined area. Adding acceleration created the proper forces. Two different accelerations were defined based on the conditions. While the machine is running, the canister would be under a maximum of 3-G of acceleration, as requested by the company. While stationary, the human and canister would be at 1-G of acceleration (normal gravity). By simply swapping one of the accelerations out for the other, the team could run the same simulation with different accelerations without having to set up each simulation separately.

The locations of the load were defined in positions that would provide maximum stress and deformation on the system. These were in the far sections from the anchor points, to provide more moment torque on the system. After many locations were defined, each was tested and compared. As it turned out, only two of the locations had areas of failure, and those areas were close together and created similar enough deformation and stress that the more extreme of the two was used for the location on all the redesigns. This ensured that all locations of failure were properly analyzed. The locations chosen and the resulting location of interest are shown in the results.

Define mesh

For the most part the mesh is generated completely by ANSYS. A few inputs were added to allow for smaller mesh sizing around bends and features, to increase accuracy at those locations. Additionally, the overall mesh size was adjusted some to provide the most accurate results, while still completing the simulation in under an hour. ANSYS Workbench provided options to allow the program to designate all other mesh properties. A check of the mesh was performed each time to ensure proper meshing of the parts, and ensuring that each component would provide adequate accuracy and precision for the simulation.

3.1.2 Solution Calculation

Once the mesh was created, and all pre-processing components were defined, the solution was run. ANSYS Workbench handled all of the calculations, and did not require any further input. One difficulty the team encountered was to know if the answers given were reasonable. A few times the team had issues were the set-up was not quite right, and the results did not seem to match the expected outcome or stay in line with the other results. This would require a check of the pre-process set-up and to run the simulation once more. Some solution calculations took hours at a time to solve due to the number of equations that go into the system for Finite Element Analysis.

It was recommended that the variables requested, such as total deformation, and equivalent stress and shear were defined and added before the simulation takes place. This allows ANSYS to known how to handle the end results while presenting it to the user.

3.1.3 Post-Processing

In a more manual application of finite element analysis, one would have to do a lot of post-processing to acquire any coherent and organized data. ANSYS Workbench provided most of the calculations, and some of the organization. ANSYS Workbench completed the following steps in providing the presentable data:

- Sort element stresses by magnitude.
- Check Equilibrium.
- Plot deformed structure.
- Produce color-coded plots.

Once the solution was done, ANSYS would provide the ability to view the model in the deformed state with color coding to denote how deformed each element is. Additionally, the same could be done with the stress and strain. These charts and models were adjusted so the scale would be uniformed or adjusted to make the variation within the system visible. Some colors like red were denoted to areas where the material was found to yield due to the loads, allowing for the team to quickly find areas of concern and failure to allow for the team to move on to the redesign portion of the project.

3.2 Strategy and Purpose - Standard Engineering Design Procedure

For creating and analyzing the team's designs for improvements to the fender component, the team decided to use a relatively standard design flow, beginning with defining the requirements and objectives, creating a variety of designs to achieve the goals, ensuring each design meets the goals, and then evaluating the designs to find the best alternatives out of the remaining designs. As will almost any engineering design project, there are an infinite number of solutions to a problem, but as a team the objective was to find the best solutions possible. Each step is repeatable and could kindle the inspiration for another step. It is crucial to have all members involved in this procedure to maximize creativity and variety in the thinking. This procedure serves as rules and directions for the team.

3.2.1 Design Goals

There are three components to the design goals. The first were those provided by the company at the start of the project. These include:

- Deformation must be smaller than 10mm maximum.
- There can be no plastic deformation.
- The canister alone must be simulated at 3-G acceleration to simulate a running machine.
- The human and canister load are to be simulated at 1-G acceleration.
- The product must provide 20,000 hours of life or more.
- Material must be CAT grade steel.
- Must adhere to ISO 2867:2011 standards.

Furthermore, the team developed a list of requirements from research and other sources, including that of the ISO requirements:

- Any surface, including steps, used for walking, crawling etc. shall be able to withstand the forces given with elastic deformation less than 10mm. (Elastic Deformation) [ISO 2867:2011(E), 10]
- A foot barrier shall be provided wherever a foot could slip from the edge of a walkway or platform. (Safety) [ISO 2867:2011(E), 10]
- Design must be able to be fastened with the current supplier tooling. (Supplier Tooling)
- Design must not increase the current cost more than 30%.(Cost)
- Design must be able to be produced, lifted, manipulated, and installed. (Manufacturability)

Above on each design goal is also a category that is later used to evaluate the designs. That will be discussed in a later section.

The final component of the design goals is to address the specific components of the fender that were found to be failing. The team then developed requirements that the improvement designs would complete, and then were evaluated and analyzed to ensure they meet all three categories of goals. Each specific goal based on the element analysis is listed in the results section of the report.

3.2.2 Design Generation

Members of the team developed designs based on what they thought would be the most ideal solution, but also took effort in trying to find alternatives, and solutions that were not as obvious or predictable. The creativity of the designs relates to how confident the team is in their decision that the resulting design is the best option for the system. Therefore, the team developed a variety of designs, even if it seemed obvious that one was better than the other, and all were developed within reason.

3.2.3 Design Analysis

Each design was analyzed with the same method as the original fender. The same loads and constrains were applied to the new design, with adjustments as needed to match any new components from the redesign. After the revisions were simulated, the results were compared and checked that they would meet all of the goals. The designs that were successful in meeting the goals were then evaluated and compared together. For specifics on how the models were simulated, see the pre/post-processing sections of the methodology and results.

3.2.4 Design Evaluation

After the analysis, the team was left with a few possible solutions to the issues found in the original design simulation. These designs were then evaluated to compare and pick which of them would be the best alternative for the situation. Not all designs were incompatible with the other solutions. It is possible that two or more of the resulting options could be used together to complete the design.

The following were used as categories in a decision matrix to define the best alternatives:

- Elastic Deformation (score is based on how much deformation occurred due to the design).
- Safety (score is based on how safe the design is expected, and if it improves the safety of the platform).
- Supplier Tooling (score is based on the ability to use the same tooling the company already has as well as materials and parts in stock).
- Cost (score is based on estimated cost increase for the design).
- Manufacturability (score is based on manufacturing difficulty, labor requirements, and time requirements for the design. More advance features with greater installation requirements would score less with manufacturability).

Furthermore, each category was assigned a weight value that the team decided on, to influence the importance of some categories over the others. Each design was scored between one and five for each category, with higher values being more desirable. They were then totaled and compared to show the resulting evaluation score.

4.0 Results and Discussion

The following section contains results and discussion with respect to the project for each of the processed components. This includes original design processing and analysis; as well as design development, analysis, evaluation, and summary for the redesigned components.

4.1 Current Model Analysis

4.1.1 Pre-Processing

In order to assign the model material properties, the created profile for CAT Steel was added using the manufacturing steel default and adjusting values given by Caterpillar Inc. for their steel values. Figure 4 shows the created profile for the CAT Steel used in the simulation process.

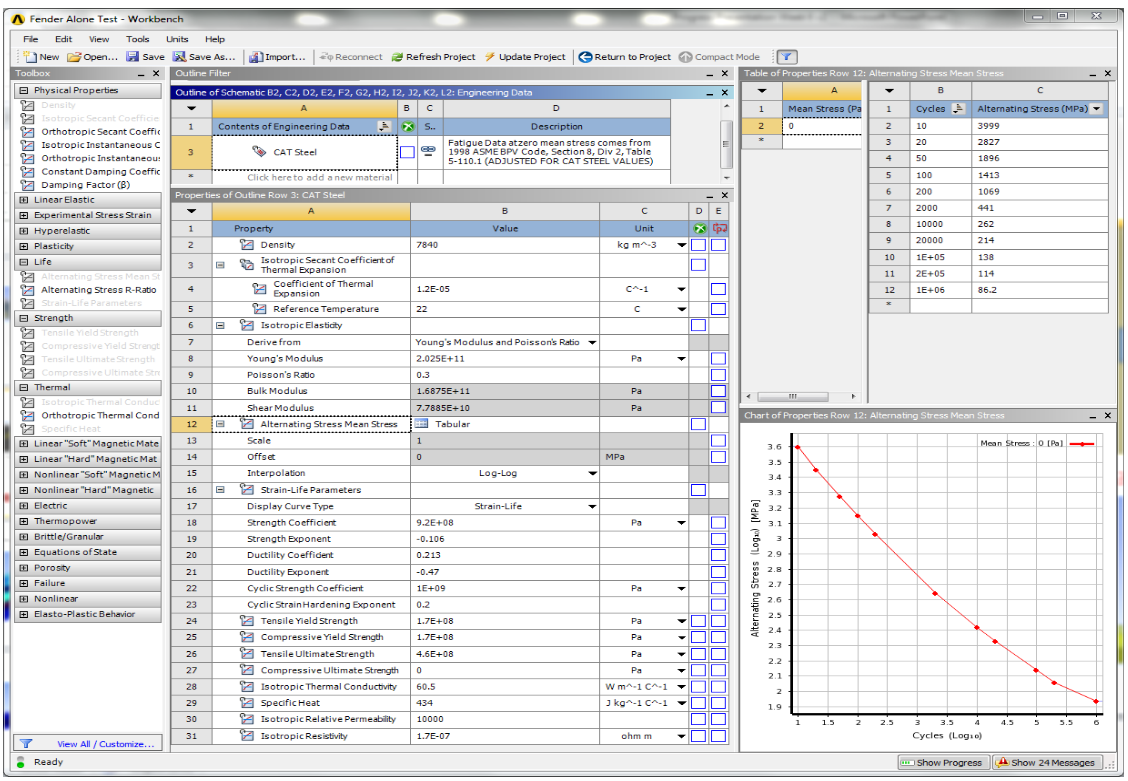


Figure 4: Material Properties for Altered CAT Steel

After the model was imported, and the material properties were assigned to the parts, the model's boundary conditions and loading was defined. Figure 5 and Figure 6 show the physical constrains applied to the system. There are a total of 5 anchor point regions on the system. For the larger anchor, the surface was rooted in place, providing enough accuracy for that section.

For the other anchors, the internal geometry of the holes was used as the fixed surfaces, to simulated contact with the bolts. Anchoring the entire surface would not provide accurate enough details. The center underside anchor points (zoomed-in), shows how the side of the material was referenced as the fixed surface. This simulated the welding contact region, and provided more accurate stress through the component than fixing the larger flat surface of the component.

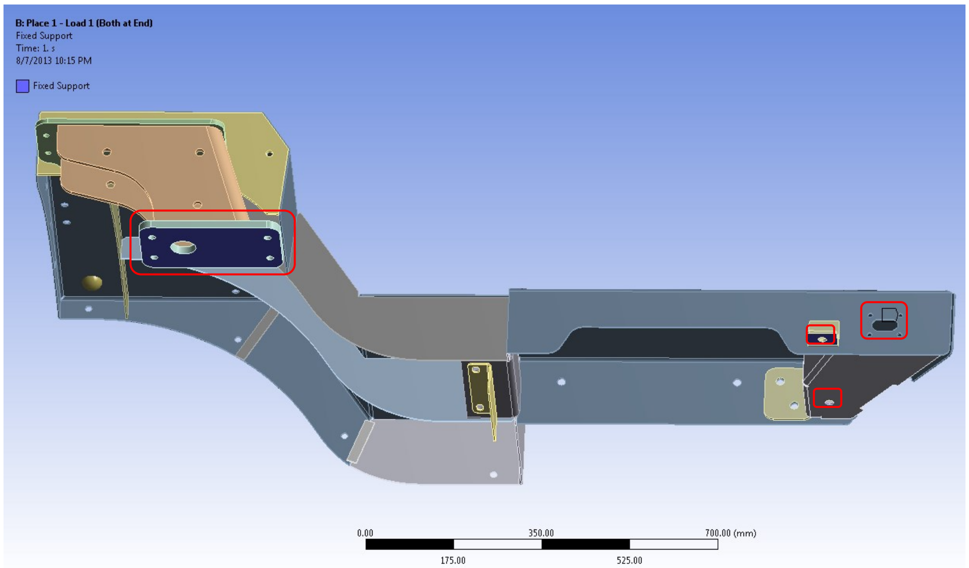


Figure 5: Fixed Support Part 1

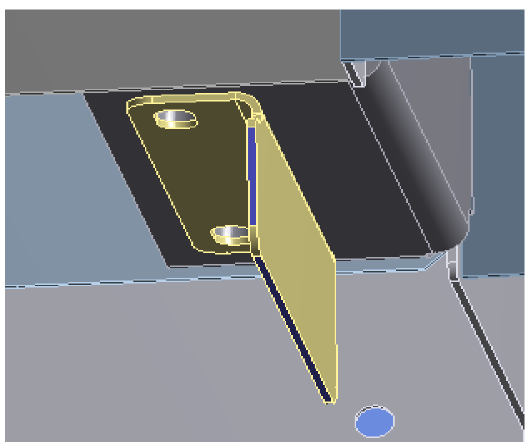


Figure 6: Fixed Support Part 2

The next step completed was the loading conditions. As shown in Figure 7, the loading was done over the circular area created, and applied a point mass (represented by the sphere) which was distributed evenly across the area.

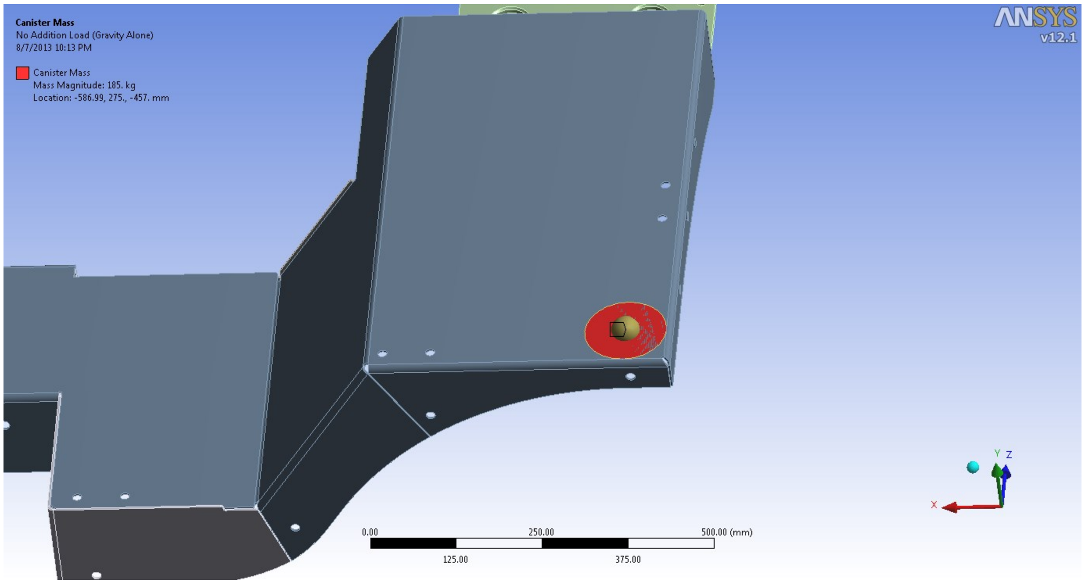


Figure 7: Example Loading

The same method was applied over eight separate locations to determine the point of greatest stress and deformation in the system, and to work from there to see what areas of focus were needed.

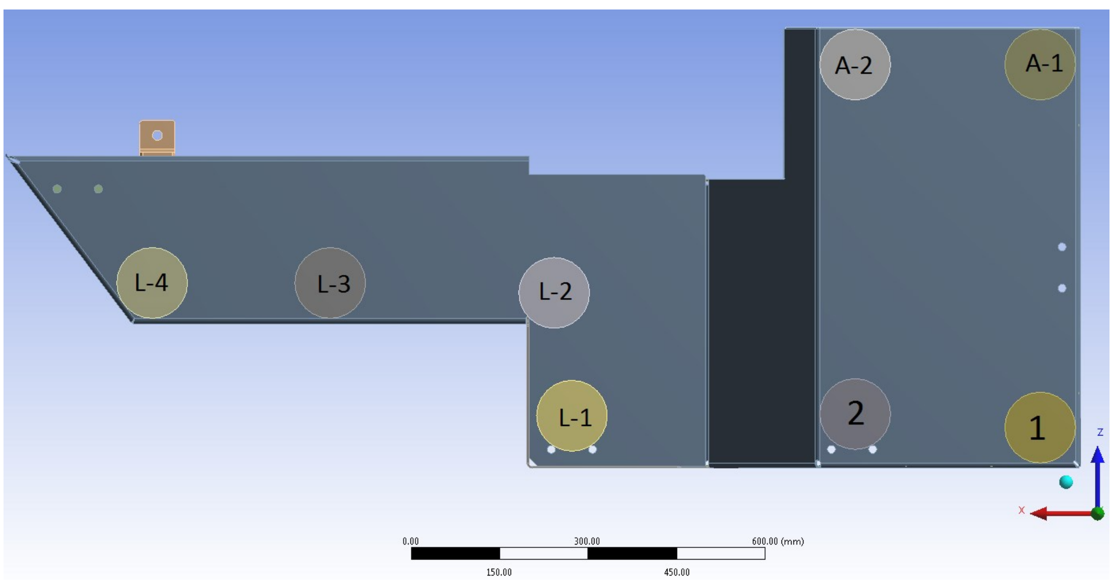


Figure 8: Test Load Locations

When the load was applied to locations L-1 though L-4, there was no failing points, and no deformation that was greater than the maximum set. None of those locations were areas of focus. A-1 and A-2 also produced the similar results, with no areas of concern. Both locations 1 and 2 created points of stress failure, but point 1 created the maximum deformation. Furthermore, the stresses produced by location 1 were greater than those of point 2 throughout the various features, leaving the group to conclude that by focusing on point 1, one could use it as the exclusive test location. If everything tested properly at location 1, then location 2 as well as the "L" and "A" locations would also be within the required specifications.

The actual loads included were human and canister at 1-G acceleration and canister alone at 3-G acceleration. Originally the team defined the system to test the canister alone at 1-G acceleration, but it was concluded to produce results less than that of the 3-G acceleration test, so it was determined to be unnecessary for the test results.

Once the model was done being constrained and loaded, the mesh was created. Figure 9 shows an example mesh used for the model. It's clear to see the variable mesh sizing as well as shapes. The created mesh provides enough detail to make a valid estimate for the stress, strain, and deformation of the system under the applied loads, while also keeping the calculations and process time to reasonable limits.

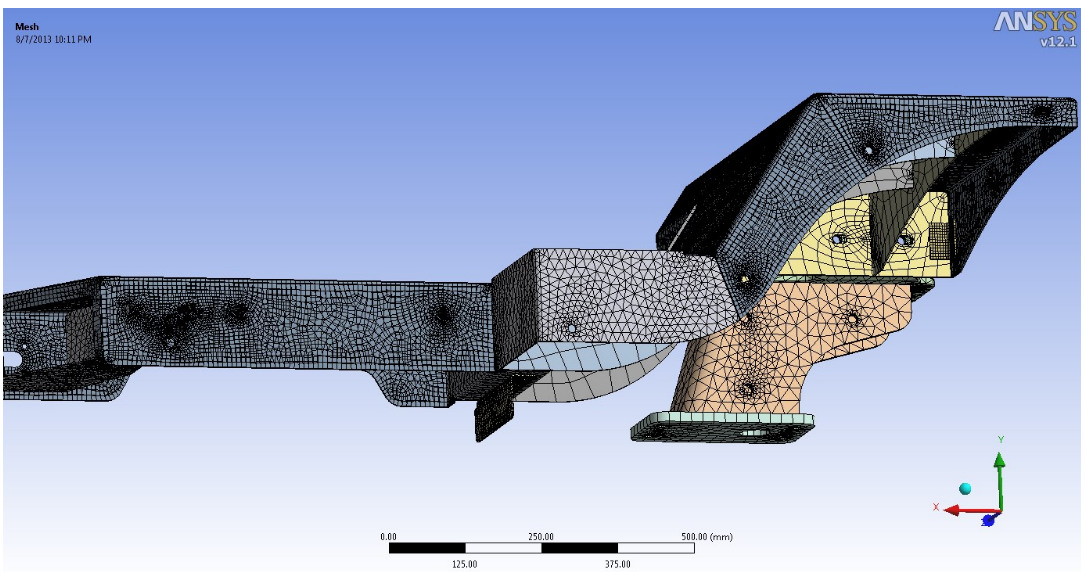


Figure 9: Mesh Model Example

4.1.2 Solution Calculation

The solution processing was completed in the ANSYS Workbench program without user input. The resulting values were exported and presented in the post-processing information. To expedite and organize the results for comparison the Workbench was organized to evaluate the designs in a defined order, and used common resources. The figure below shows how the same geometric model and material properties was used in the many different tests.

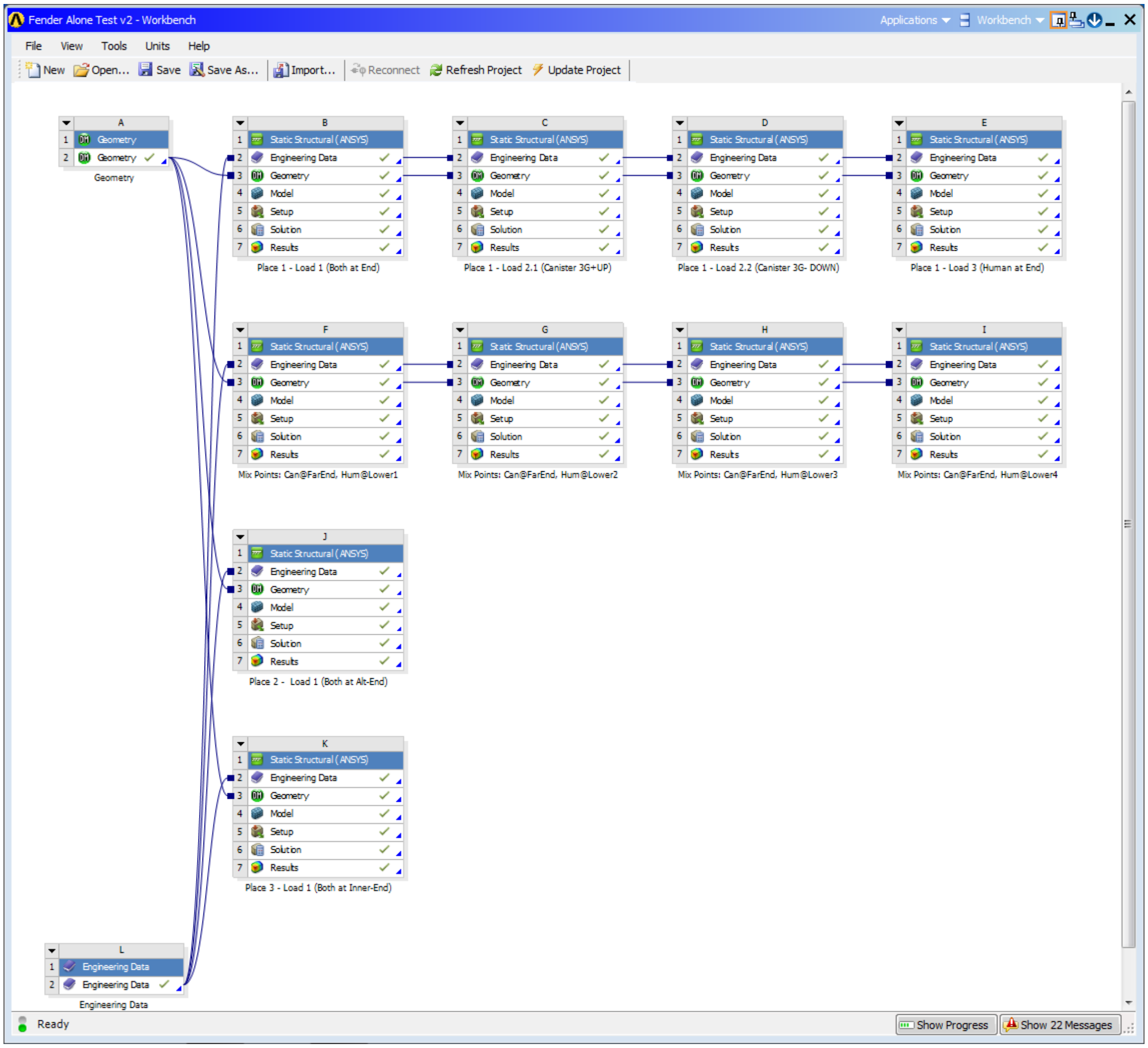


Figure 10: ANSYS Workbench Layout for Current Design Testing

4.1.3 Post-Processing

The first set of test conducted were to decide which produced a greater stress and deformation result on the system; the canister at 3-G or the human and canister at 1-G. Theoretically, the canister at 3-G should produce greater values because the total force applied on the platform (185kg*3-G) is greater than the force of the 1-G test ([185kg+200kg]*1-G). This expected result was found to be consistent with the results of the test.

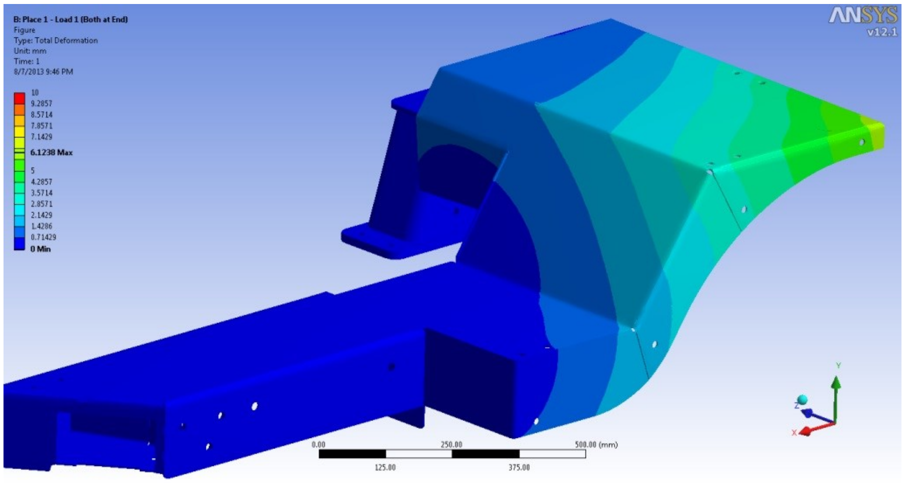


Figure 11: Load 1: Canister and Human at 1G. Total Deformation.

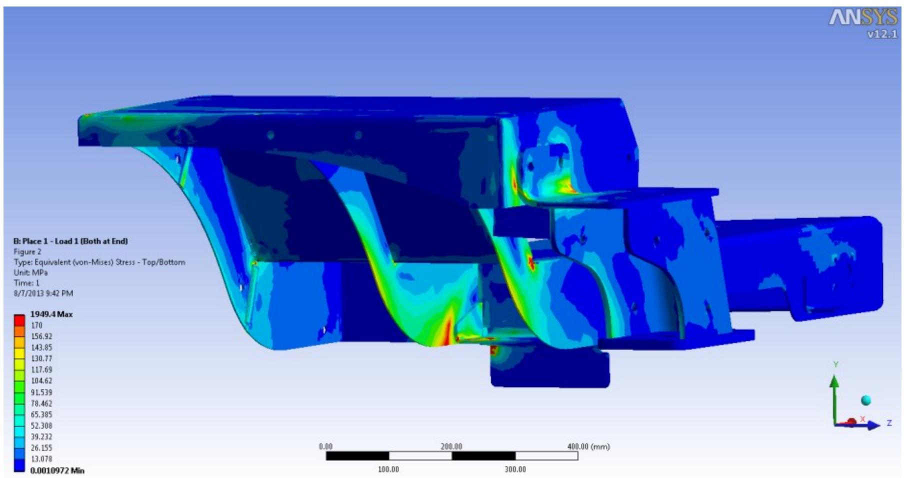


Figure 12: Load 1: Canister and Human at 1G. Equivalent Stress.

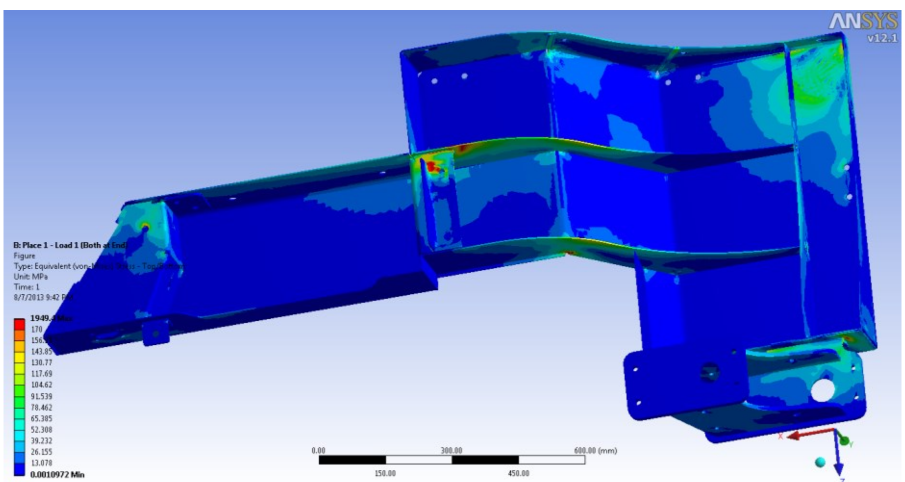


Figure 13: Load 1: Canister and Human at 1G. Equivalent Stress. Alternative View

Figure 11 though Figure 13 shows the stress and deformation of the 1-G simulation with the canister and human load together. The maximum deformation is less than 7mm, within the range required, but the stress does reach points that would cause the material to yield.

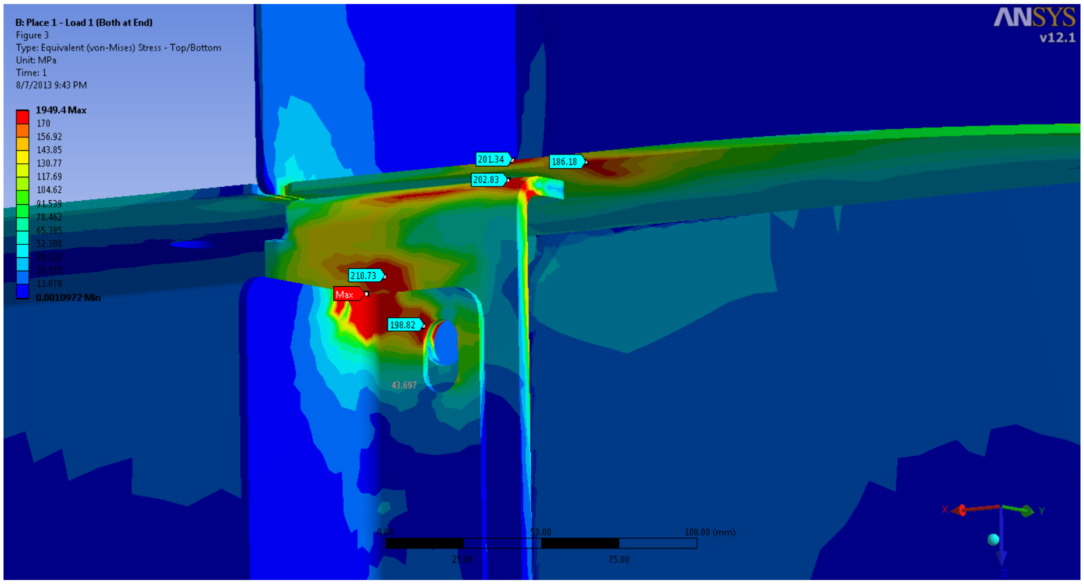


Figure 14: Load 1: Canister and Human at 1G. Equivalent Stress. Area of Interest 1.

Above Figure 14 shows the first area of interest, on the underside of the fender system. The team determined this as an area that needs revisions, as it's shown the load on the support rib transferred through the anchor is too great for the current design. Note, while the scale says there is somewhere a maximum of almost 2000 MPa of stress, it is not accurate because that is a calculation error caused by the mesh. To combat this the team probed all areas of yielding to determine if the yield is substantiated with a gradient of stress, or not in the case of the maximum, where it was on a corner mesh without a proper transition to a lower stress area.

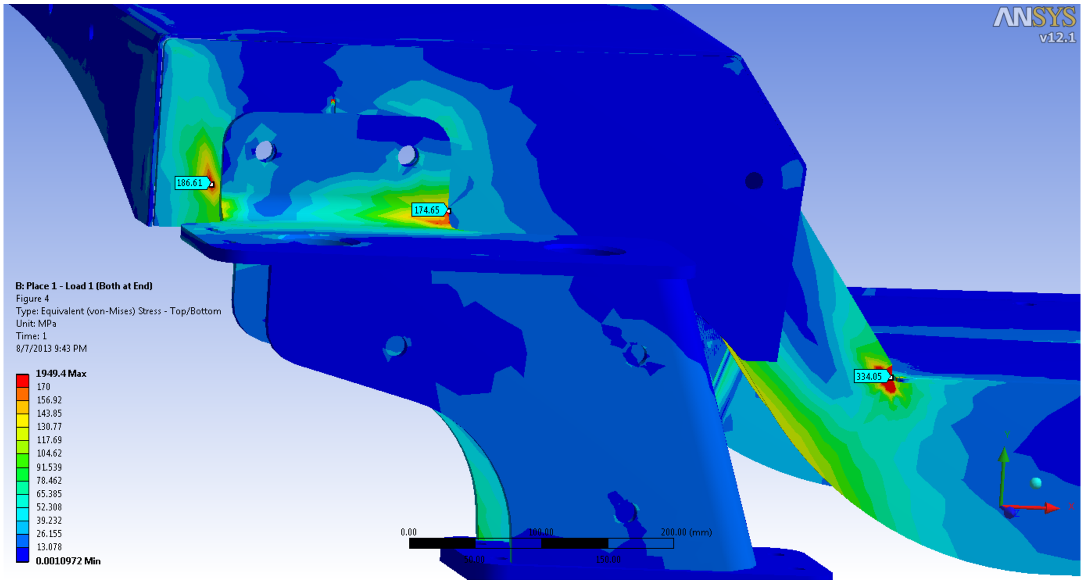


Figure 15: Load 1: Canister and Human at 1G. Equivalent Stress. Area of Interest 2.

The second area of interest was on the main anchor point for the upper section of the platform. The values are close to the yielding point of 170MPa, and are kept as an area of interest for later. The corner stress on the right side of the figure is likely due to the mesh, as shown it does not properly fade and translate like the other sections do. Furthermore, welding would be completed on the section that the model does not include, and that will help direct the forces better to the lower section without creating the yielding point. It was determined that that location was not a point of interest. Another point of yielding shown in Figure 16 does not include the welded material that would be added, and was advised to not be an area of concern for the project.

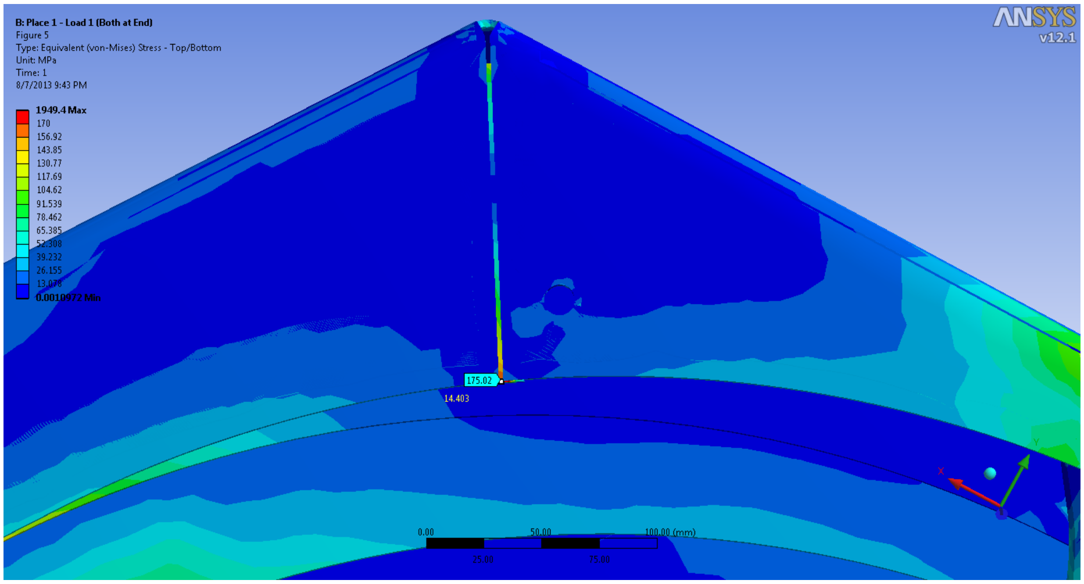


Figure 16: Load 1: Canister and Human at 1G. Equivalent Stress. Yielding Due to Mesh

The next result set was created from the 3-G simulation with the canister alone. The whole system undergoes more stress and strain, as well as greater deformation. Any further points of interest were extracted from the results of the 3-G testing.

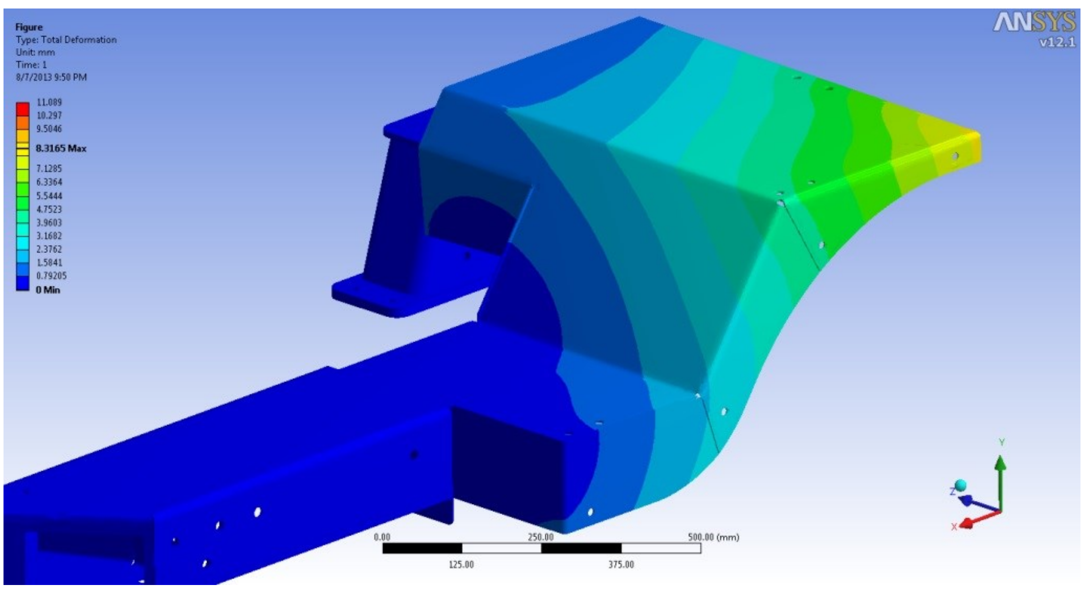


Figure 17: Load 2: Canister at 3G. Total Deformation.

Figure 17 Shows how the deformation reach values just above 8mm, greater than that of the 1-G simulation. This is the expected outcome, and provides confidence in the results. Furthermore, the load produced greater yielding points on the underside of the platform, included some areas of the ribs that were not yielding before. Another section, near the corner highlighted in Figure 18 shows a new area of concern that was focused on for one of the created designs.

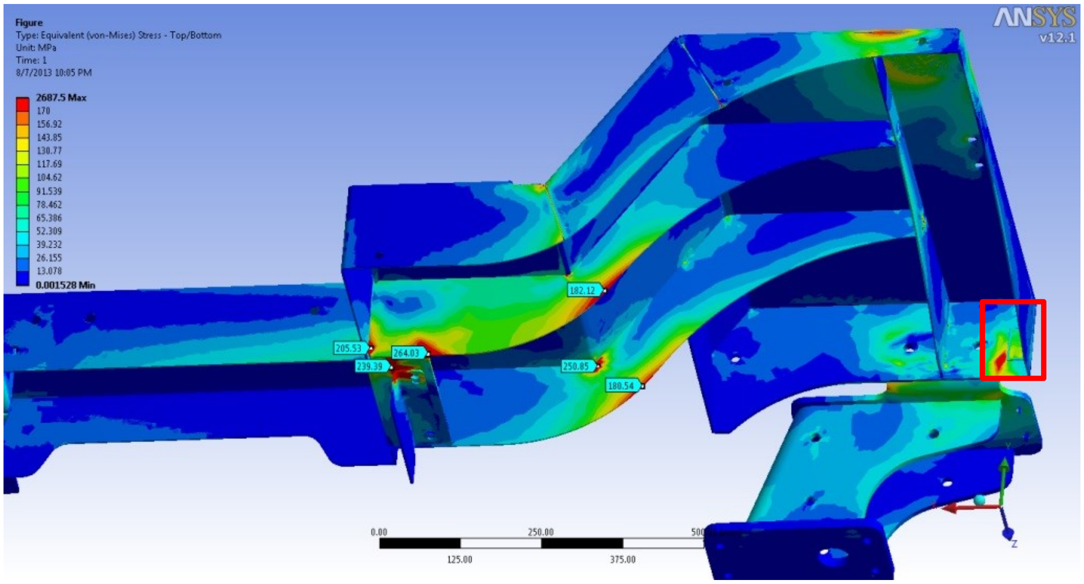


Figure 18: Load 2: Canister at 3G. Equivalent Stress.

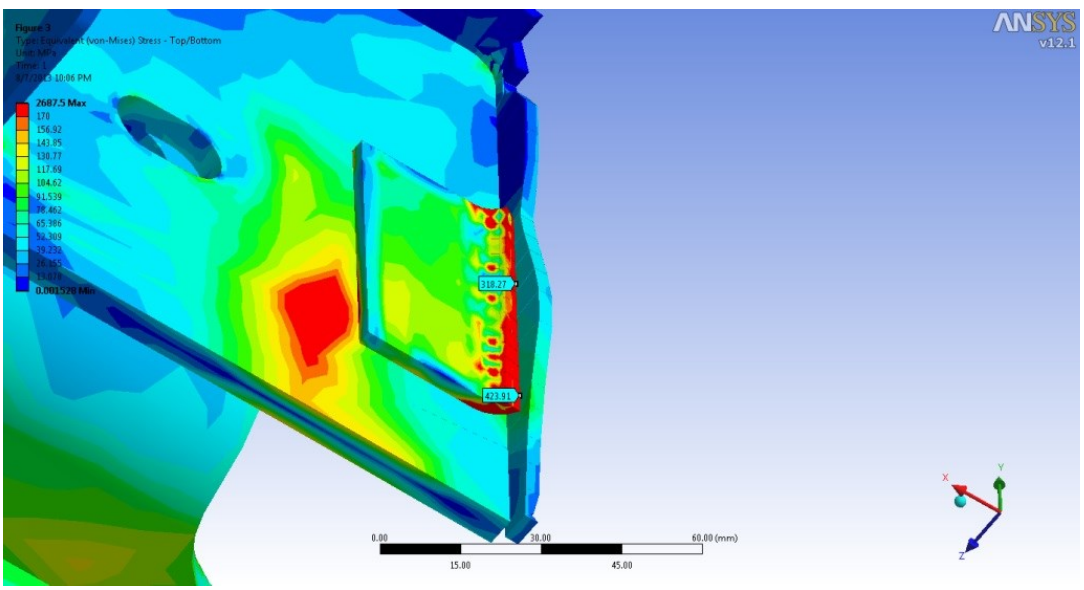


Figure 19: Load 2: Canister at 3G. Equivalent Stress. Corner Piece Section View Failure.

Taking a closer look at the highlighted section, while cutting the view to show the internal stress, reveals a component with values over twice that of the yielding point. This connection piece at the corners will likely fail due to the stress. The team used this piece as a redesign area, to improve the stability and strength of the corner attachment component.

4.1.4 Accuracy and Error Discussion

While the team had confidence in the results, there were some concerns for inaccuracy and errors. As mentioned a few locations showed inaccurate results, with stresses much higher than what reasonable values for the location would be. Further analysis and research showed this is likely due to the way the model is connected.

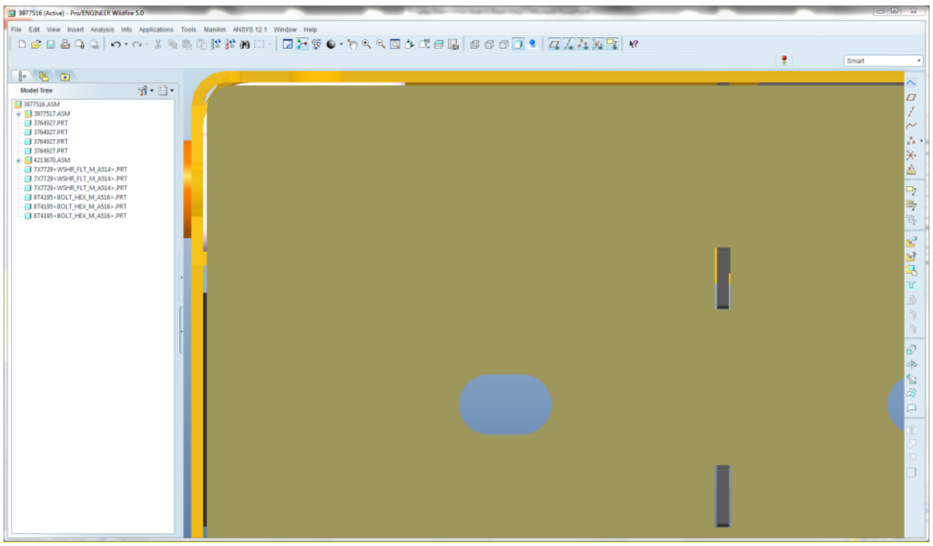


Figure 20: Accuracy and Error Example

Figure 20 shows the back-plate of the platform, and how it was modeled with respect to the main sheet metal platform. As shown, there is a clear gap between the two bodies. This was checked with measurement tools in the modeling software, and the gap was intentionally put in in the design. When connecting the model, the gap is treated differently than that of the material, and remains in the ANSYS Workbench model. It still relates the two surfaces together, bonding them, but the gap acts as a bonded, but not rigid, space between the two surfaces. For that reason the team believes there to be some inaccuracies near these bonds, because of the space having compliance as well. While this means the results are not as accurate as a fully connected model, they still provide adequately accurate results and give an idea to the locations that need revision, and whether or not the adjusted designs meet all the required expectations or not.

4.1.5 Summarized FEA Results

Given the simulated results, the team developed a list of locations and features that need redesigning due to the fact that they yield under the given load. These locations and features were:

- Ribs on underside of platform (Figure 18).
- Under body anchor point region (Figure 14).
- Upper anchor point region (Figure 15).
- Corner bond piece connected to back plate on upper portion (Figure 19).
- Foot-guard on railing (not present, but listed on ISO requirements).

4.2 Design Results

4.2.1 Design Goals

The design goals and specifications for the created components include all of those of the previous design goals including:

- Deformation must be smaller than 10mm maximum.
- There can be no plastic deformation.
- The canister alone must be simulated at 3-G acceleration to simulate a running machine.
- The human and canister load are to be simulated at 1-G acceleration.
- The product must provide 20,000 hours of life or more.
- Material must be CAT grade steel.
- Must adhere to ISO 2867:2011 standards.
- Any surface, including steps, used for walking, crawling etc. shall be able to withstand the forces given with elastic deformation less than 10mm. (Elastic Deformation) [ISO 2867:2011(E), 10]

- A foot barrier shall be provided wherever a foot could slip from the edge of a walkway or platform. (Safety) [ISO 2867:2011(E), 10]
- Design must be able to be fastened with the current supplier tooling. (Supplier Tooling)
- Design must not increase the current cost more than 30%.(Cost)
- Design must be able to be produced, lifted, manipulated, and installed. (Manufacturability)

New goals were also developed for each part, to facilitate in the design process to ensure the design was a solution to the focused issue. The redesign of the ribs, anchor point regions, and corner piece require the following goals:

- Design does not alter assembly and component creation methods.
- Provides structural support to desired location so that they do not yield.
- Effectively distributes load and does not waste material (is lightweight).

The Foot-guard would require a different set of goals. The railing was not a point of interest for yielding, but instead its characteristic design had improvements to be made:

- Provides safety from accidently stepping off platform, as well as keeping objects on platform.
- Is not made of new part types, and is easily assembled.
- Provides access to a rear-ladder system.

The last requirement for the new railing design was to incorporate the work another team working with Caterpillar Inc. at the same time. They were designing a rear-ladder to allow the user a route of escape in case the main ladder is blocked. The redesigned railing would provide the necessary access to this ladder, something the original design lacked.

4.2.2 Design Generation

Reinforcement Plates

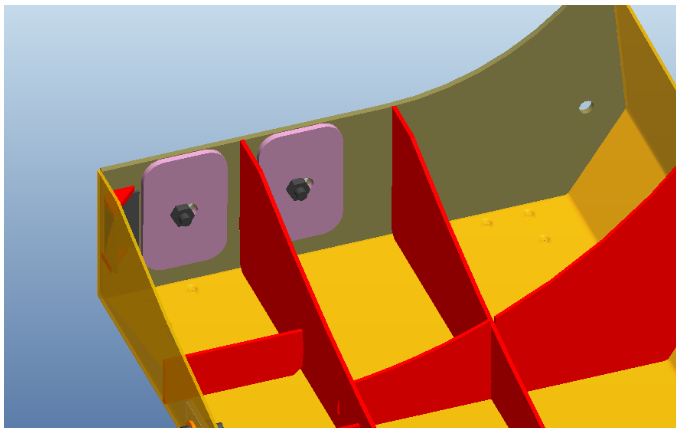


Figure 21: Reinforcement Plates 1

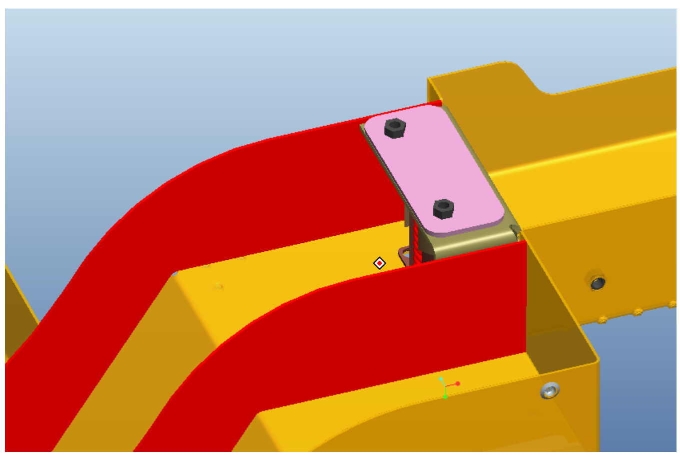


Figure 22: Reinforcement Plates 2

The first design for the platform was to add reinforcement plates to the areas that had yielding stress. These three plates would provide an overall thickness greater than the original design, better distributing the stress though it. It would be welded on the edges, and spot welded along the surface to ensure better performance. This design requires the creation of these simple plates, but does not change the original assembly. Because the reinforced platform the team developed is a special order options, the majority of customers would not be concerned with the designed plates, or the other improvements for that matter. Those that would be interested in them would likely be willing to pay a larger price for the reinforced platform design.

Corner Pieces

The team developed two iterations to the corner piece's design. The first was to replace the corner attachment part with two trapezoidal shaped inserts to distribute the load while providing more support for bending at the corner.

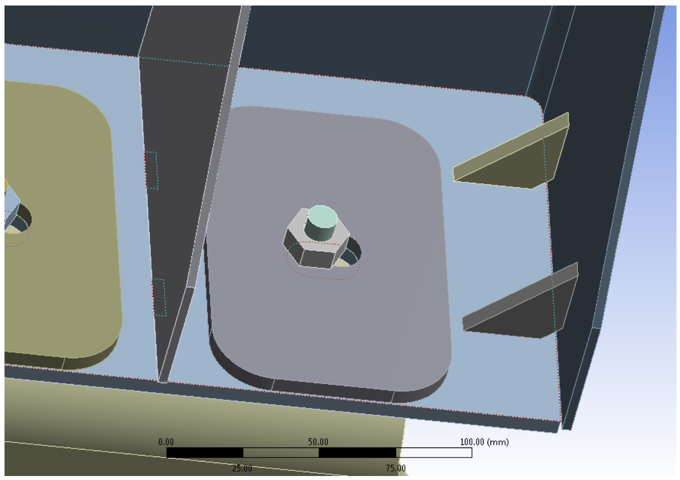


Figure 23: Trapezoid Corner Piece

The second design for the corner piece was a combination of the original part and the trapezoidal part, but also connecting fully without skipping the corner. This would provide vertical and horizontal shear protection, as well as provide the strengths of the trapezoidal shapes with the corner bending resistance.

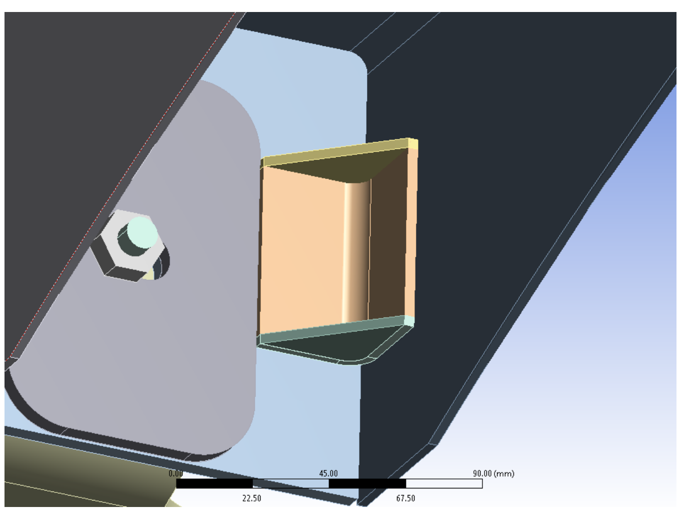


Figure 24: Combination Corner Piece

Foot Barrier & Handrails

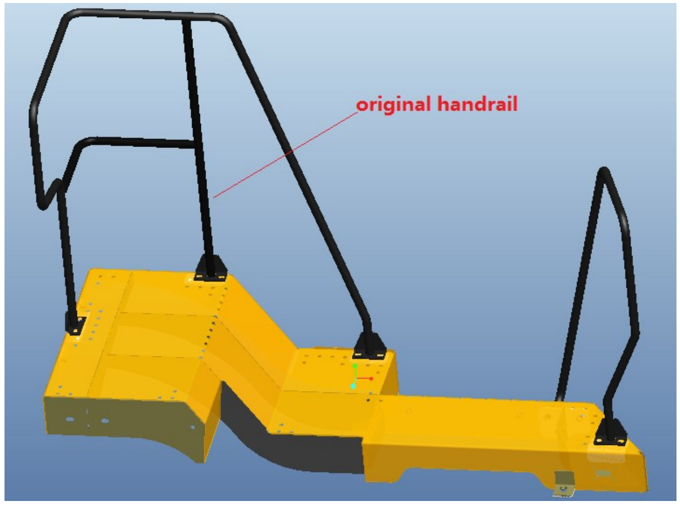


Figure 25: Original Handrail

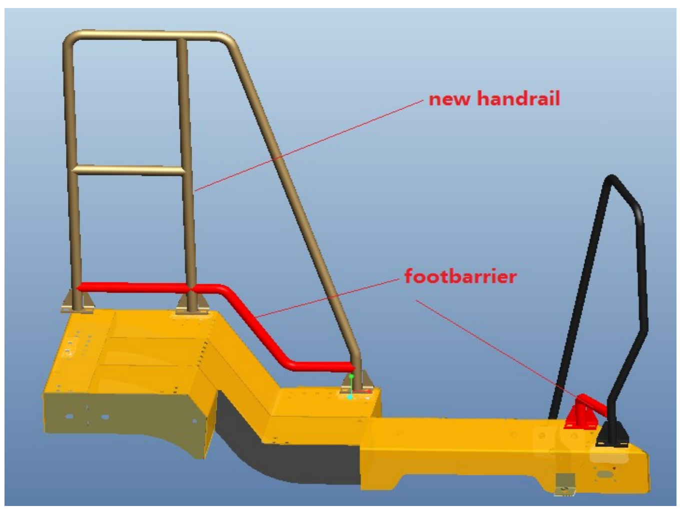


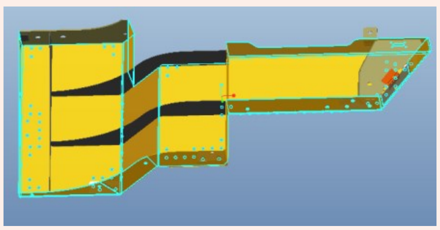
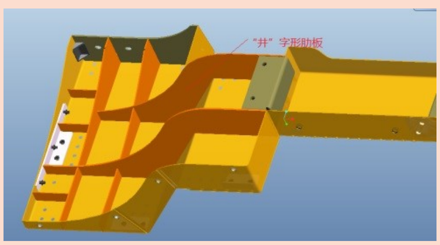
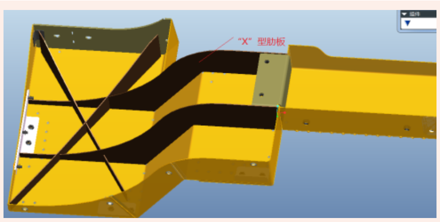
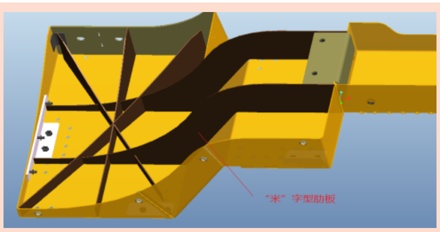
Figure 26: New Handrail

Features of the new handrail design include the prevention of objects with maximum height of 100mm from falling, and allowing access to the rear egress ladder developed by the other team. The current handrail cross sections were made from cut pipe. The foot-guard would be made of the same material parts, reducing any new parts in the process. There would be some increase in the assembly procedure complexity, but was determined necessary to fulfill the requirement.

Rib Designs

The team developed three distinctly different rib designs. Each was designed to help distributed the load, and support the platform, while directing any load to the anchor locations. The table below shows a comparison between the three new designs and the original fender rib design:

Table 1: Rib Design Comparison

Shape	Structure Model	Manufacturing	Assessment
		node / 十 / Y	
Current design		2/0/2	√
井		6/3/3	√
X		5/5/0	X
米		7/7/0	X

The three fender designs are compared above in Table 1. The “node” value, represented by the first number, is how many connection nodes there are in the rib design. The “十” denotes how many pass through nodes there are (four members on each node), and the “Y” represents how many nodes have three members. As shown, the original design has no pass through nodes, and contains two “Y” nodes. The last column is the assessment the team did on each of the options.

The “米” design was eliminated for a few reasons. While it does appear to have many supporting ribs, they are not as evenly distributed as other options, and thus have some areas where the ribs are redundant. Additionally, the manufacturability of this design is less than the others. The ribs meet at the corners where there is already stress exceeding the threshold. There are also components that could be in the way on the inside near those corner connections.

The “X” design was eliminated for many of the same reasons as the “米” design. Attaching the corners to the ribs seems like it would help some, but only if the corners were strong enough to support the load from the ribs. However, since the corners are already under a large load, the addition of the ribs connections could cause greater stress on the model.

The team’s third option, the “井” shaped ribs provided even distribution over the underside of the fender platform. The cross style design also allows the ribs to be welded to location other than the corners, distributing the load better. Additionally, the only new parts needed are the small connection parts to the left of the second rib. These small parts are easy to manufacture. The other two designs required a longer rib design to be produced, and would increase manufacturing cost and difficulty. This rib design was designated as the Parallel Ribs Design during later analysis.

Spring Damper

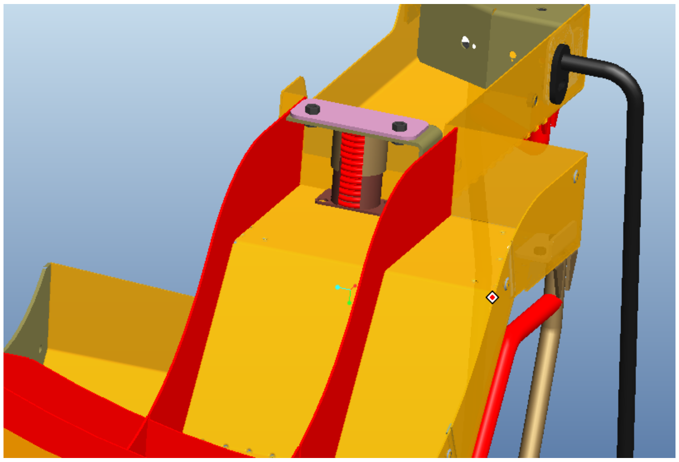


Figure 27: Spring Feature

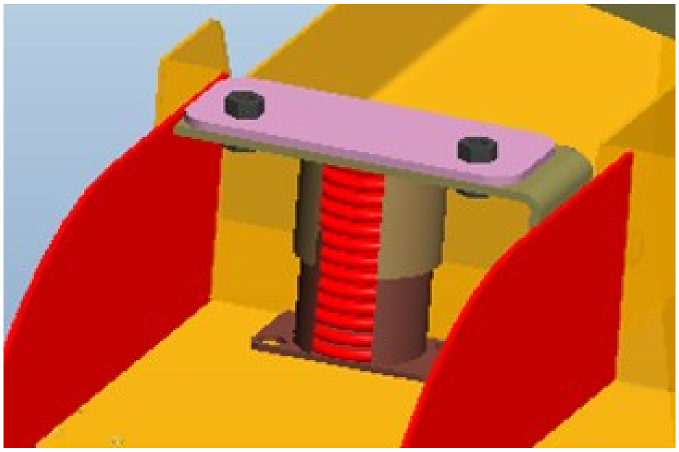


Figure 28: Spring Feature (Zoomed)

The inspiration for the damped spring design modification was acquired from a passenger vehicle. Like a vehicle shock absorber, the spring would help transfer energy through the system by first converting it to potential energy, and then in a more controlled manner release it to the rest of the platform. However, if one considers the platform in a steady state the change in energy is negligible. Instead, the spring assists the platform load by increasing the strength of the section. The spring can be further strengthened by adding a pre-load compression so that the spring pushes in tension on the platform even before the load is applied. One key characteristic is that the spring would reduce the strain on the rib connections to the anchor plate, and the bend the anchor plate takes by creating an alternate route through the spring for the load to pass through.

4.2.3 Design Analysis

Design modification and additions were put through the finite element analysis under the same constraints used for the original design. This included the 3-G canister load in the far corner of the platform. The results helped validate the designs and shows which methods did not provide an adequate solution.

Some modifications were able to be combined together, to create a modification set. For example, the reinforcement plates were also combined with the rib adjustments, as they do not conflict in their position in the system. The combined systems contained one of the rib adjustments, or a rib adjustment and a corner adjustment. The layout of the modification analysis is shown below:

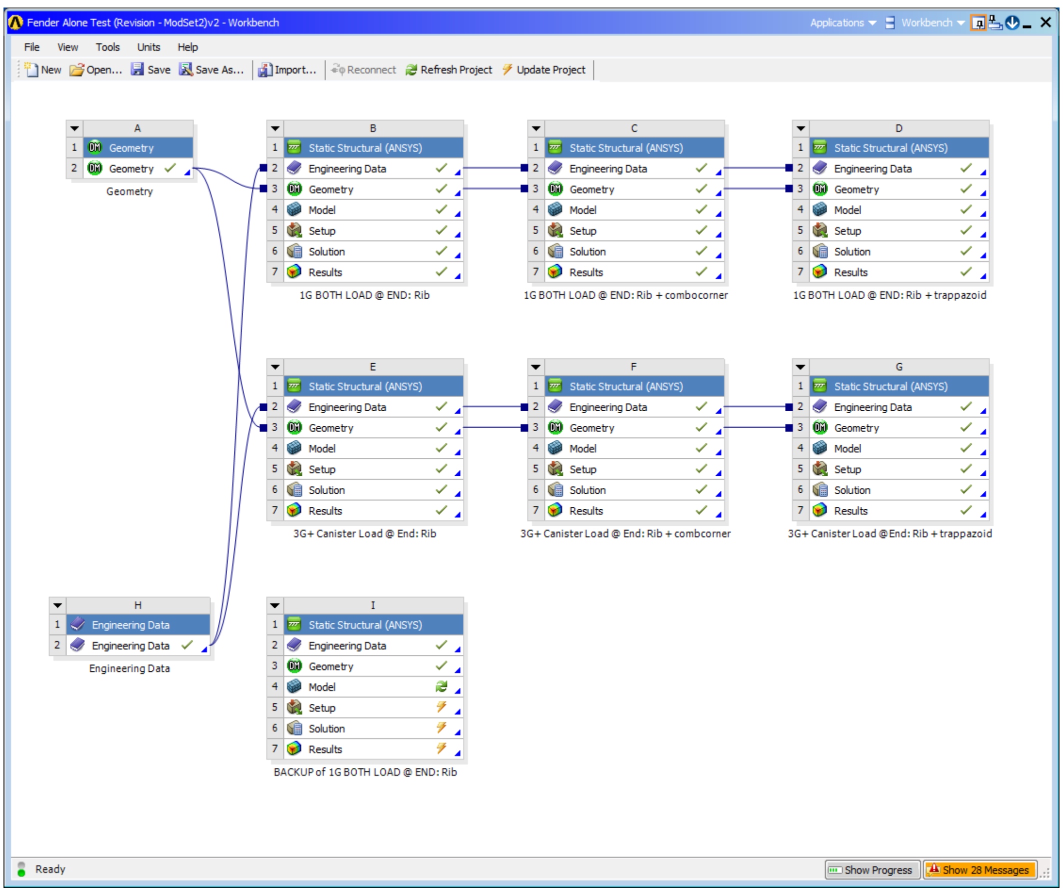


Figure 29: ANSYS Workbench Layout for Design Modifications

The damped spring and railing adjustments were not put though ANSYS Workbench. The handrails were removed to simplify the model and solution and to not rely on the handrail to provide enough stability for the platform. Relying on the handrails for stability would require that the handrails to be constantly tightened and would increase the likelihood of failure. The damped spring was not added because of the limitations of the program with the system. The idea and expected results of the damping spring were evaluated and compared in the later section.

Reinforcement Plates

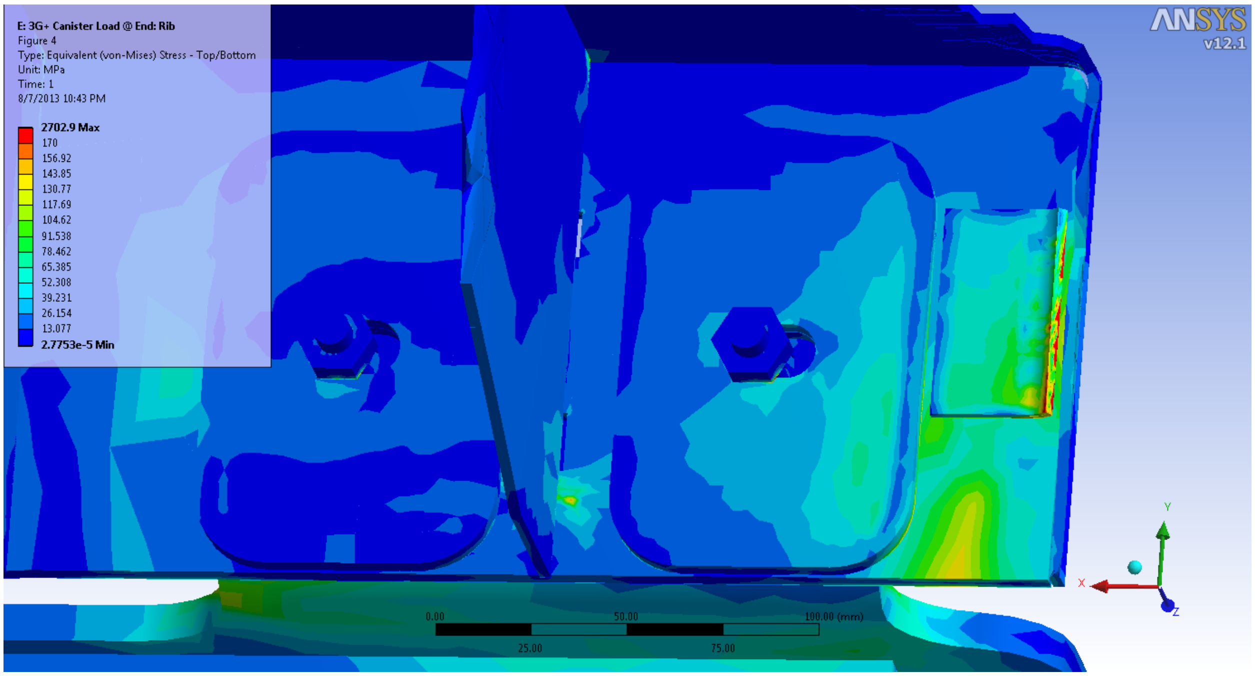


Figure 30: Reinforcement Plates FEA

The reinforcement plates used greatly improved the resulting stress around the anchor point holes. All stresses are within desired values. The reinforcement plates through the FEA process are shown to be effective to rectify the issue area. The corner part shown on the right still has some issues, and will be addressed below.

Corner Piece

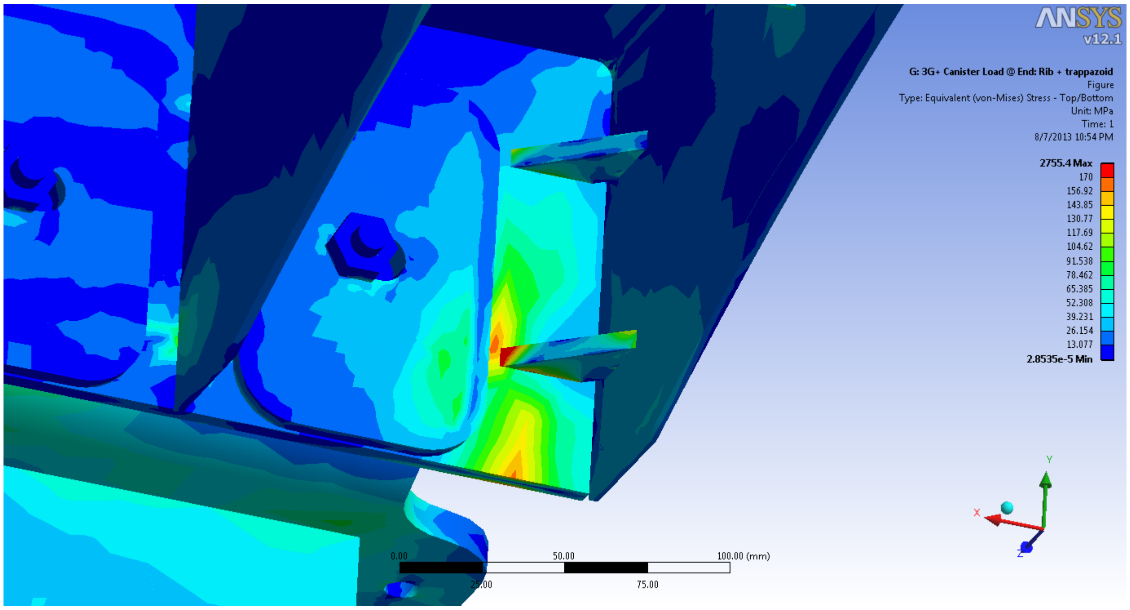


Figure 31: Trapezoid Corner FEA

The results of the trapezoidal shaped corner piece showed some promise but also room for improvement. It's clear that it helps distribute the force properly across the rib style corner piece. The issue with the design is that the tip on the lower trapezoid produced failing stresses and its design was not as reliable.

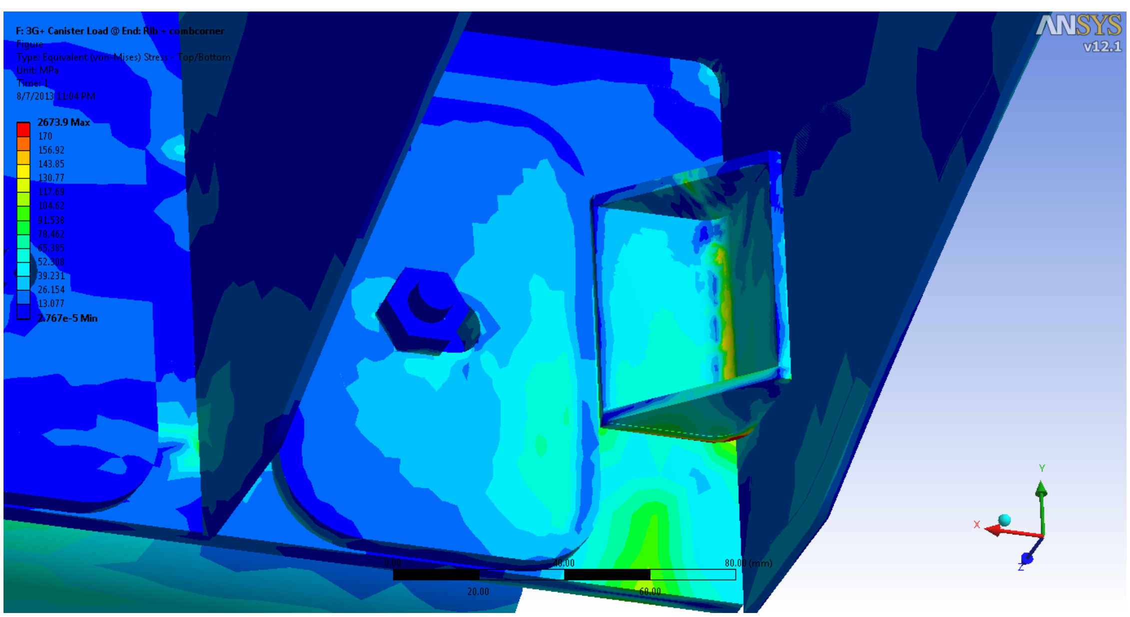


Figure 32: Combination Corner Piece FEA

The combination corner piece showed even more improvement. The backside of the original piece shows values close to that of yielding, but was determined to be acceptable. The combination design provides the advantages of both the original design (against vertical shear), with the advantages of the trapezoid style design (horizontal shear and bending). The more desirable design would be the combination corner piece.

Rib Design

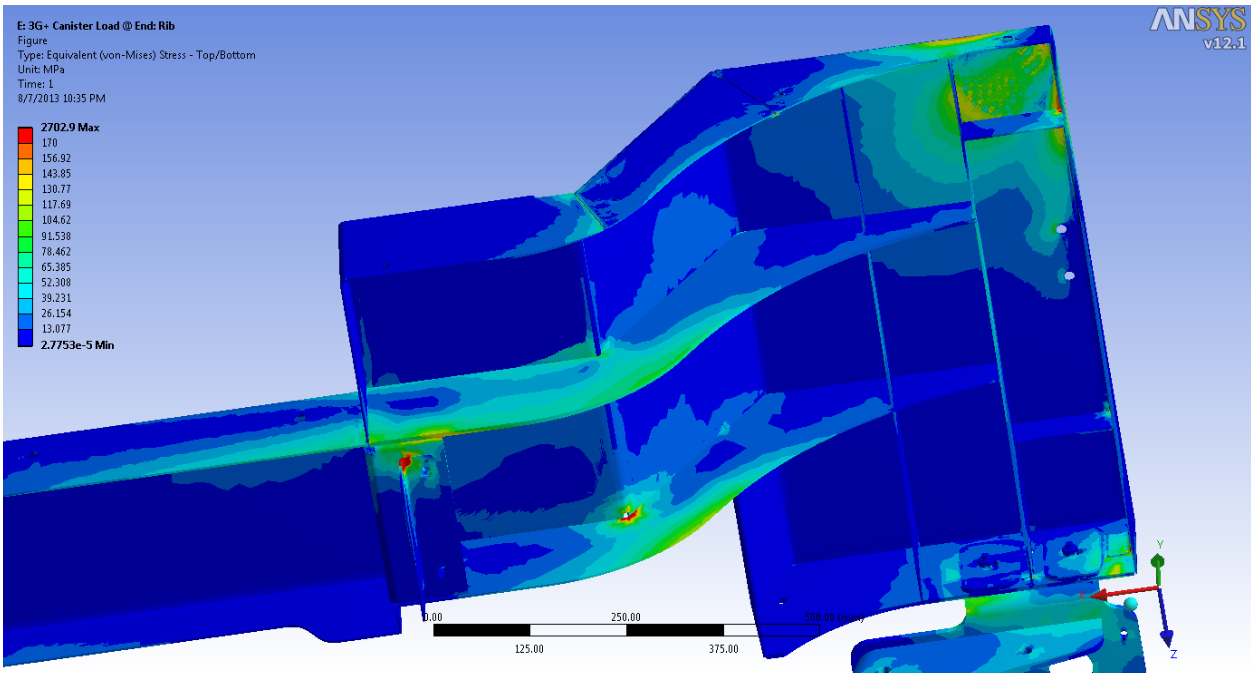


Figure 33: Selected Rib Design FEA

After eliminating two of the three rib design options during design development, the final Parallel Rib Design was processed in ANSYS. As shown above in Figure 33, there are great improvements on the stresses in the ribs themselves. Additionally, there is more balance of stress on the outer edge of the platform. The only place of concern that remains is on the anchor point on the underside of the fender system. The dampened spring was seen as a possible solution to the yielding in that location. As well, it's important to note that the surface parallel to the platform floor will also be distributing some load due to contact pressure, and therefore the full load will not be distributed along the single edge of the lower anchor part. Because the part barely reaches yielding values, and is likely due to the inaccuracy of the boundary conditions, it was assumed the new rib design produced adequate results.

4.2.4 Design Evaluation

The team structured the evaluation so that it can be performed in an efficient manner while ensuring that the final design would be effective as a solution to the problem. Each design goal topic, as mentioned in the previous sections, was compared against each other. The one with the highest score has the most importance in evaluation. This preliminary process gave a ranking of the goals. Inelastic deformation and safety proved to be the most important factors, while cost was the least important. While cost is important to the product sales, the fender design target audience is already a small group, and the custom modification cost is already a known cost to the buyer. The cost would weigh more for the standard fender without the fire suppression canister load accommodation.

Table 2: Assigned Weight Factors

Design Specifications	Ranking	Pairwise Score	The Weight Factor
Inelastic Deformation	1	2	4
Safety	1	2	4
Supplier Tooling	2	1.5	3
Cost	3	1	2
Manufacturability	2	1.5	3

Weight Factor = 2 * Pairwise Score (This equation is for the convenience of computation)

Once the weight factor for each category was developed, each design was evaluated on each category. While some of the values were based off quantitative results like the FEA portions, others were discussed and estimated based on reason and logic within the group discussion. Rating is done regarding each design's performance under each design goal. The results are displayed in the following section.

Rating Basis:

1 – Poor, 2 – Fair, 3 – OK, 4 – Good, 5 – Excellent.

Each design and its corresponding value for the decision matrix are listed on Table 3: Decision Matrix Key.

Table 3: Decision Matrix Key

Design	Corresponding Value in Decision Matrix
Reinforcement Plates	D1/S1
Foot Barrier and Handrail	D2/S2
Parallel Rib Design	D3/S3
Combination Corner Piece	D4/S4
Spring Damper	D5/S5

Table 4: Decision Matrix

	WF	D1	S1	D2	S2	D3	S3	D4	S4	D5	S5
Inelastic Deformation	4	5	20	4	16	4	16	4	16	4	16
Safety	4	3	12	4	16	3	12	4	16	3	12
Supplier Tooling	3	3	9	3	9	3	9	3	9	1	4
Cost	2	3	6	3	6	3	6	5	10	2	4
Manufacturability	3	4	12	3	9	2	6	4	12	3	9
Total			59		56		59		63		45

Table 4, the decision matrix, uses the following equations and variables: Total = Σ the Weight Factor * Rating. WF is the Weight Factor for the category. D# is the Design Rating value. S# is the calculated score for the design based on the Weight Factor.

The decision matrix provides an evaluation of the five designs the team generated. Design 4, the Combination Corner Piece design, has the highest confidence level with regard to all the design goals. From the matrix the team feels it would provide the best improvements in comparison to its increase in parts required and manufacturability. Other designs may have greater impact, such as the rib design, and the spring system, but they also have more difficulty in implementation.

The spring dampener would require at least three new parts, completely unique from any others to be developed and assembled in the system. As well, having a pre-loaded spring requires special techniques in installation of the part. The reinforcement plates and the rib design scored high on the decision matrix and are highly recommended in the design.

4.3 Design Summary

It was concluded that all the designs evaluated would benefit the fenders response to the load application. The corner piece offered to meet the expectations the best at minimal cost and hindrance to the current manufacturing process, but is not the only design the team recommends be implemented. Each design was applied to a specific section of the fender system, and no designs contradict or interfere with one another. Therefore, the team recommends that all designs be implemented to allow the fender platform to handle the increase load requirements while still maintaining all design goals and requirements.

5.0 Conclusion and Future Suggestions

In this project the team developed and executed a plan to evaluate and redesign the 972H front wheel loader fender platform, as requested by Caterpillar Inc. The increase load on the fender, as a result of installing a fire suppression system, was investigated to assess the current structural integrity and performance of the existing model design. Using logical and simulation information, the load was applied to locations throughout the fender to determine the positions where the final analysis would take place. Once analyzed, the team developed a list of areas that require improvement in handling the system stress, strain, as well as deformation from the applied load. These designs were then revised, analyzed using finite element analysis when applicable, and evaluated to show the advantage of each design in reference to both the overall platform performance as well as the manufacturability and cost. The team developed designs for reinforcement plates, handrails to meet all ISO requirements, corner reinforcement parts, damped spring support system, as well as redesigned structural ribs.

The project conducted has real world application within itself, but also helped the team to develop skills for the situation of engineering analysis and design. The group found the creation and implementation of the plan to require great organization and project planning. These skills go beyond this project alone, and can be applied to almost any engineering application. Additionally, the team's ability to creatively develop solutions to the defined problem, and then evaluate and validate the solutions logically and experimentally shows the ability of creative engineering and design processing.

The material in the project could be used by Caterpillar Inc. to develop further solutions and modifications to their 972H fender, to allow for the extra load of the fire suppression system. The results in this project could be used directly, but can also facilitate in future evaluation and creation of designs for strengthening the fender. The team recommends that the designs be developed in a prototype fashion before being put into any manufacturing process. Further investigation into the current performance of the original fender design through physical experimentation, such as strain gages or laser measurements, would provide the experimental performance of the platform to compare and contrast to the theoretical performance. Additionally, experiments and analysis into the welding process and any great effect is has on the results would be suggested to further evaluate the design and the process.

The project sponsor found the project to be creative in its solution, admitting "*that [they] have not thought of using energy absorbers to limit stress… The use of a spring as a 'stress accumulator' is actually very interesting. [They] had thought of this once previously for a different part (rear axle) on the wheel loader but never for a strictly structural part. Creativity is the reason that the team engages student teams."* The project team hopes that Caterpillar Inc. find the project useful and helpful in the design process for the 972H front wheel loader assembly and design, further assisting in Caterpillar Inc.'s core principle for safety and reliability in their products.

Bibliography

1. GB/T 17300-2010/ISO 2867:2006 Earth-moving machinery – Access systems[S] Beijing: Chinese Standards Publishing House 2010.12
2. GB/T 699-1999Premium carbon structural steel [S] Beijing: Chinese Standards Publishing House 1999.11
3. GB/T 710-2008 Premium carbon structural steel, heat treated sheet metal and steel belt [S] Beijing: Chinese Standards Publishing House 2008.12
4. He Yuchen Pro/Engineer Wildfire Modeling Basics [M] Beijing: Qinghua University Publishing House 2004.7
5. Second Generation Dragon Shock Studio Pro/Engineer Wildfire Advanced Design[M]Beijing: Electronic Industry Publishing House 2004.6
6. Zhang Peiqi, Pei Jianchang, Huang Shengjie Pro/Engineer Wildfire Advanced Design [M] Beijing: People's Post and Telecom Publishing House. 2004.1
7. ISO 2867:2011 Eart-moving machinery –Accecss systems
8. LaForest, Bradley J. "Re: Weekly Report of Group 1." Message to Weitai Hu. May 8, 2013. E-mail.
9. LaForest, Bradley J. Caterpillar (Suzhou) Co., Ltd., Wuzhong, Suzhou, Jiangsu China. June 2013. Conference Presentation.