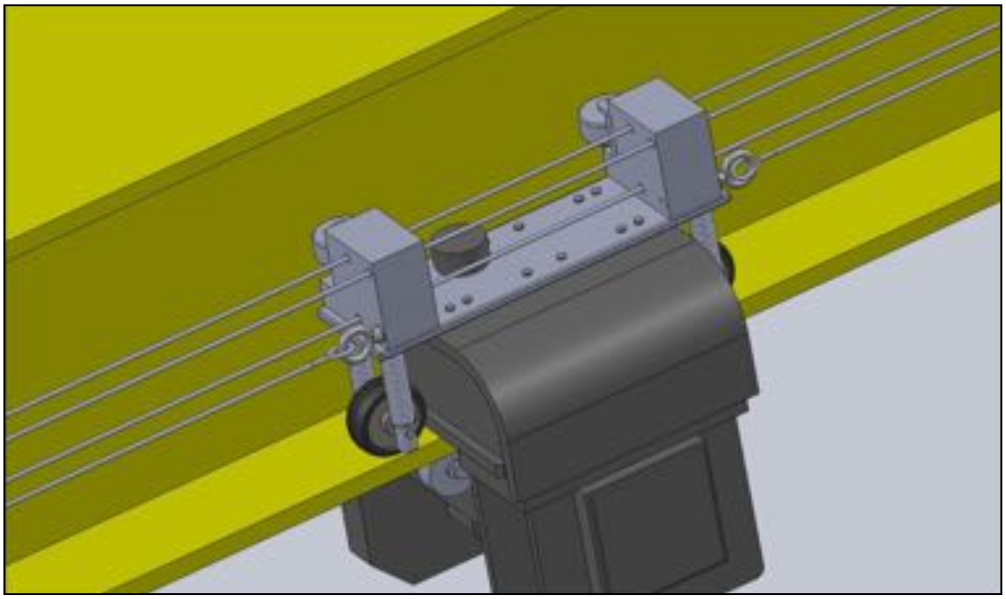

ME 461:
Finite Element
Analysis

Spring 2016

A Semester Report on:

The Development and Analysis of an Overhead Gantry Crane Trolley

Group Members:

Scott Babilya, Jay Hauseman, Nicholas Hesse

PennState College of Engineering

Department of Mechanical and Nuclear Engineering, University Park, PA

Table of Contents

Executive Summary

The purpose of this project was to perform Finite Elements Analysis (FEA) on the trolley assembly of an overhead gantry crane. This is an extension of a capstone design project performed by Jay Hauseman and his teammates, Vivek Patel, Dan Andrus, and David Wasik. Analysis was performed to determine whether the plate and shock assemblies were of robust enough design.

CAD geometry of the trolley assembly of the gantry crane was provided by the capstone team. This geometry was imported and meshed using Abaqus. A complex meshing scheme was originally implemented, which was later simplified. The end analysis omits several parts of the trolley assembly, most notably the slider, which consumed too many resources and had a geometry that was not easily meshed. The final analysis replaces the omitted parts with boundary conditions.

The design criteria for the plate were set at a factor of safety of 2 after 100,000 loading cycles, and a maximum deformation of 0.1 inches at any point in the assembly. The max stress generated from the finite element analysis was 14710 psi. When compared to the yield stress of 54,000 psi, this yields a factor of safety of 3.6. However, this part experiences dynamic loading, causing fatigue. The fatigue limit of steel is assumed to be one half of the yield strength. This causes the factor of safety to drop down to 1.8, which is below the design criteria of 2. This suggests redesigning the plate, making it thicker. The maximum deformation of the plate was 0.0297 inches, which is well below the design limit of 0.1 inches. The body of the shocks experiences a relatively small stress, which means that the body of the shocks are over designed.

Acknowledgements

We would like to thank Dr. Reuben Kraft for all of his help throughout the duration of this project. We would also like to thank the capstone team of Jay Hauseman and his teammates, Vivek Patel, Dan Andrus, and David Wasik for allowing us to use their capstone project and their CAD models.

List of Tables and Figures

Section 1: Background and Project Plan

Objective:
To determine the failure points and loads in a plate used as part of an overhead gantry crane trolley assembly.

Background Information:
High Concrete has commissioned a Penn State Team of senior mechanical engineers to design a motorized system to move four trolleys along an I-beam gantry crane. The conceived solution uses a shock system that has a few unique components that have not been tested. These parts include a plate that attaches the shock system to the trolley, two vertical rods, a wire guide and an eyehook. The main area of concern is the two guide holes that the vertical rods fit through. This area could create a stress concentration high enough to fail and requires an FEA analysis to prove that the concept is structurally sound. The loads are known in this system and include a tensile force that is supplied by the wire which pulls the trolley and a weight force that originates from the weight of the trolley. See the attached Figure 1: Trolley Assembly with Applied Loads for a clear visual of the assembly and Figure 3: Detailed Plate Drawing for an isolated view of the plate in question.

General Approach:
We will create an assembly in a CAD software consisting of the plate and two wire guides. In ABAQUS, we will apply relevant surface contacts and constraints to ensure each part behaves correctly in the context of the assembly. Concentrated force loadings will be applied to relevant surface of the eyehooks. The mesh in particular will vary in density, with high numbers of elements near corners, holes, and other stress concentrators and areas of high stress gradient.

Section 2: Development and Description of the CAD Geometry

Preexisting CAD:
The CAD files for this project were all previously created for a capstone project completed in Fall of 2015 by Jay Hauseman and his teammates, Vivek Patel, Dan Andrus, and David Wasik.

Table 1: Material Properties

Property	Plate and Slider	Shock
Material	1018 Steel	4140 Steel
Density (lb/in^3)	0.283	0.283
Yield Strength (psi)	54000	60200
Poisson's Ratio	0.29	0.28
Young's Modulus (ksi)	29700	27557
Hardness	Rockwell B 70	Rockwell B 92
Shear Modulus (ksi)	11600	11600

Loading Conditions:
The loads in this structure arise from two sources. The body force from the weight of the assembly, which is roughly 400 lbf, is the primary load. This value was calculated by determining the volume of the parts and multiplying by the density. The secondary load is a pair of 100 lbf tensile forces that are attached at the shocks. These forces are caused by guide cables that are used to move the trolley laterally along the beam. The cables were designed specifically to have up to 100 lbf of tension to prevent sag along the beam length.

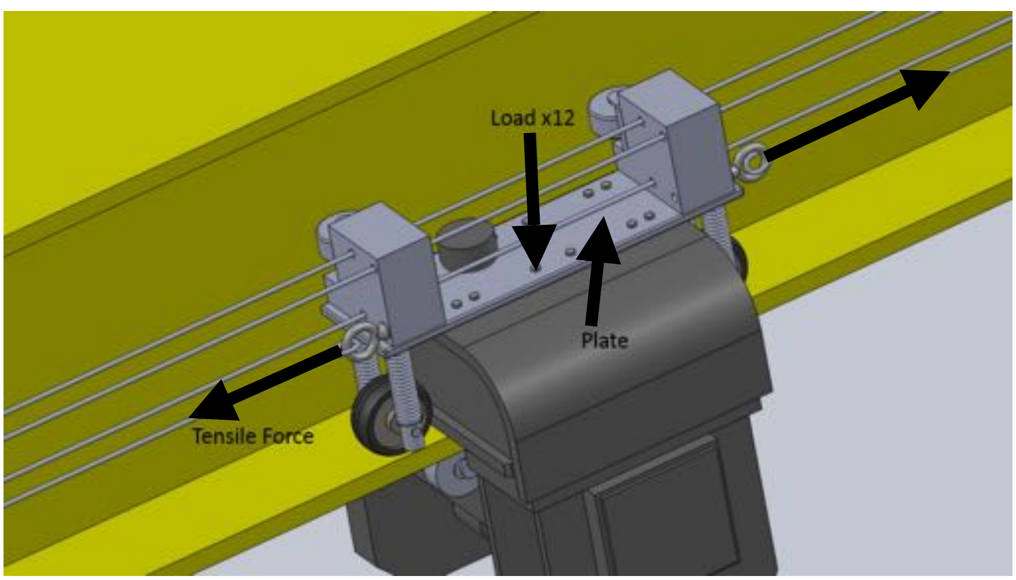

Figure 1: Trolley Assembly with Applied Loads

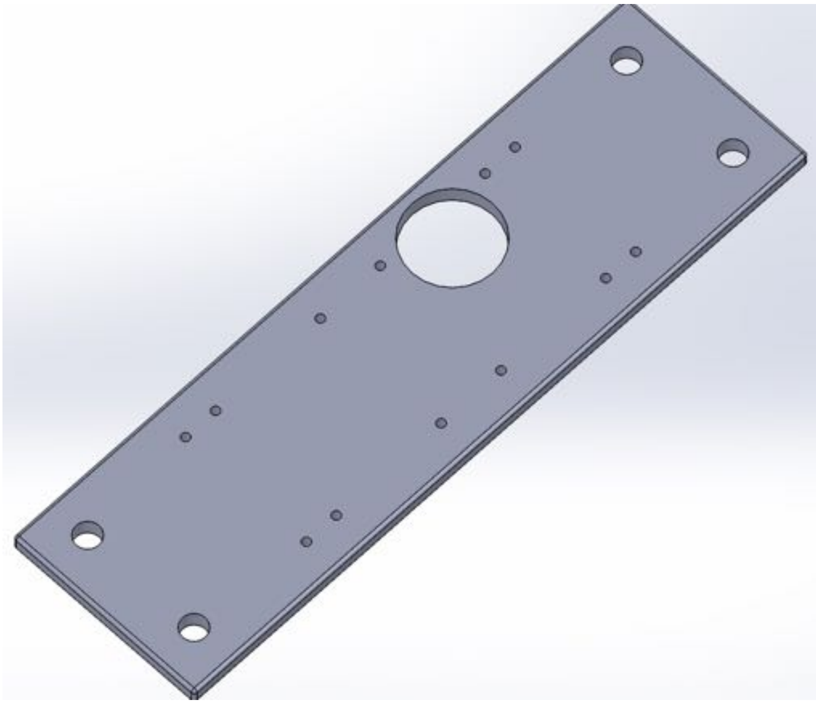

Figure 2: Trolley Plate

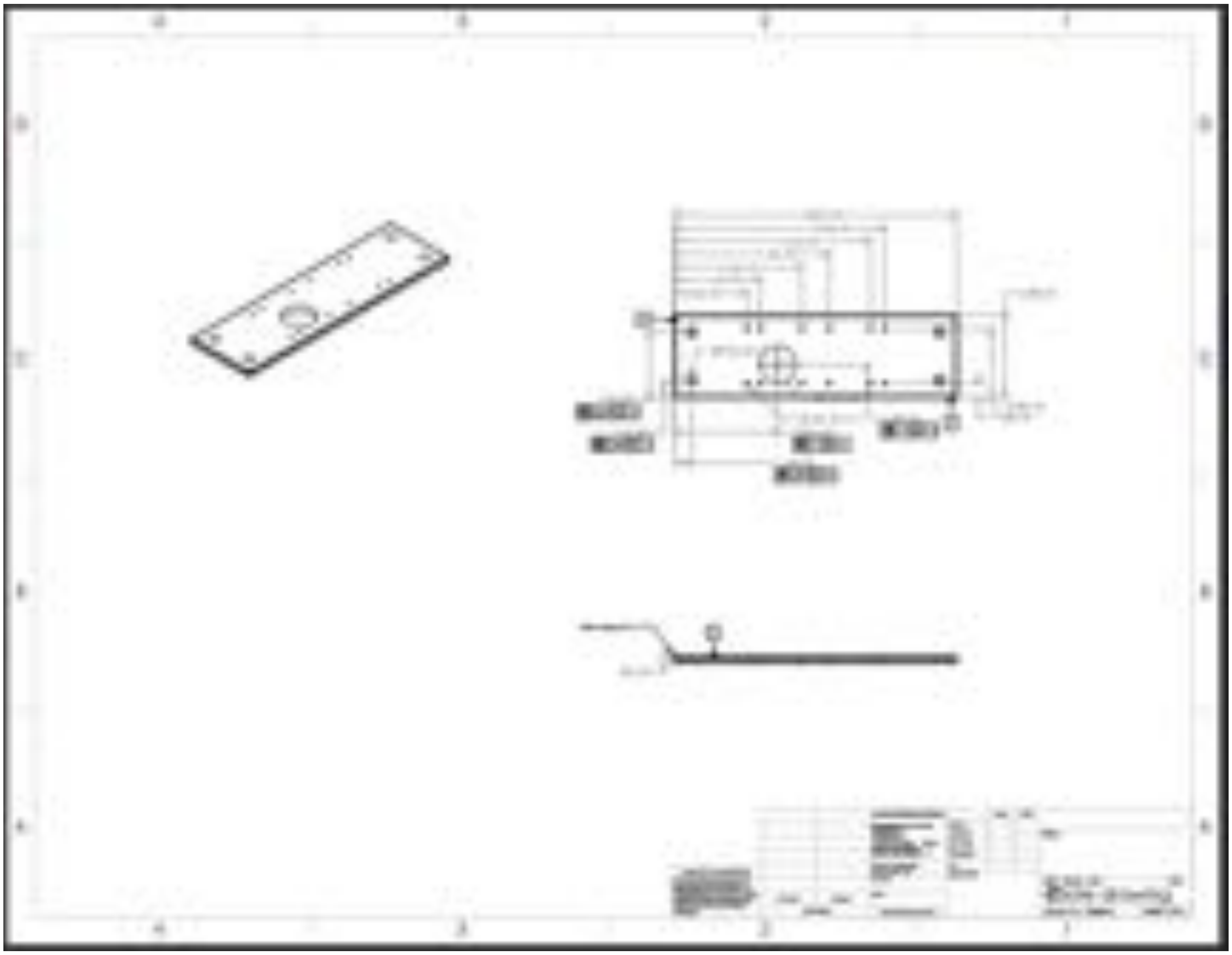

Figure 3: Detailed Plate Drawing

Section 3: Development of Finite Element Meshes

Plate Mesh

Early in the analysis of the gantry crane assembly it was determined that a major focal point of stress would be in the holes of the plate that secures the shock system to the original slider. Because of this the mesh for the plate part needed to be crafted in a precise manner. To begin, the initial seeding was put at a global size of 1. The areas that were targeted for finer meshing were the edges that are close to the holes the shock's shafts interact with. This local seed size was decreased to 0.1 to add detail. The seeds around these holes were also decreased to a size of .1. The verify mesh tool was used to find any problem areas which occurred along the long edges. Making the elements 0.5 global size fixed the issue along with making a partition through the thickness of the shaft. With these fixes there were 5 warnings out of 123,137 elements. A standard steel material with properties identical to those given in class (but converted to appropriate units) was assigned to the component.

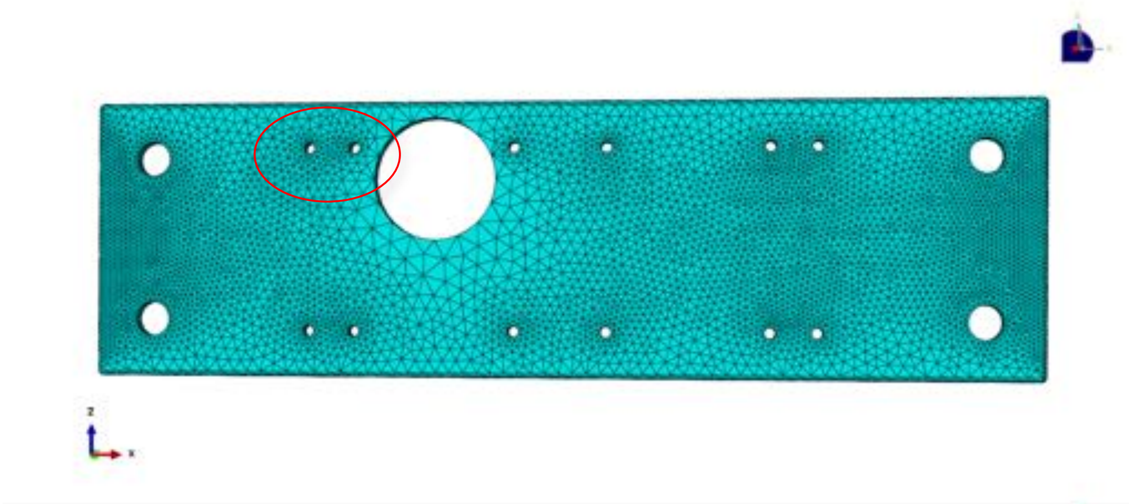

Figure 4: Plate Mesh

Figure 5: Plate Mesh

After some initial Abaqus runs were made, a simpler mesh needed to be put in place. The resources of the fine mesh took too much time to complete, and in some cases crashed the simulation. The simplified mesh of the plate is shown below. This mesh keeps the default mesh that Abaqus creates. As shown, the holes still have finer meshes around them in comparison to the rest of the plate.

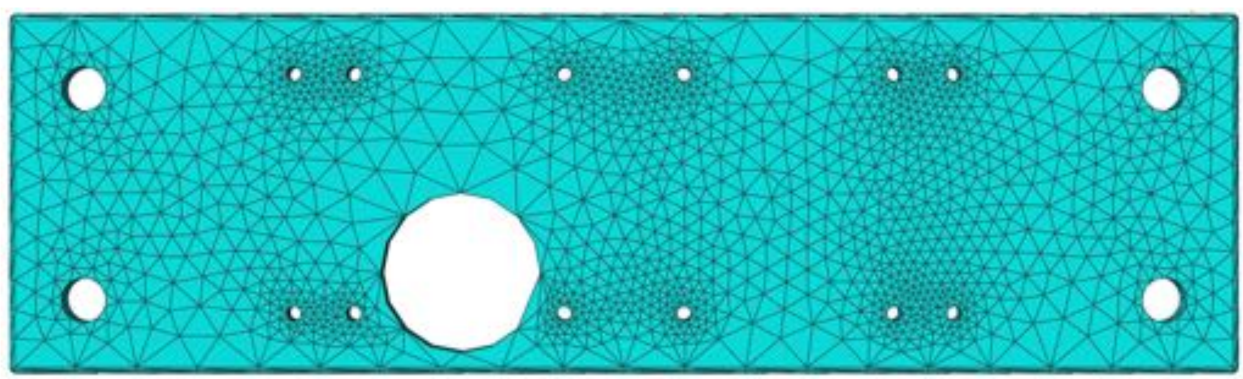

Figure 6: Simplified Plate Mesh

Shock Mesh

Another major part of the gantry crane assembly is the shocks. To create this mesh, a global tet mesh was first applied over the whole part. This mesh was seeded with a global value of 2. In earlier version of the mesh, the density was increased in areas of stress concentrations, but it was found that stresses were so low that the increase in resolution was not worth the extra computational time. The mesh consists of 4371 elements, and produced no warnings or errors. The shock is made of standard 4140 Steel.

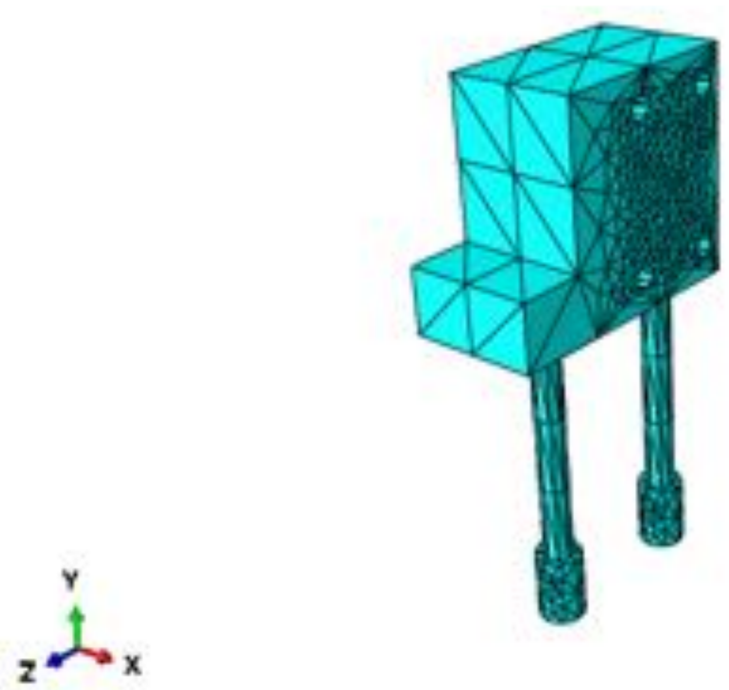

Figure 7: Shock Mesh

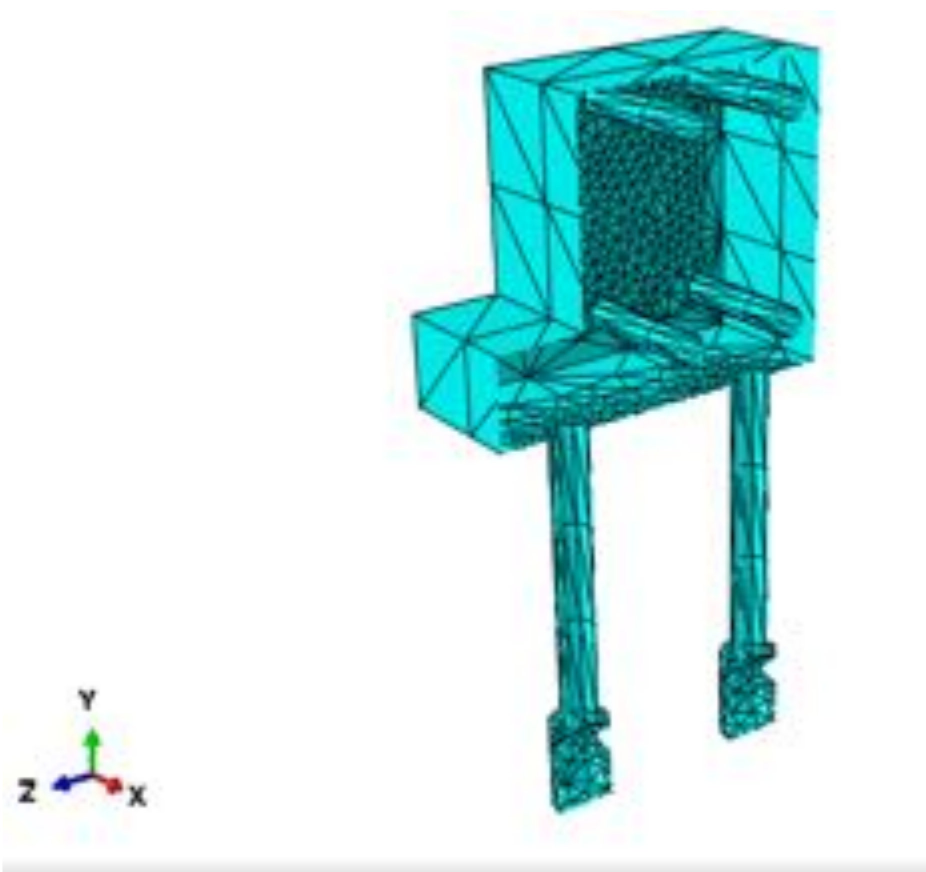

Figure 8: Shock Mesh x-plane Cut

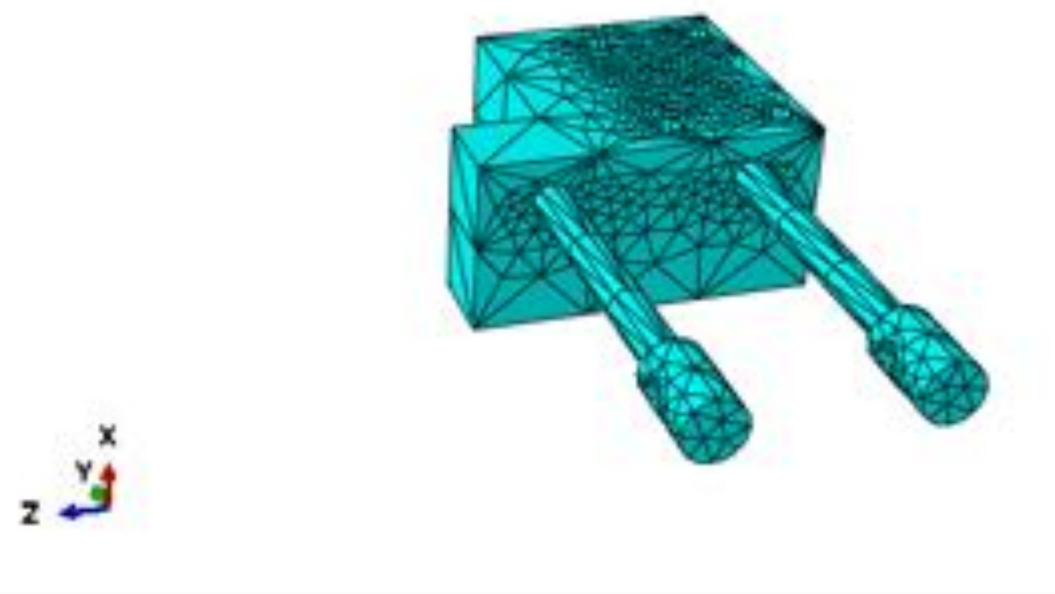

Figure 9: Shock Mesh Underside

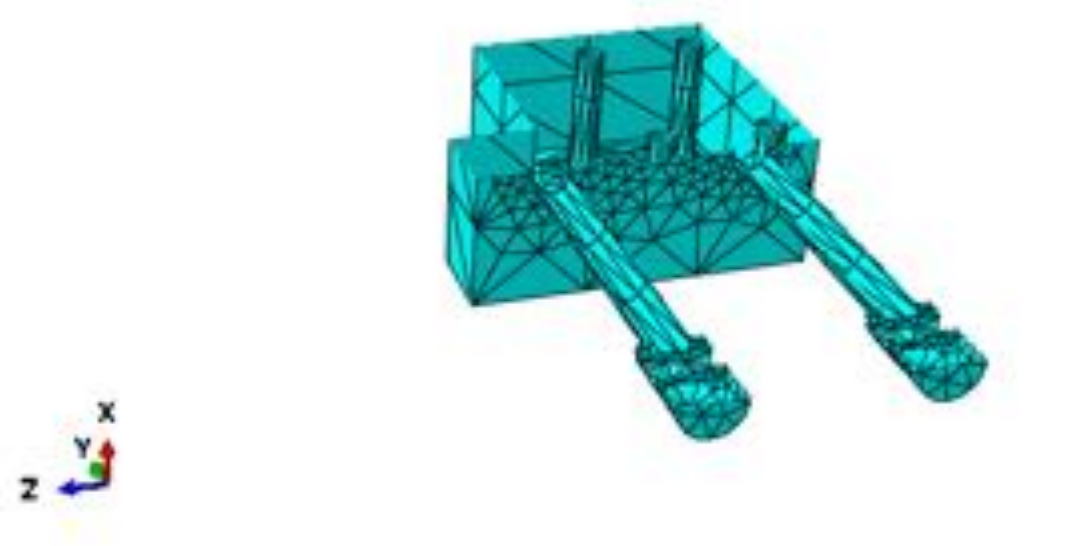

Figure 10: Shock Mesh Hole Cutout

Pin Mesh

The pin rod took a global hex mesh of 0.1. This yielded no warnings. The pin rod is also made of standard steel. The pin was eliminated from the final assembly for run time purposes.

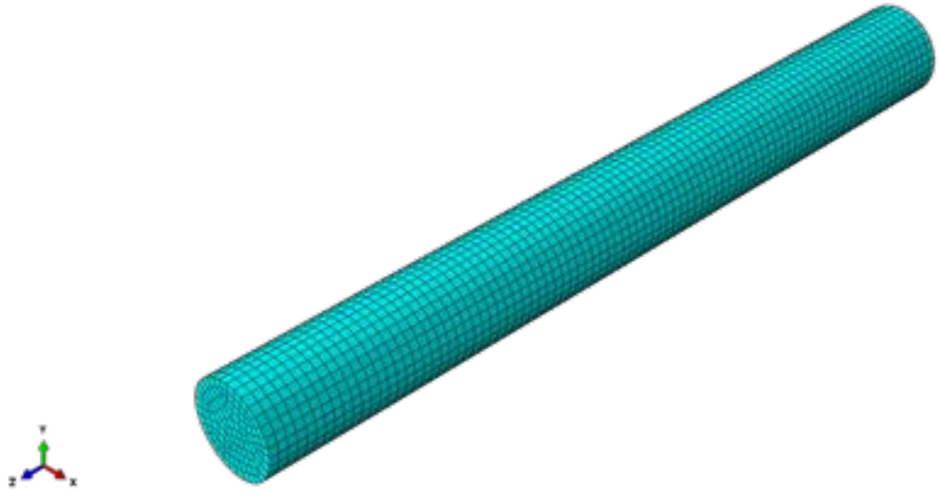

Figure 11: Pin Mesh

Slider Mesh

The largest component of the assembly, though not the one mainly of interest to this analysis, is the large slider that supports the main load of the gantry crane. This part is designed to carry a capacity of 13.75 tons, so failure of this part is not expected in any portion of this analysis. It mainly serves to act as a heavy load that exerts a force on the other components of the assembly. Because the accuracy of this mesh is not paramount, it was seeded with a 1 inch global mesh. At several inner 'steps' that appear to be stress concentrators, an edge mesh control of 0.5 in was applied. After mesh analysis, only 3 of 34170 nodes were found to have warnings, and all of these were in the lower corners of the overhang found in the bottom left of Figure 12: Slider Mesh. The material applied to this component was standard Steel.

After some initial trials were run, the team eliminated the slider from the assembly. The complex geometry was leading to problems. Also, the slider consumes a large amount of resources, which slowed the simulation down. The wheel was also eliminated from the final assembly to cut down on run time.

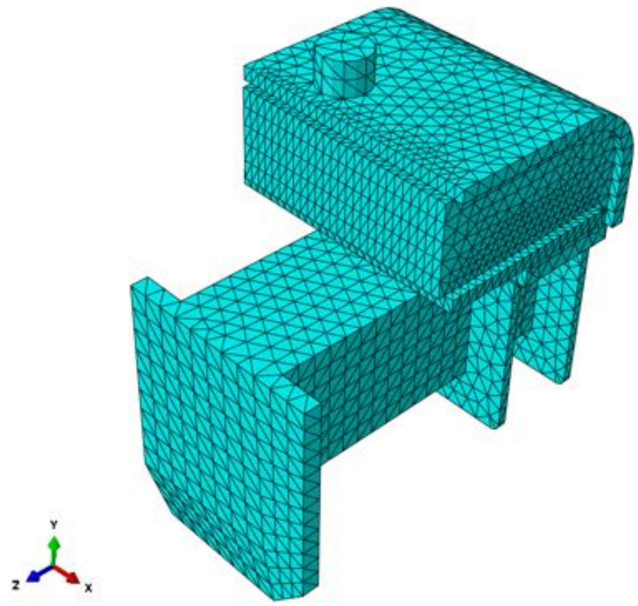

Figure 12: Slider Mesh

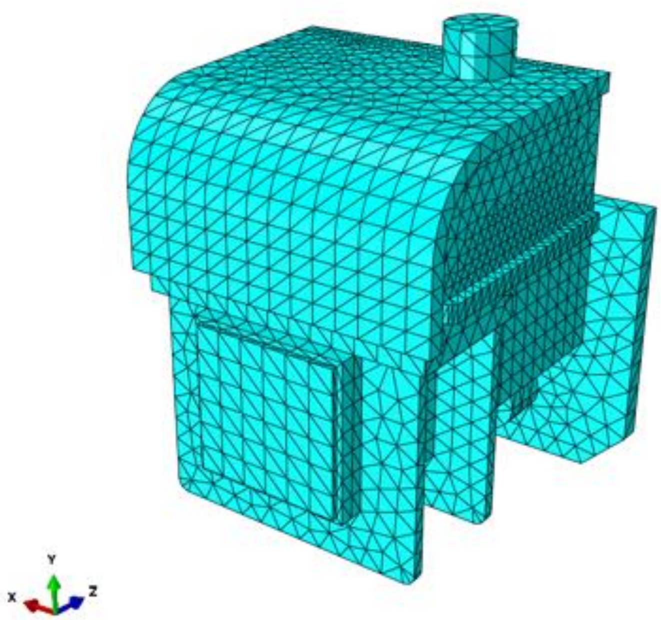

Figure 13: Slider Mesh Opposite View

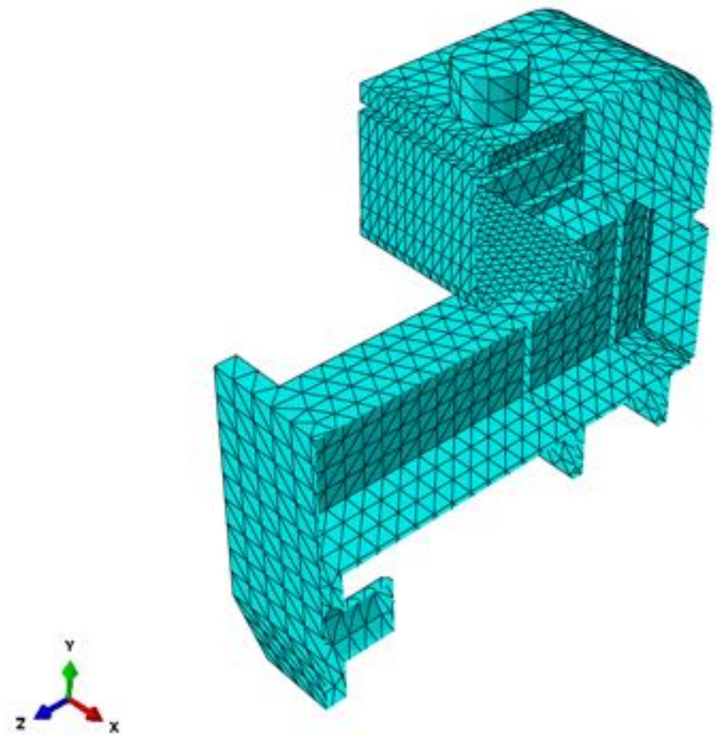

Figure 14: Slider Mesh Cutout

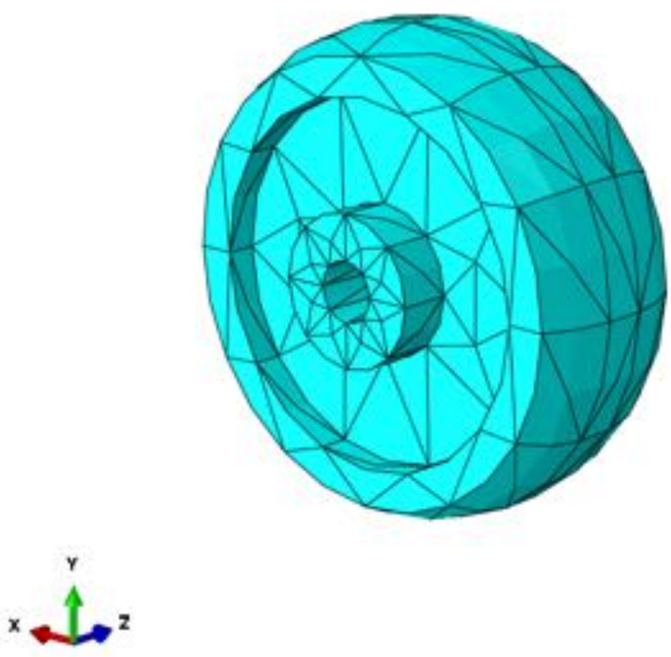

Figure 15: Wheel Mesh

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

Boundary Conditions

To accurately model the I-beam and missing components in this simulation, boundary conditions had to be carefully applied. The first boundary condition is an ecastre boundary condition on the bottom end of each shock. This simulates a fixed or braked trolley.

Figure 16: Boundary Condition One

Loads

The first load applied to the assembly is to simulate the weight of the slider across the bolt holes. To do this, a compressive force around the edges of the bolt holes were applies. The force is applied on the top surface to match the compressive force of the bolt head contacting the plate.

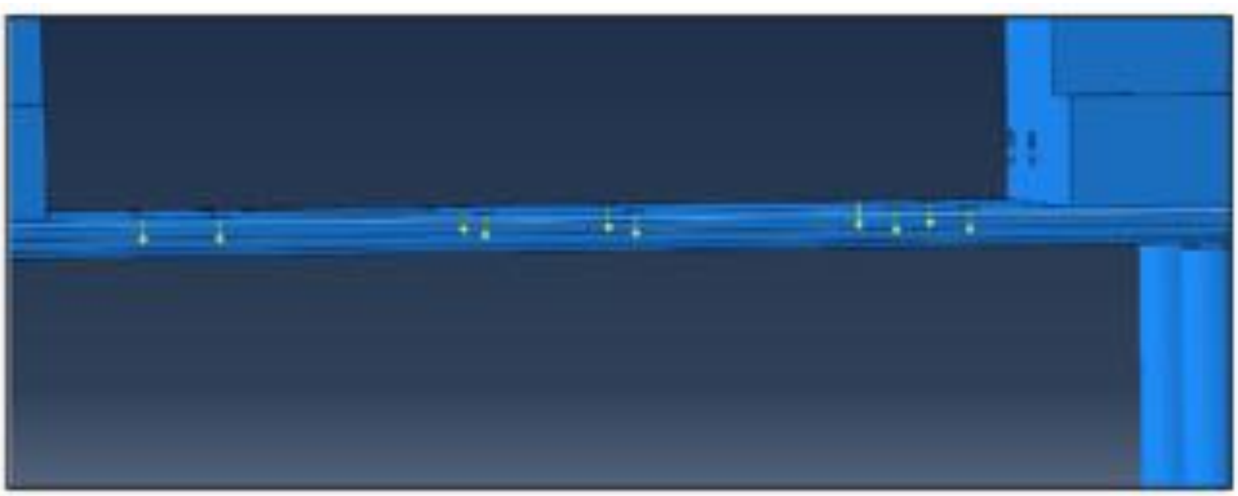

Figure 17: Simulated Slider Load

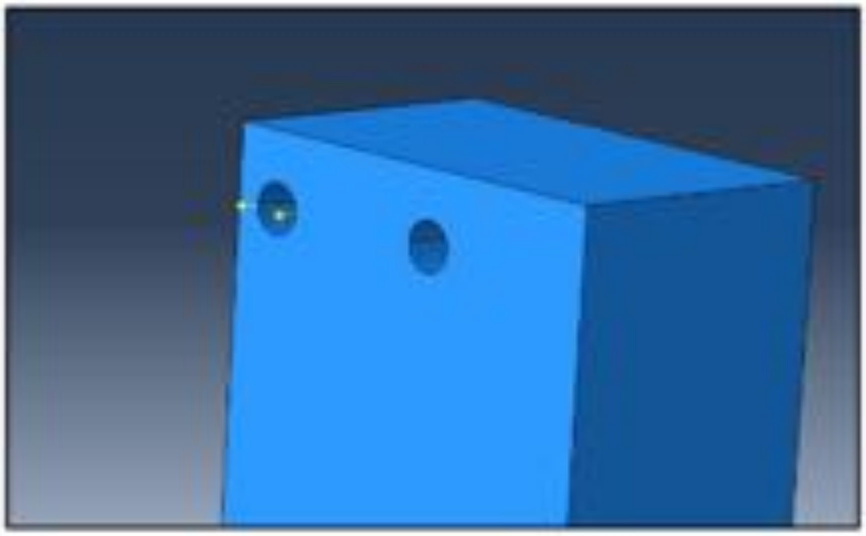

Figure 18: Wire Tension Load

The load that the project focuses on is the tension force of the wires that pull the trolley. This force was pre calculated and determined to be 100 lbs. The eye hook that the wire loops through has been removed because the eye hook has a certified factory of safety and is not of our concern in this project. Instead the force is applied directly at the hole where the eye hook would be threaded in. This load is split into two 50 lb concentrated forces on the wire guide hole.

Section 5: Development and Description of Model Interactions

After the boundary conditions have been applied, the interactions between the different parts need to be defined. The only interactions that happen are the friction interactions between the different steel parts. This is done by applying a tangential behavior interaction, with the penalty friction formulation method. The friction coefficient between steel and steel is 0.65. This interaction is applied globally, as each part in the assembly is a different type of steel.

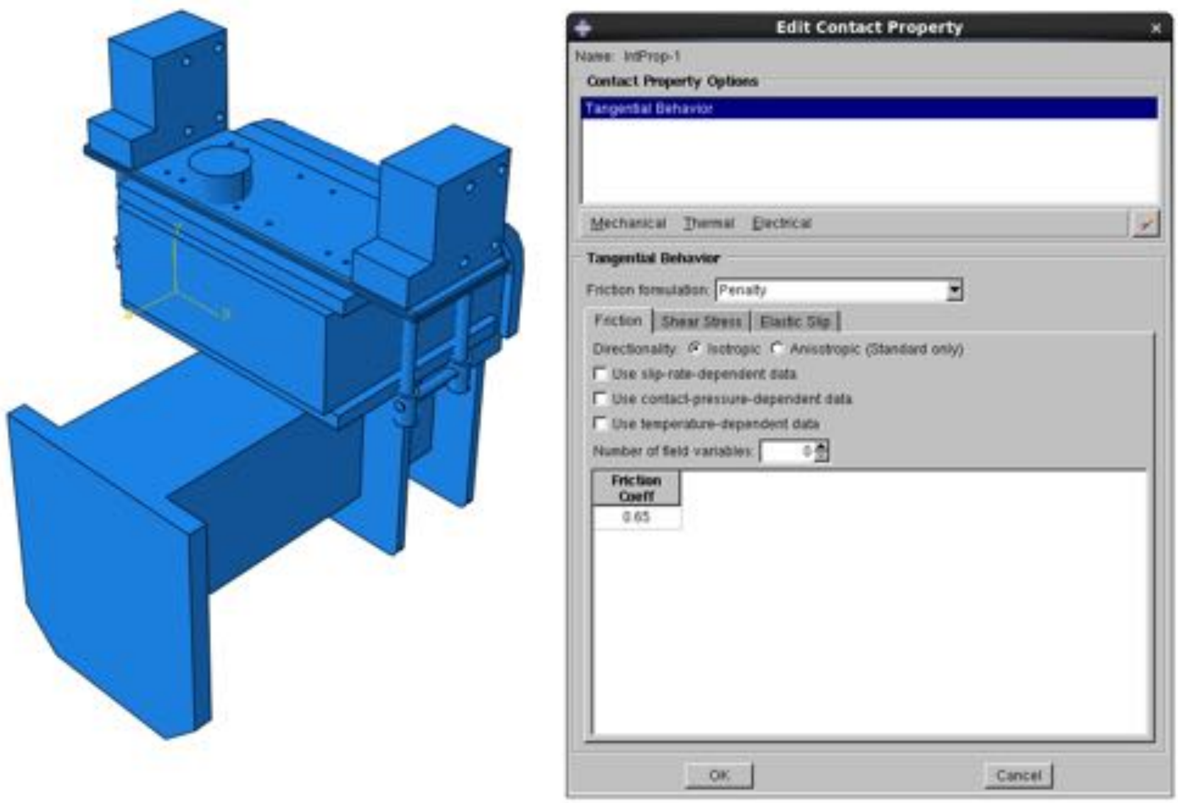

Figure 19: Friction Application

Another interaction that was intended to be modeled are the fasteners that connect the plate of interest to the slider. This is done in lieu bolts, which modeling would consume computational power. The bolt holes are shown in Figure 20: Fasteners for Bolt Holes. The slider was eliminated from the final assembly, so the fasteners were not applied. In lieu of the simulated bolts, a compressive force is applied. This

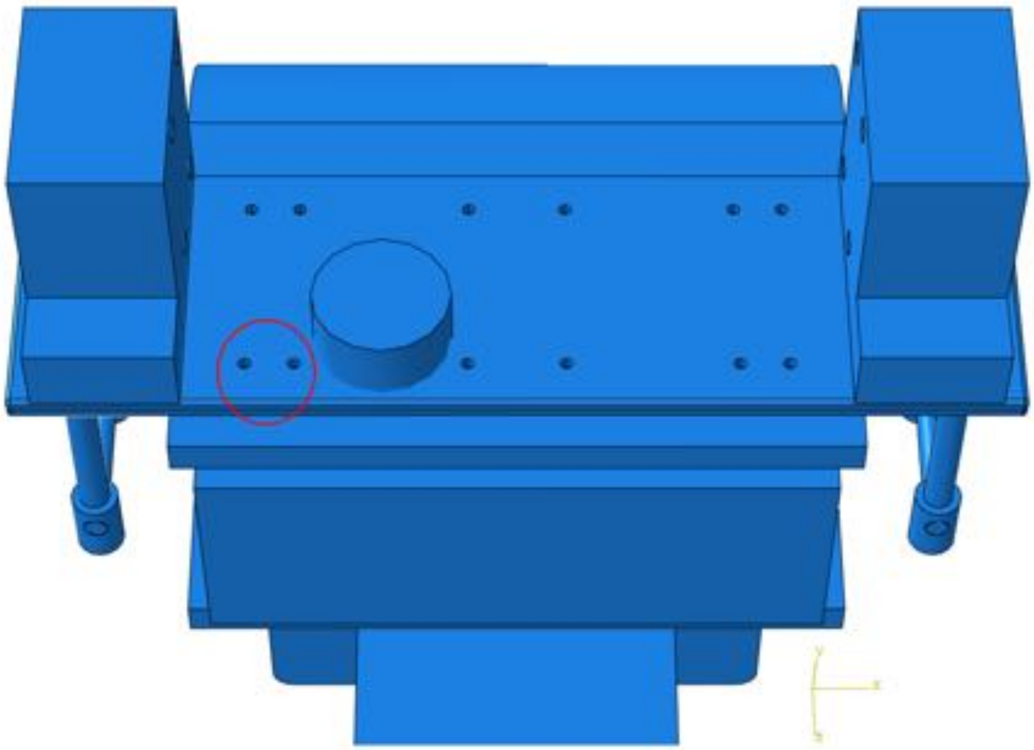

Figure 20: Fasteners for Bolt Holes

Section 6: Analysis of Finite Element Model

This model does not include any dynamic events, so a relatively simple Static (General) step was used. There are two steps, the initial, unloaded one, and a final step with loads. The design criteria for the plate were set at a factor of safety of 2 after 100,000, and a maximum deformation of 0.1 inches at any point in the assembly. The max stress generated from the finite element analysis was 14710 psi. When compared to the yield stress of 54,000 psi, this yields a factor of safety of 3.6. However, this part experiences dynamic loading, causing fatigue. The fatigue limit of steel is assumed to be one half of the yield strength. This causes the factor of safety to drop down to 1.8, which is below the design criteria of 2. This suggests redesigning the plate, making it thicker. The maximum deformation of the plate was 0.0297 inches, which is well below the design limit of 0.1 inches. The graphs below show a characteristic stress and strain along the length of the plate.

Also shown below, the body of the shocks experiences a small stress, which means that the body of the shocks are over designed. Redesigning the shocks so that the body is not as large can save on material costs.

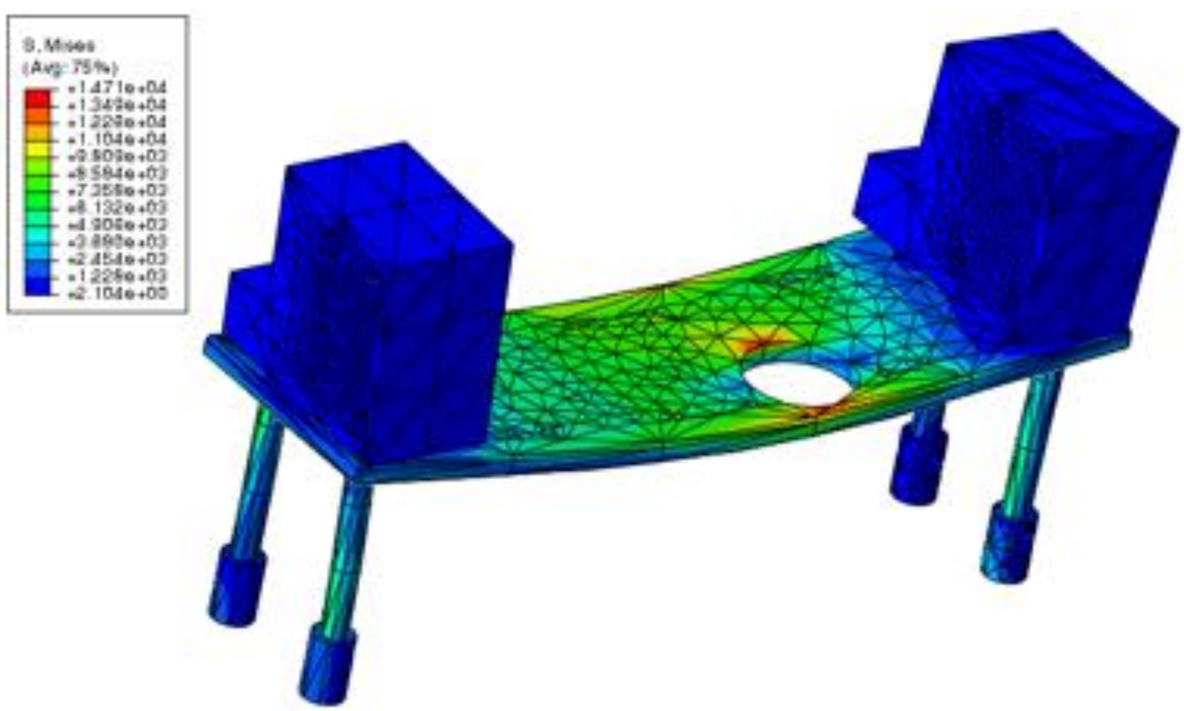

Figure 21: Von Mises Stress of Assembly

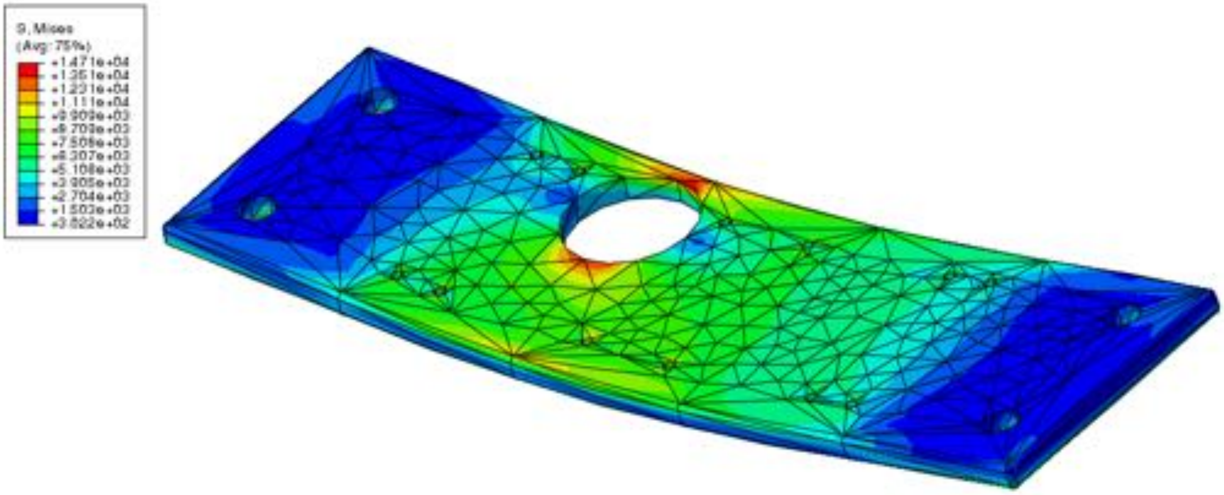

Figure 22: Von Mises Stress Plot of Main Plate

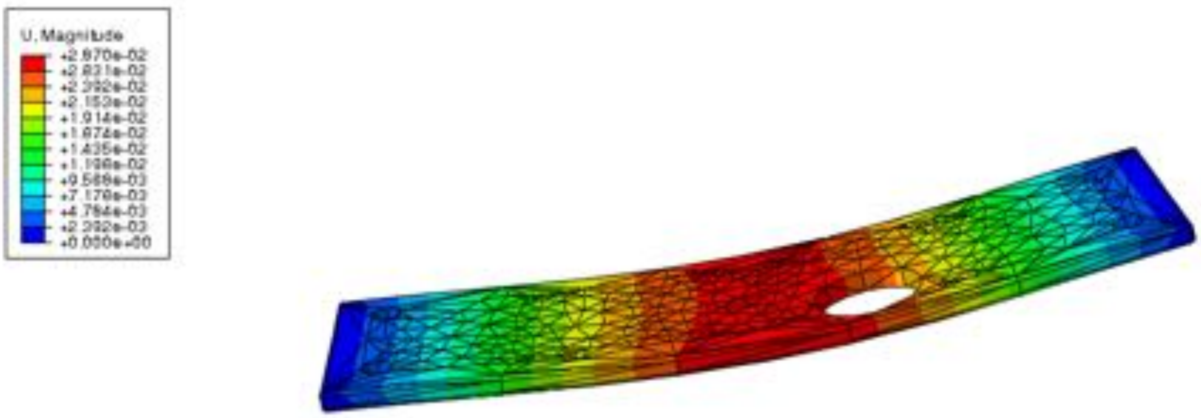

Figure 23: Deformation Plot of Main Plate

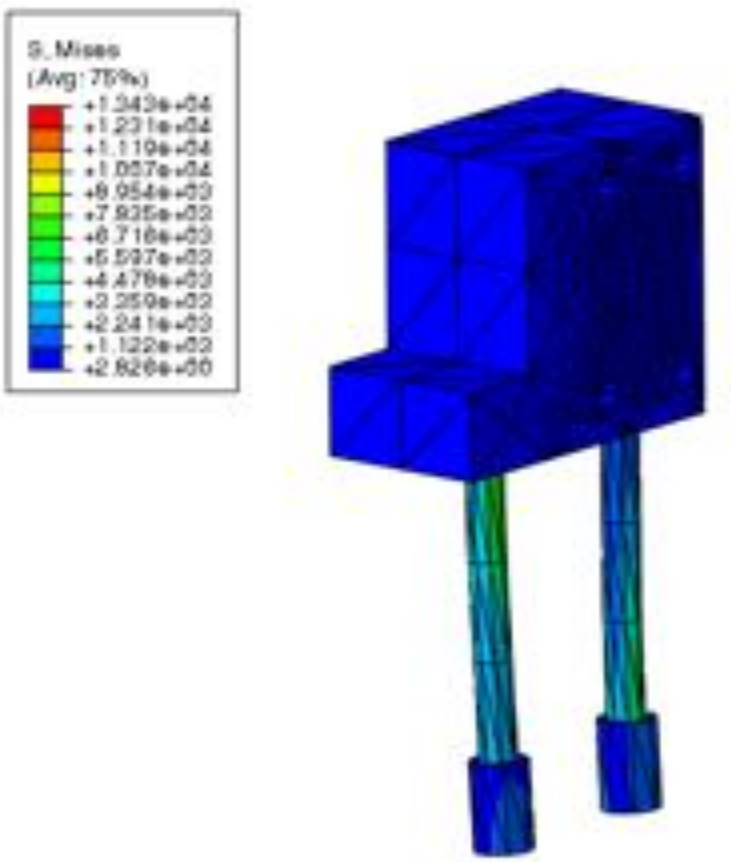

Figure 24: Stress in Shock

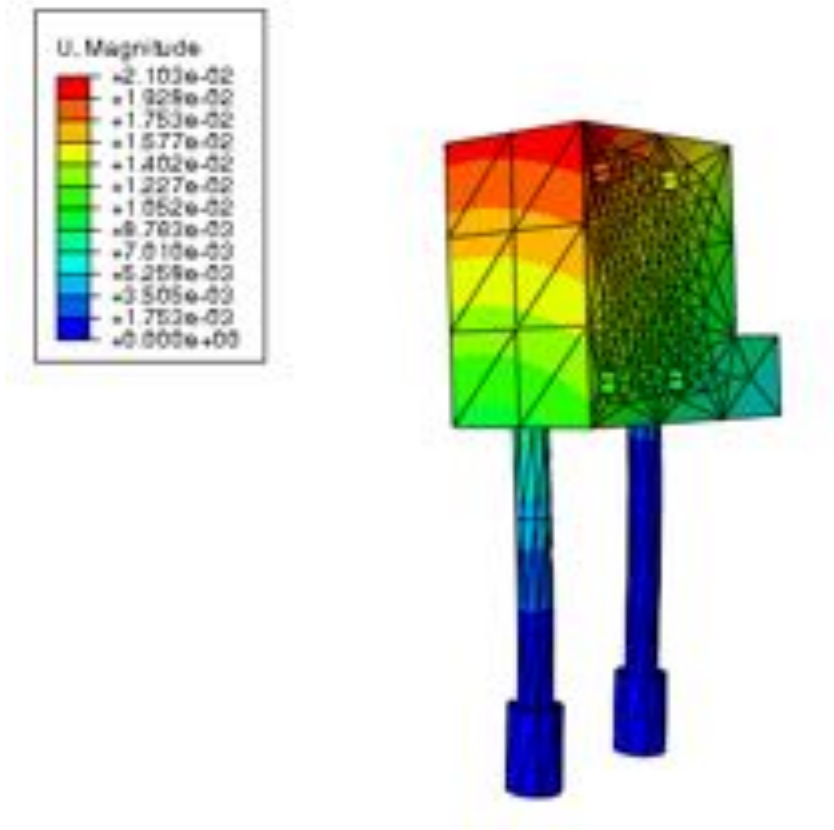

Figure 25: Displacement in Shock

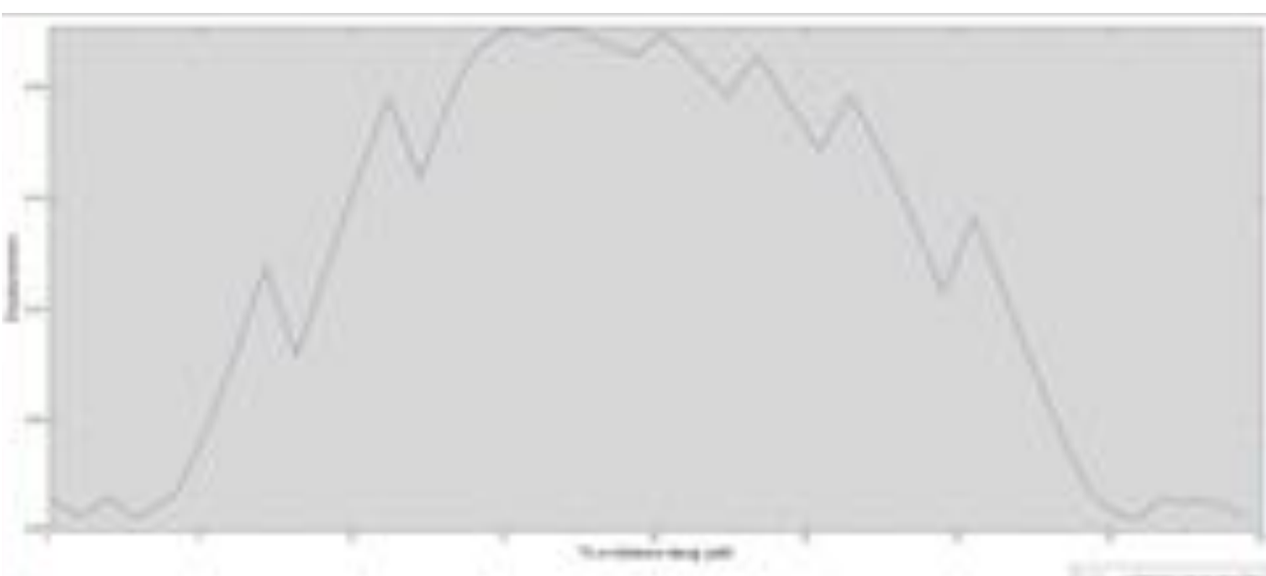

Figure 26: Displacement vs True Distance

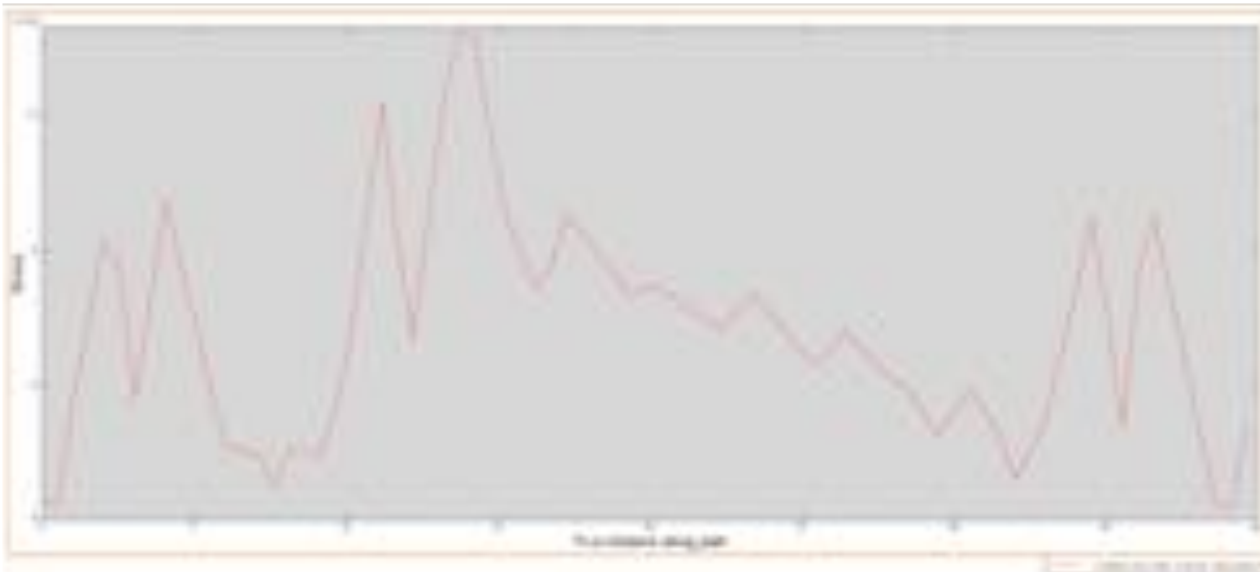

Figure 27: Stress vs True Displacement

Section 7: Summary of Major Findings

In conclusion, the finite element analysis of the gantry crane trolley assembly shows that the assembly could use some design changes. The stresses in the plate did not reach a factor of safety of 2 once fatigue of the part was taken into account. The design criteria for the plate were set at a factor of safety of 2 after 100,000, and a maximum deformation of 0.1 inches at any point in the assembly. The max stress generated from the finite element analysis was 14710 psi. Since the part is in dynamic loading, with forces being applied and then reduced in a cycle, the part fatigues. The fatigue limit of steel is assumed to be one half of the yield strength after 100,000 cycles. This causes the factor of safety to drop down to 1.8, which is below the design criteria of 2. The maximum deformation of the plate was 0.0297 inches, which is well below the design limit of 0.1 inches. To improve the plate, that plate could be made thicker to reduce the maximum stress. Another way of reducing stress is to reduce the stress concentration around the hole where the slider fits into the plate. Fillets could be added, or that hole could be enlarged, both which would reduce the stress concentration.

Also shown below, the body of the shocks experiences a small stress, which means that the body of the shocks is over designed. Redesigning the shocks so that the body is not as large can save on material costs. The legs of the shocks did not deform as much as expected. This shows that the legs are properly designed.

To further improve this project, the team would like to use the original meshes generated. These meshes would give a more accurate representation of how the parts deform. Two areas in particular are around the holes in the plate and the legs of the shocks. These are the areas of greatest concern. Also, including the slider may have made the simulation more accurate. The slider's restriction of both the plate and shock was approximated using boundary conditions, however these are not as accurate as having the slider in the assembly.

In conclusion, this finite element analysis shows that the plate has a factor of safety of 1.8, which is lower that the desired lower limit of 2. The team recommends that the capstone group improve the plate for safety.

Section 8: Works Cited

Capstone Design Team: Jay Hauseman, Vivek Patel, Dan Andrus, and David Wasi