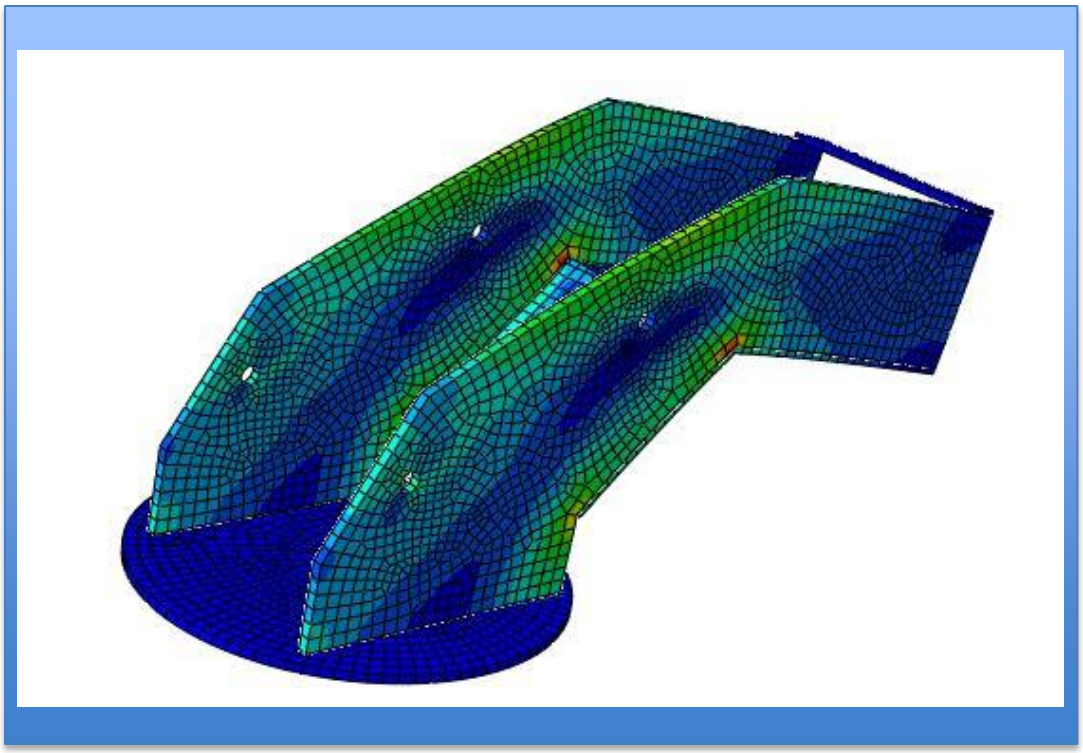


ME 461: Finite Element Analysis

Fall | 2015

A Semester Report on:

The Development and Analysis of Failure Criteria of a Crane Base

Group Members:

Jasper Van Der Sluys, Benjamin Zurilla, Kevin Wilhelm, Joe Murphy

PennState College of Engineering

Department of Mechanical and Nuclear Engineering, University Park, PA

Table of Contents:

Executive Summary

The following project involves the structural analysis of a crane's boom and base under normal and extreme operating conditions for the purpose of evaluating failure conditions. The CAD geometry and crane design were made to emulate that of an actual companies, of which team member Benjamin Zurilla has had recent working experience with. An iterative analysis process of the crane's central structure was done to optimize both the analytic approach as well as the finite element analysis. After creating the CAD geometry, mesh types were compared for performance and assessed for accuracy of the simulation. In this analysis Hexahedral meshes were chosen over Tetrahedral given how significant deformation in some areas of the body are. Upon running the analysis, the results were close enough to expected estimated values to confirm that our mesh and boundary conditions were applicable to describe this problem.

Extreme-operating boundary conditions were developed to understand the failure criteria.This includes application of heavy loads and high wind conditions acting on wide-loads. In order to simulate these occurrences the loading conditions were changed in magnitude to an estimated value of said extreme loads. The mesh was refined in areas where stress and deformation were highest, and the interaction between meshed parts was changed from simple face ties to node-to-node connection to more accurately describe welded features.

In order to determine if the parts actually failed at any point in a dynamic simulation a plastic interaction was added. This allows for the simulation to determine the stress and strain at any point in the application of the extreme loads. The only difference in the creation of this plastic simulation is the change in the interaction conditions. This is further explained in the creation of boundary conditions section of this report.

List of Figures

Section 1: Background and Project Plan

In order to prevent a Grove RT 530 crane from tipping a counterweight is placed opposite the reach of the boom.

Fig 1 – Counterweight Mounting and Placement

As seen in the above Fig 1, the counterweight, the big box on the back of the crane, is cantilevered and in some cases can weigh 50 tons or more. If incorrect dimensions are used then the crane will fail.The purpose of this report is to analyze the superstructure, which includes the attachment point for the boom and counterweight, for failure during operation and under extreme loads. This is critical to the operation of the crane in order to maintain a long operating life and insure that the safety of the public is maintained.

General Plan:

1. Determine how complex the analysis will be and decide on what section of the assembly should be chosen to be analyze.

2. Model the part manually as well as possible or potentially get a model from a crane company that already has a model prepared.

3. Set up a model for analysis in Abaqus.

4. Run the analysis.

5. Check results.

6. Determine if they are reasonable and recommend improvements to the design if any are needed.

Section 2: Development and Description of the CAD Geometry

Our cad geometry was developed with the background knowledge of the size and shape of a crane superstructure. The superstructure is the term for the frame of the weight bearing part of the crane. This can be seen in the figure below.

Fig 2 - Crane Superstructure

This part of the crane carries all the forces of the load, boom, and the counterweight that you can see in the picture. It is a critical part of the crane, and we want to see if we can improve upon the design in some way.

Next, we began to model simple parts in creo parametric. Once we had accurate but simplified representations of our geometry, we assembled them in their respective orientations. The file was then exported as a step file so that we could transfer it into Abaqus.

Section 3: Development of Finite Element Meshes

In order to create the mesh for the parts make sure that the active module is meshed.

Steps:

1) Create a material is has been made for each part and has the material properties of density = 7861 kg/m^3, E = 200 E+9 Pa, and poisson's ratio = 0.26. This will be used for all the parts and will be needed to apply to each of the parts shown in Fig 3a &3b.
    a. In order to model the reference state only these values are need in the material.
    b. In order to model the plastic deformation a plastic property needs to be added using the numbers seen in Fig 3b.

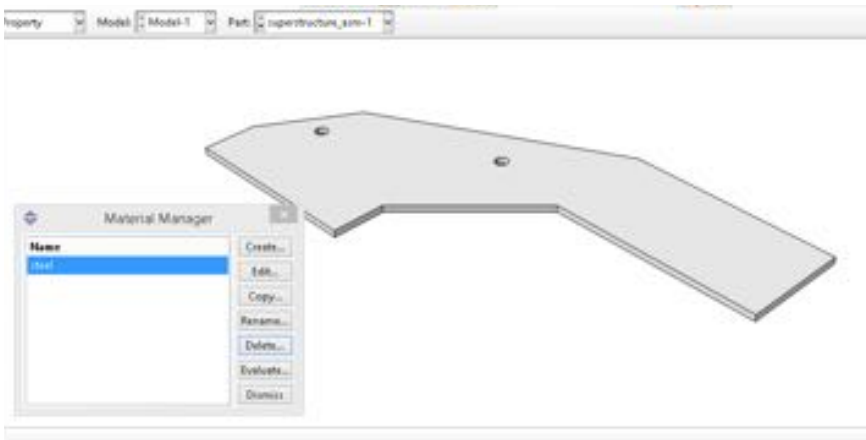


Fig 3a - Creation of the Side Part ref.

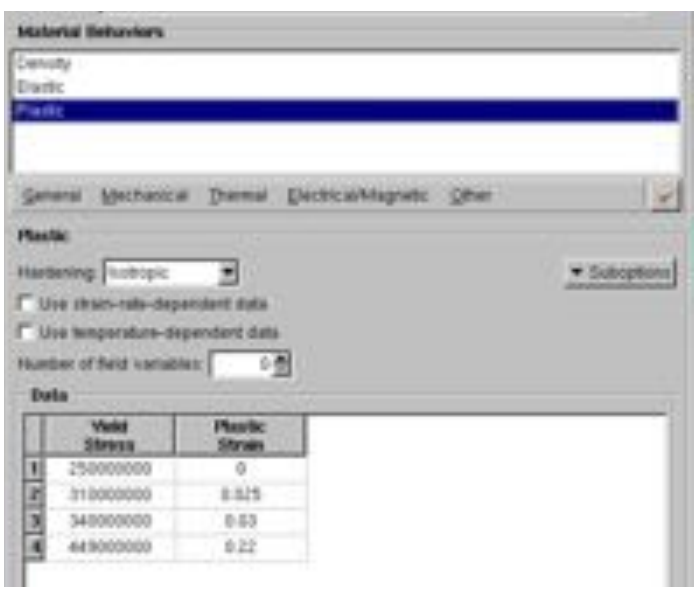


Fig 3b - Creation of the Side Part plastic.

2) Create a solid homogeneous section for each part while still in the property module (see Fig 4). If there is a section present when the section manager button is selected move on to step three.

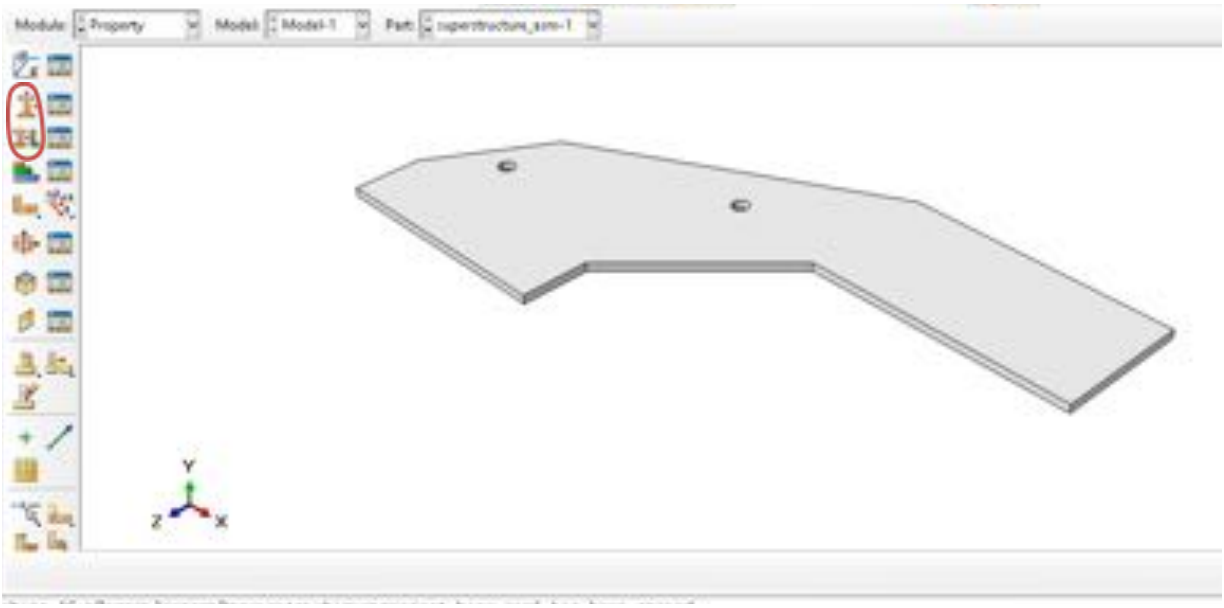


Fig 4 - Creation of a Section.

3) Create a dependant assembly within the assembly module using the create instance button as shown in Fig 5.

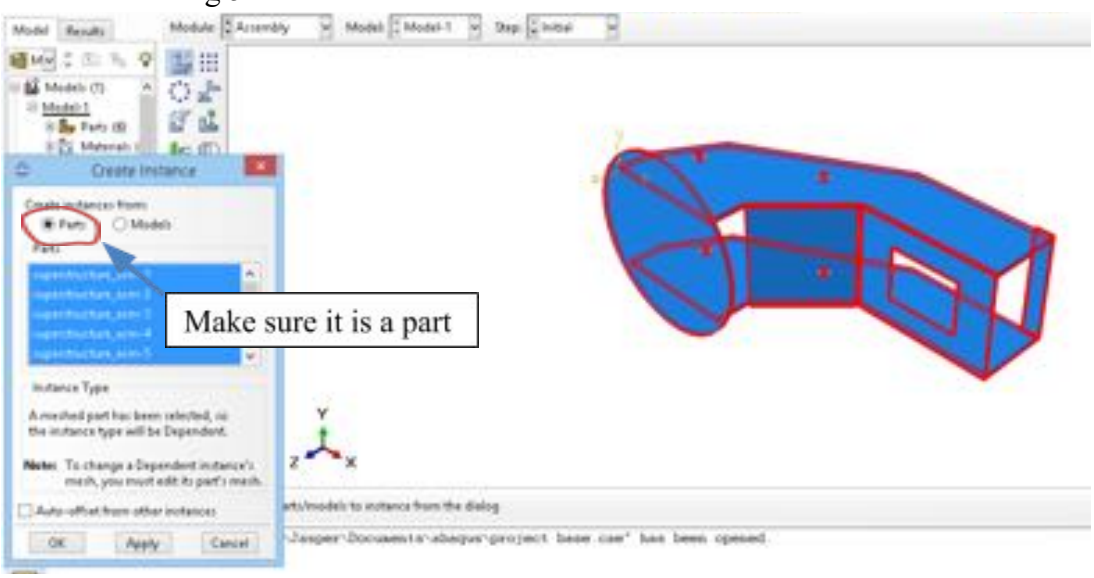


Fig 5 - Assembly Creation.

4) Seed the mesh at a global seed of all parts except the two side plate in order to prevent errors resulting from mesh discontinuities.

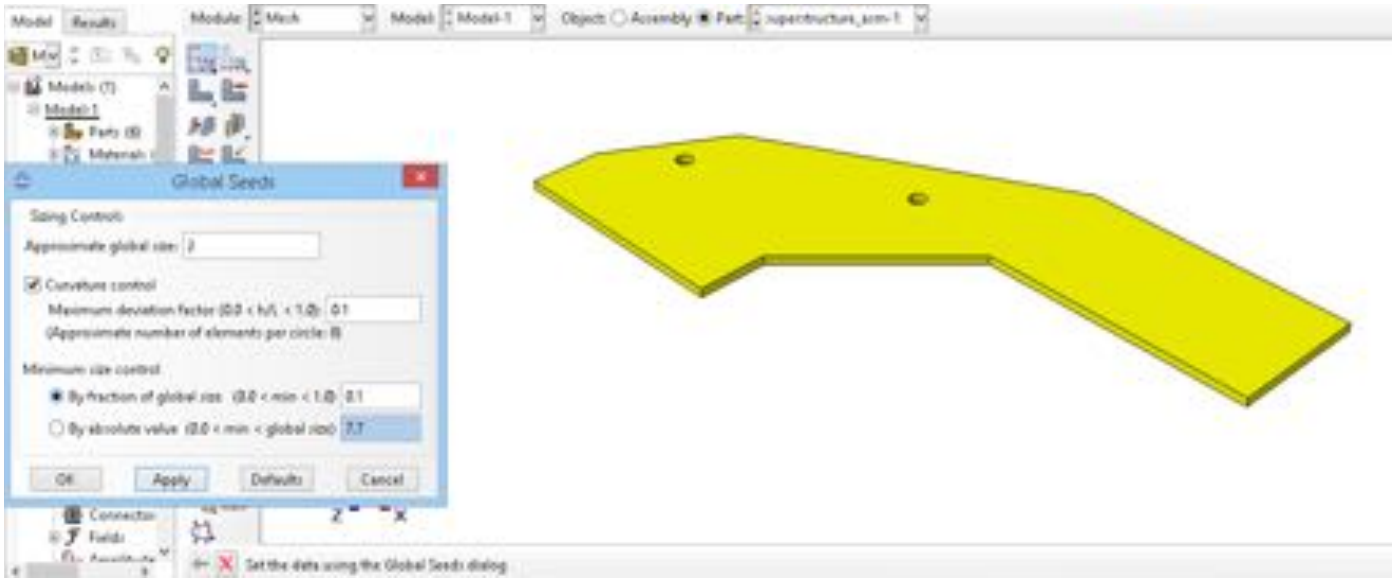


Fig 6 - Mesh seeding.

5) Assign an element type and choose **HEX.**

6) Mesh the part and visually check the density of nodes that appear on the edge of the part. If the mesh looks to dense it probably is and should be changed.

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

In order to develop the assembly and apply the boundary conditions the application of the model needs to be considered logically. Therefore, the boundary conditions were applied to the bottom of the base of the model to keep it from moving. While the traction force that results from the counter-weight and reaction forces due to the pins are applied at the end of the superstructure and the holes. The full assembly can be seen in Fig 7.

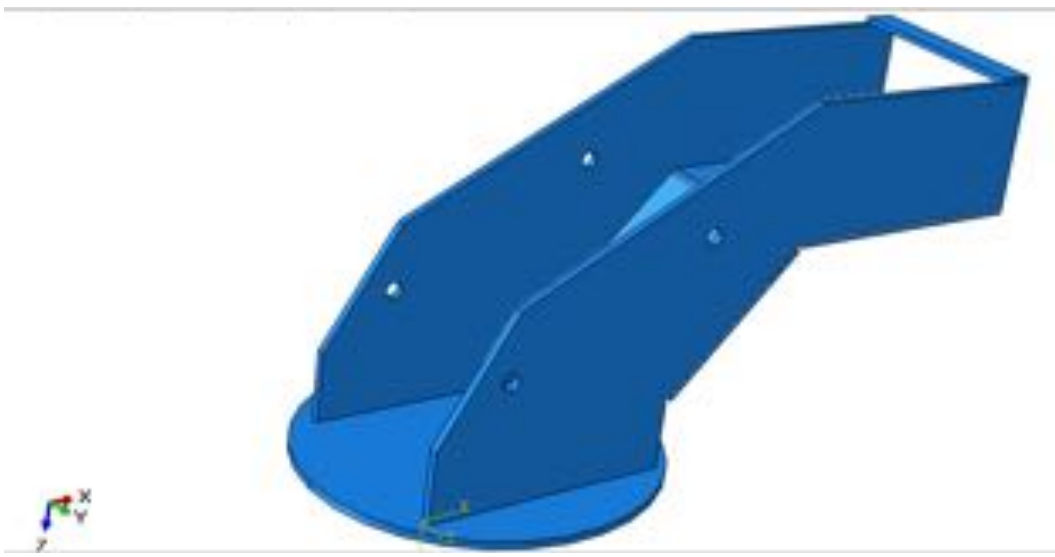


Fig 7 - Crane Superstructure.

In order to mimic the behavior of a weld boundary conditions were applied to each interaction surface. The boundary condition type that was chosen was a TIE boundary condition this means that the boundary nodes do not move in relation to the other instance it is bounded to when the mesh deforms. This is explained further in section 5.

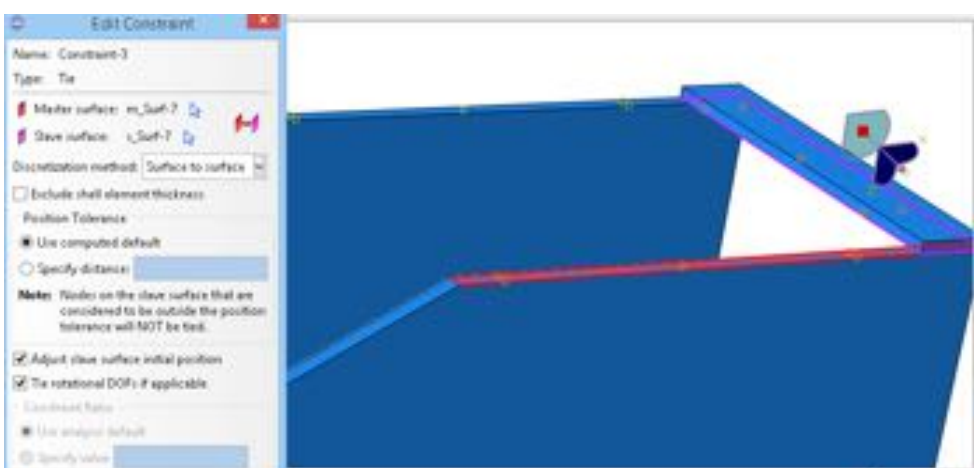


Fig 8 - TIE Boundaries.

However the model will not run properly if there are no boundary conditions applied to the model that will prevent movement or even create movement. Therefore, the base part had boundary conditions applied to the bottom surface so that all degrees of freedom (dof) were constrained to zero. In order to get the load force a traction boundary condition was applied to the ends of the two side plates, as well as the bottom and top of the lower and upper holes respectively. Both the boundary condition to the base plate, the traction force from the weight, and the traction force from the pins can be seen in figures 9, 10, and 11 respectively. The coordinate system needed to be moved in order to properly apply the pin forces that result from a boom that is fully extended at 45°.

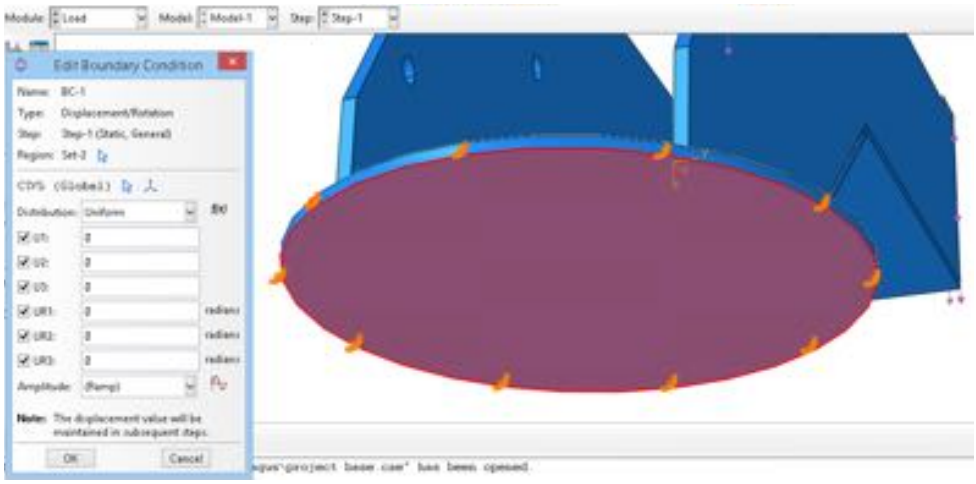


Fig 9 - Boundary Conditions 1.

Fig 10 - Boundary Conditions 2.

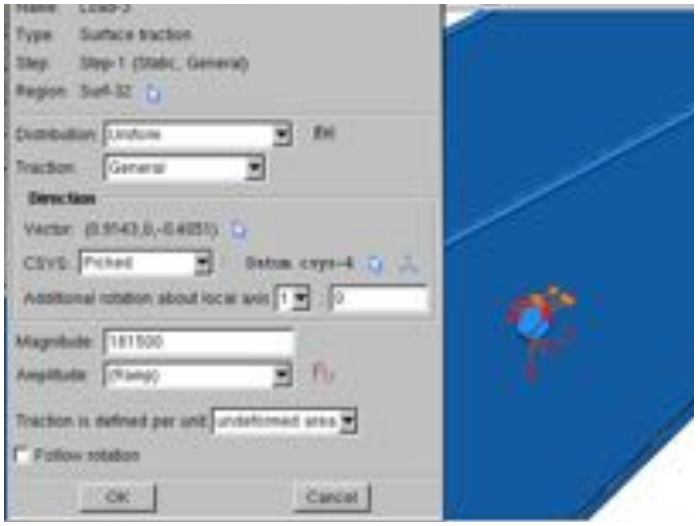


Fig 11 - Boundary Conditions 3

Section 5: Development and Description of Model Interactions

Goals for Model Interactions:

- The model must mimic welding between each part
- The model must have a fixed base

Description of Interactions

In order to mimic the effect that welding has on the structure of the crane several options are available. The first way to apply the welding interaction would be to merge the parts allowing the model to treat all parts as if it were one object. This means that at the interface there would be no empty space, but instead a mesh that had the material properties that were defined for the part instances. The second way to mimic a wield would be to apply a boundary condition to the two contact surfaces such that there would be a function that would drive the boundary condition. The function would be defined so that it would force the boundary condition to mimic a weld, which is a weaker combination of the material that makes up the two parts in question, as the model ran. The last method of interaction would be to use a TIE interaction to constrain the nodes on the surface of each part. This will cause the distance between the two nodes on the two separate parts to maintain the same distance from each other. This allows for meshes to be made for each part independent of each other and is the method that was chosen for this project.

Application of Interactions:

The TIE interaction was applied to each surface that was in contact with another part using the TIE operation that is available through the create constraint button as can be seen in Fig 12. In order for this model to work an instance needed to be applied to each part separately. The reason this was chosen is explained further in section 7, but generally the TIE interaction was chosen due to the meshing of individual instances.

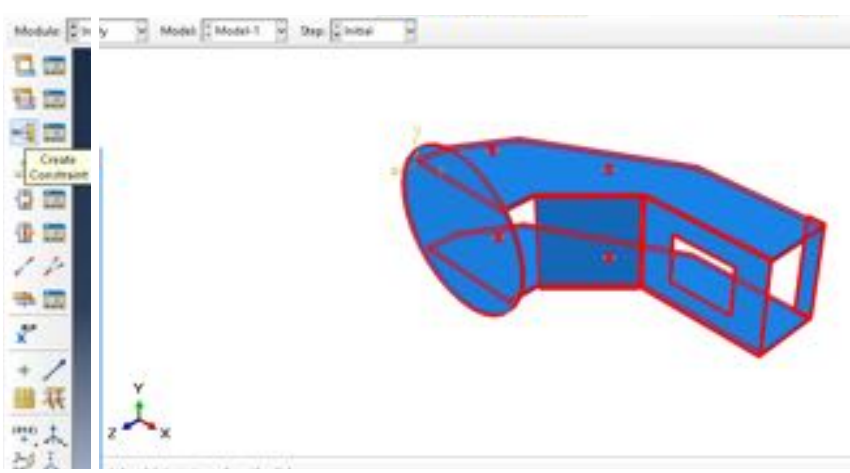


Fig 12 - Creation of a TIE Constraint.

Section 6: Analysis of Finite Element Model

Summary of variables:

The model was run at 100 steps as a static model with an increment size of 1 with a maximum of 1 and a minimum of 1 x 10^{-5}. After the simulation was started it ran for about 2 to 5 minutes. This is probably due to the fact that there are so many nodes and the fact that there are six parts. Both the number of nodes and the number of parts require a lot of memory when used in a simulation. The fact that this was also a 3-D simulation also could have affected the fact that the simulation ran at a slower pace.

The mesh was varied between each part but stayed within the range of a mesh seed of 48-85. For the side parts and the square with a hole in it tetrahedral meshes were used. This was chosen due to the geometry and the predicted outcome of the strains. In order to correctly simulate the stress in this section of the model a Hexahedral mesh was chosen to be used and a mesh refinement was performed around the hole.

After the simulation ran alterations were made to the boundary conditions and mesh in order to make a more accurate model and to compare different results due to said changes. When no alterations were needed for a specific magnitude of loading and specific type of interaction conditions analysis of the model could be done. The findings can be found in the results section of this report.

Analysis:

When the part is meshed, due to it's unique geometry, several problems can occur. For example, when a geometry has a hole it is difficult to see the stress that radiates through the piece at the corners due to a stress concentration. If a tetrahedral mesh is used, rather than a hexahedral mesh refined at the hole, a more accurate stress strain relationship is likely to result from the model. This is because of the fact that there are nodes in spots where a square mesh would not place them. If a tetrahedral mesh is chosen though there will be a reduction in quality and accuracy of the results for the corner on the side plate where the two square parts are meet.

Since the superstructure is not expected to fail at the welds it is not very important that they have a high mesh density where the welds are. However, due to the fact that interaction conditions are TIE boundary conditions, the fact that welds still have elastic properties is completely neglected. In respects to completely modeling every aspect of the part this is not a good model. Abaqus does have the ability to model a boundary condition as a function of some variables. If such a boundary condition were applied not only would it take up more processing time but it would be unnecessary for the purpose of this report. The reason that it is unnecessary is because of the fact that the TIE condition only constrains the surface node of a mesh on one part to the surface node of a mesh on another part. The mesh one layer in can therefore accurately model the weld's elastic properties.

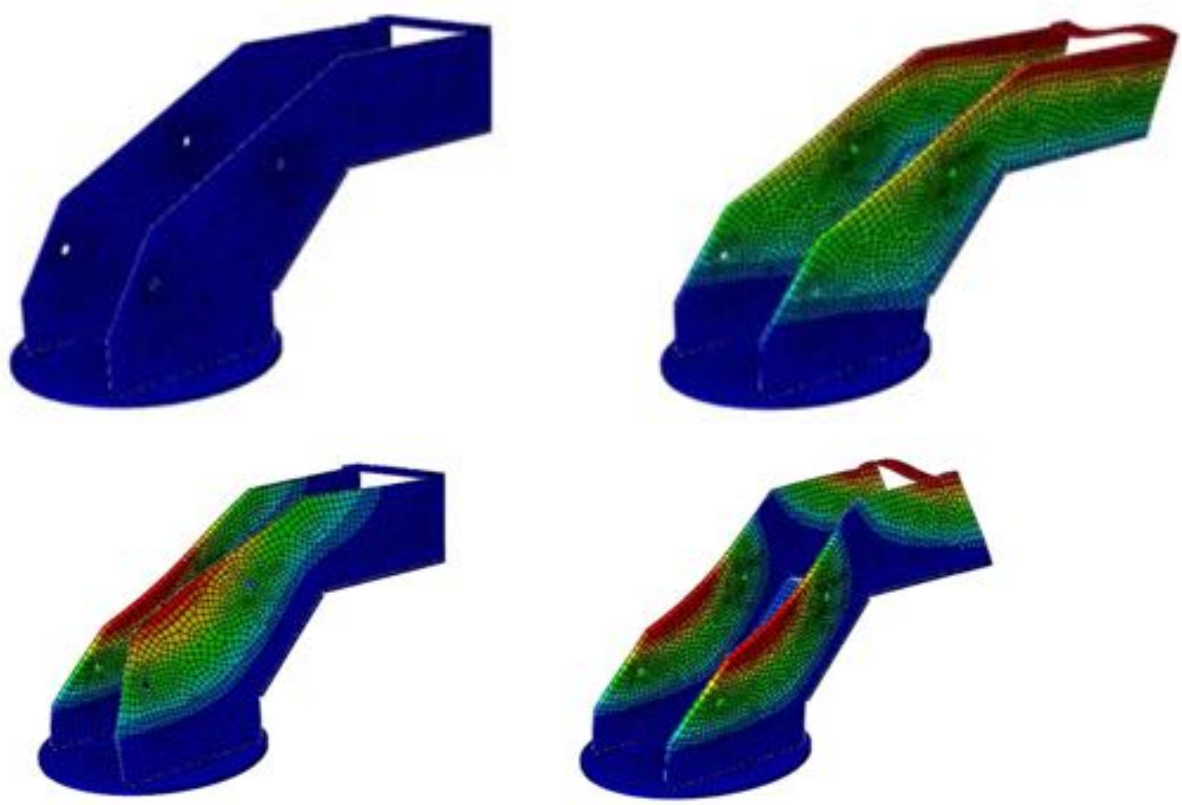
Overall the model, while not providing the finest detail of the simulation, is sufficient enough for the purpose of determining failure. With the mesh alterations around the hole the accuracy is increased in that region. With the uses of a medium mesh seed the quality of the mesh is good. Finally due to the use of a TIE interface the modeling behavior of weld is maintained even if the result is not detailed in the regions where they exist.

Section 7: Summary of Major Findings and Conclusions

The purpose of this analysis was to determine whether or not the crane's base would fail under certain loading conditions. Therefore, the failure criteria is met whenever any stress exceeds the yield stress for steel of which the part is made of. The maximum stress occurred on the side plates where the square plate that has a hole in it and the other square plate meet. This is due to the stress concentration caused by the corner, where the two plates meet, and the moment caused by the counter weight for lower loads. When higher loads were applied the maximum stress location switch from the corner to the pin holes. The results change when the model mesh and constraint types are altered. If the constraints were changed to have a surface to surface interaction, minor changes occurred than compared to a node to node interaction. The analysis is non- symmetric due to the meshing distribution across the base. When plastic modeling occurs for 5 seconds under both normal and extreme loads the part will not fail under a reasonable load.

Modal Analysis:

A modal analysis was also set up and run using three eigenvalues. The resulting natural frequencies that we saw in our results were between .1 and .2 cycles per second which can be considered very reasonable for such a stiff and resilient part. These frequencies should be avoided in the any cyclical loading interactions with other parts. Alone they are interesting but the entire crane would have to be analyzed to see what the interactions would result in and how the natural frequencies would then change. The modes can be seen in figure 17 below and they are greatly exaggerated to show clearly where the part would deform.



0	Increment	0: Base State		
1	Mode	1: Value = 0.44469	Freq = 0.10613	(cycles/time)
2	Mode	2: Value = 0.98593	Freq = 0.15803	(cycles/time)
3	Mode	3: Value = 1.2284	Freq = 0.17640	(cycles/time)

Fig 13 - Modal Results

General Static Results:

The resulting stresses of interest for a load of 51796.8 N are presented in *Table 1 - Result Stresses1*. In order the locations that are mentioned are for the holes closest to the base part (location 1), the holes closest to the top of the part (Location 2), the corner where the two square plates meet, and the intersection of the two side plates and the base plate.

Location of Interest	Maximum Von mises Stress (N/m) Ref	Maximum Von mises Stress (N/m) Plastic
Location 1	5.238*10^5	4.109*10^5
Location 2	5.238*10^5	4.109*10^5
Location 3	1.571*10^6	1.641*10^6
Location 4	2.612*10^5	5.476*10^5

Table 1 - Result Stresses1

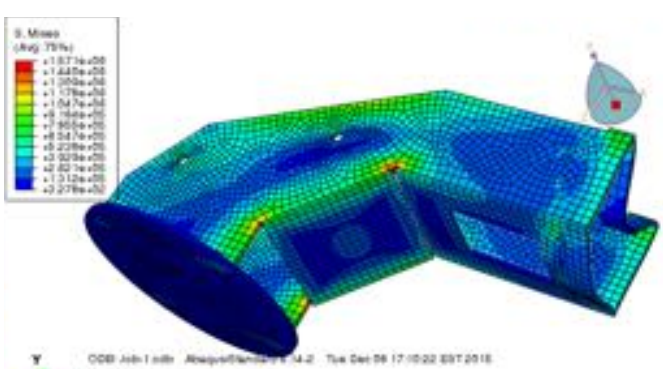


Fig 14 - Deformed State of Crane Ref

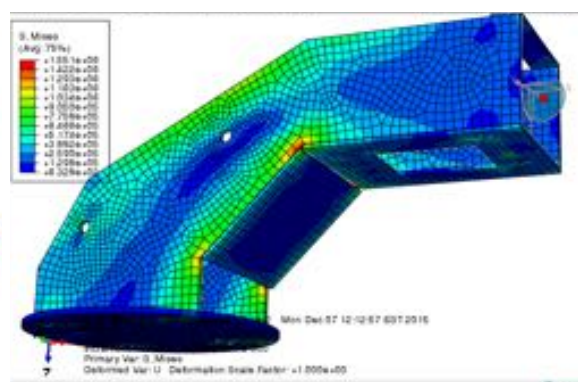


Fig 15 - Deformed State of Crane Plastic

As is shown both in the table and *Fig 13 - Deformed State of Crane* the highest stress is on the side plates and radiates out a short distance from the corner where the two square plates meet. Because the mesh density is only moderate at the point of interest and the material is so stiff the distance the stress radiates out to is most likely correct. However, the accuracy for the overall shape may be wrong due to the size of the element placed used. Due to the fact that the base part has a less uniform mesh on one side there is more stress on one side part and on one side of the base plate as seen in Fig 15. This stress does not exceed the yield stress of steel which is 250 MPa or 3600 psi.

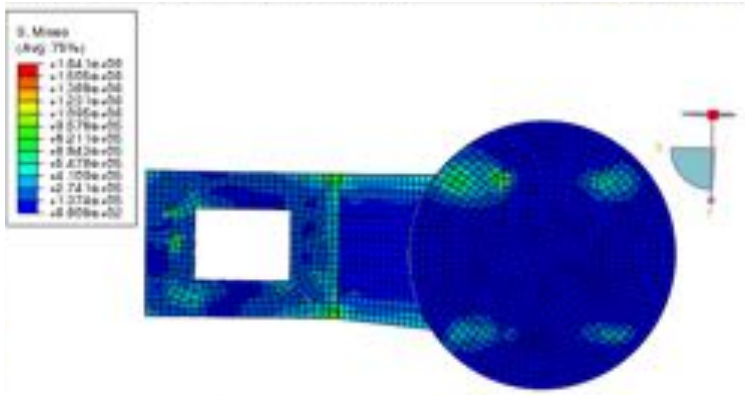
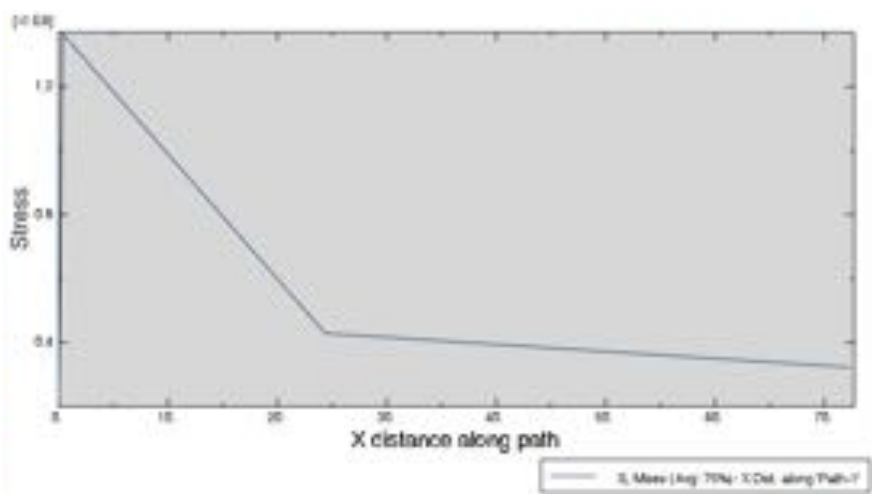


Fig 16 - Non-symmetric stress

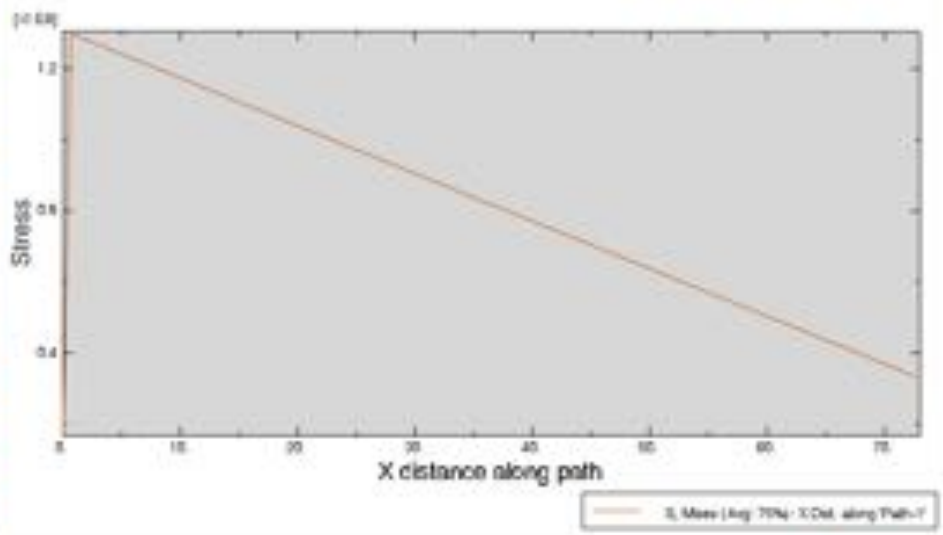
Plastic and Extreme Loads for Base and Plastic Simulation:

When the part is considered to be under it's extreme load it's stress reaches . This means that the part (will still not/ will) fail. When determining the component of the model that would cause the failure it was noted that the counter weight can be increased to unreasonable values such as 500000N without causing failure. With the weight fixed at 3744.8 N with Loads applied at 51796.8 N, 98100 N, and 1*10^8 N to view the full effect of when the part will fail. It is determined that there will be no failure at a reasonable load and focus was shifted to just the application of the 51796.8 N load. However, as the load increase the stress at the corner on the side plates that meets with the bottom corner of the full square plate also increases until the maximum stress begins to occur at the pin holes.

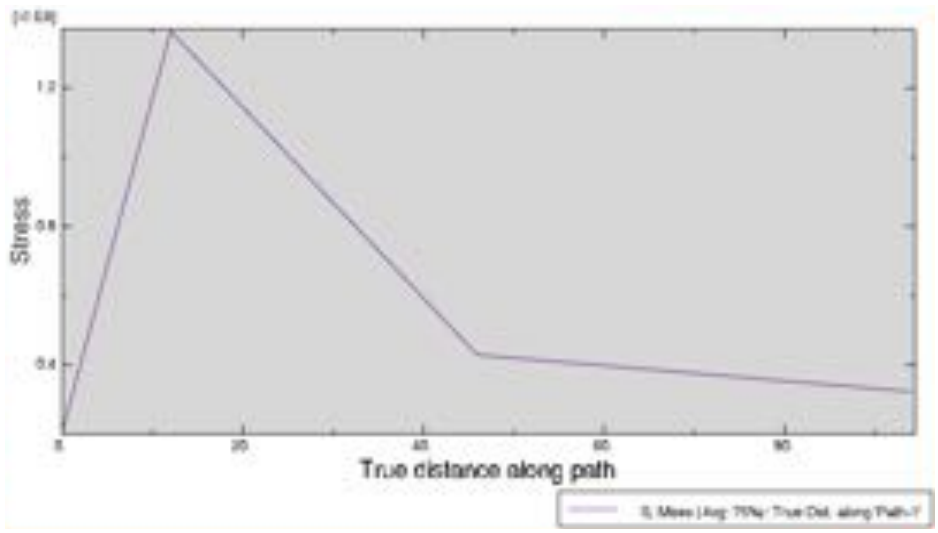
Compared to the results of the reference model the plastic simulation as can be seen when comparing plots 1 & 2 to 3 & 4.



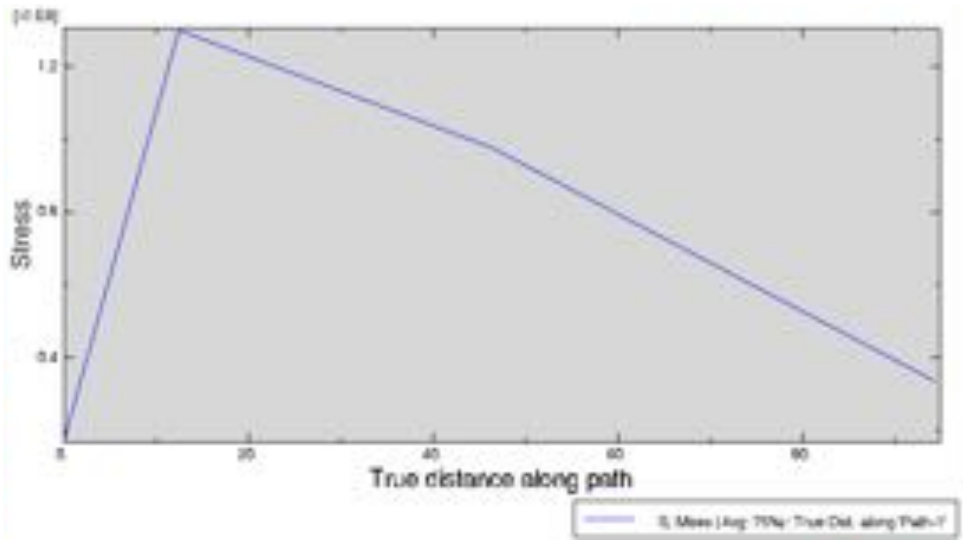
Plot 1 - Von Mises Stress vs. x Direction Ref



Plot 2 - Von Mises Stress vs. x Direction Plastic



Plot 3 - Von Mises stress vs. Distance Along the Path Ref



Plot 4 - Von Mises Stress vs. Distance Along the Path Plastic

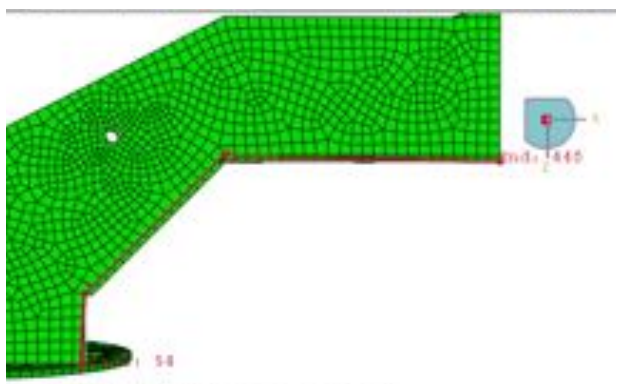


Fig 17 - Path for xy Plots

As can seen in a comparison between Fig 14 & 15, plots 1,2 & 3,4, and Table 1, not only does the plastic simulation result in higher maximum stress but less stress distribution over the part. Because of the fact that the stress vs. strain relationship is included in the material properties the simulation could logically be said to be more accurate. If the static general case is produces a stress at the end where the counterweight is applied which was unexpected. Again the fact the material property is not included in the general static case can account for the fact that there is more stress near the counter weight.

The only problem with this model is the fact that the stress concentration might be off due to the type and size of the mesh that was chosen. This skewed the results but left a good representation of whether or not the part will fail. This error cause by the mesh in a sense makes a safety factor inherent in the model. Overall the model is well represents loading conditions accurately while allowing showing that there is little variation between modeling techniques.

Section 8: Works Cited

ABAQUS (6.14) `ABAQUS Documentation', Dassault Systèmes, Providence, RI, USA.
Manitowoc Crane Group, Shady Grove PA
Manitowoc. *Grove RT530E-2 Product Guide*. 2012. Specifications and description of parts of a Grove RT530E-2 crane. Wisconsin.