OPTIMIZATION OF FRONT LOWER ARM

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

____

Derek Silsby

____

Francis Song

With partners
Moon-Ki Bae (T.A.)
Myung-Su Chae (T.A.)
Ha-Sik Jung
Xin Sun
Of Kyungpook National University in Daegu, South Korea

Date: August 5th, 2008

Approved By: _________________________
Professor Yiming Rong

Approved By: _________________________
Professor Zhikun Hou

Professor Young-Suk Kim (KNU)
Professor Choon-Young Lee (KNU)

100 Institute Road Worcester, MA 01609

Abstract

Our team was given the task of optimizing the Front Lower Arm of an automobile by working with students and teaching assistants from Kyungpook National University. In particular we focused on conditions where sudden acceleration or braking can cause heavy stress to affect certain areas.

By following Design of Experiments our team came up with nine possible case models to present to Hwashin. We utilized ABAQUS software to run a Finite Element Analysis for each case, of which we selected the best two examples that met Hwashin's criteria of minimizing stress while being lightweight.

Acknowledgements

Our team would like the opportunity to thank everyone at Worcester Polytechnic Institute, Kyungpook National University, and Hwashin Company, Ltd., who made this project possible. Professors Rong, Lee, and Kim all worked tirelessly to make the dream of a joint project between WPI and KNU a reality, and one that will hopefully continue well into the future.

A great deal of thanks goes to Myung-Su Chae and Moon-Ki Bae, without whose help, this project would not have succeeded. Even when they had their own projects to worry about, both T.A.s still took the time to help us with whatever trouble we encountered.

We would also like to thank everyone at the CAE & CAD Lab for providing us with a safe, efficient, and effective environment for us to work in and helping us whenever possible.

Finally, thanks to our partners Ha-Sik and Xin for their cooperation and assistance in helping us complete the project, even though they had their own projects and summer school courses to worry about.

TABLE OF CONTENTS

Table of Figures

Table of Tables

Table of Equations

1. Introduction

The automotive industry in South Korea is a very dynamic environment and currently one of the fastest growing around the world (Frost and Sullivan, 2005). Hwashin Company, Ltd., is an automotive parts supplier to Hyundai that makes small components specializing in chassis and body parts. Their lines of products include the Front Lower Arm, engine oil pan, side member, and pedal system.

Our team was tasked with the responsibility of optimizing the performance of the Front Lower Arm. The part, which is shown in detail in ***Fig. 1***, is responsible for minimization of vibrations encountered by a vehicle as it travels along the road. It encounters the highest levels of stress when a vehicle suddenly brakes or accelerates, particularly in the Critical Area (in red) as shown in ***Fig. 1***.

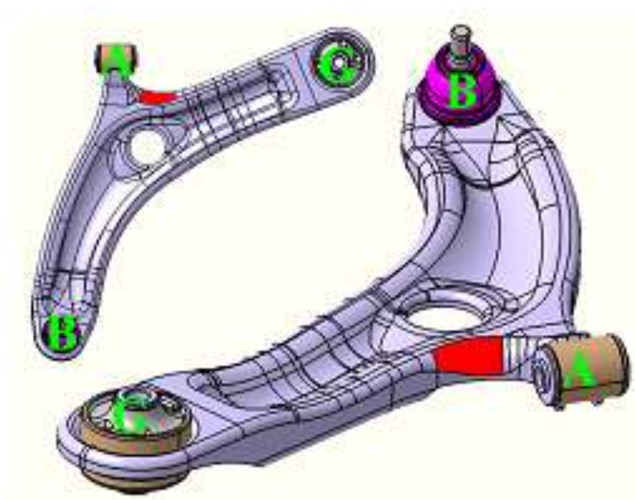

Figure 1: The Front Lower Arm and its sections.

Hwashin specified certain areas for our team to focus on, specifically the Critical Area, when completing our stress analysis. For the initial loading and boundary conditions, Hwashin specified force vectors through the location of certain points (A, B, G) on the Front Lower Arm itself. These locations are also shown in ***Figure 1***.

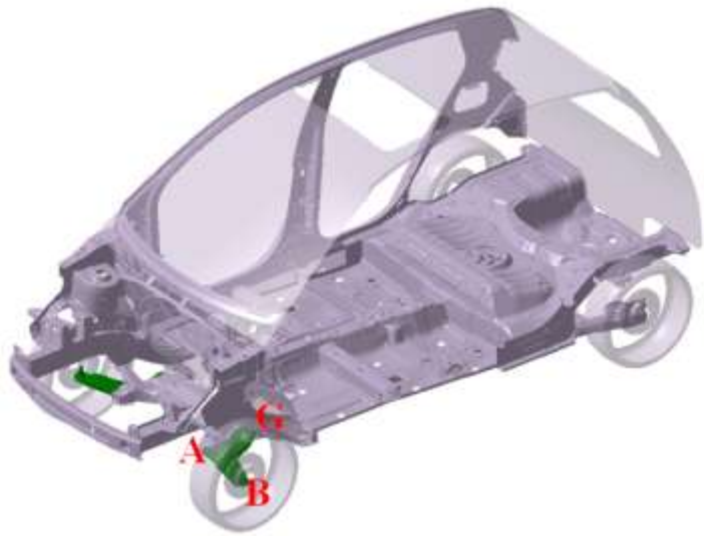

Figure 2: The location of the Front Lower Arm.

Our sponsors at Hwashin were Byung-Cheol Park (Design Team Manager) and Kee-June Mune (CAE Team Manager). Mr. Park and Mr. June were both were extremely helpful in conveying the goals of the project and updating us of any changes that were made. The actual working environment was different than that of most WPI off-campus projects in that our team was able to work at the CAE & CAD Lab at KNU where the two Teaching Assistants worked, as opposed to working at our sponsor's office. We even had two computers available that were dedicated to the project. However, we were able to visit Hwashin twice and on both occasions we met with Mr. Park and Mr. Mune.

2. Background

In order to develop a better understanding of the challenge our team was tasked with, our team needed to quickly adapt to the standards and practices that were employed by both KNU and Hwashin. Such challenges included using the metric SI system and learning new programs. Our team had to rapidly orient and familiarize ourselves with important characteristics of the project such as surface modeling of a Computer Aided Design (CAD) file, performing a Finite Element Analysis through a computer platform, and creating a Design of Experiments Table based on the Taguchi Method.

The following sections will cover a brief history of Hwashin and introduce Design of Experiments, solid modeling through CAD, and the Finite Element Method.

2.1. Hwashin Company, Limited

Hwashin Company, Limited, was founded in October of 1969. Based in Youngchun, South Korea, Hwashin has set up branch offices and plants all over South Korea, mostly in Gyeongbuk Province. Hwashin's current president is Ho Chung and honorary president is Jaehyung Chung. Its vision for 2008 and beyond is to improve and expand its capabilities by adding on to its over 30 years of experience in making automobile parts trusted by the major automobile companies while provide a safe and secure working environment for its employees.

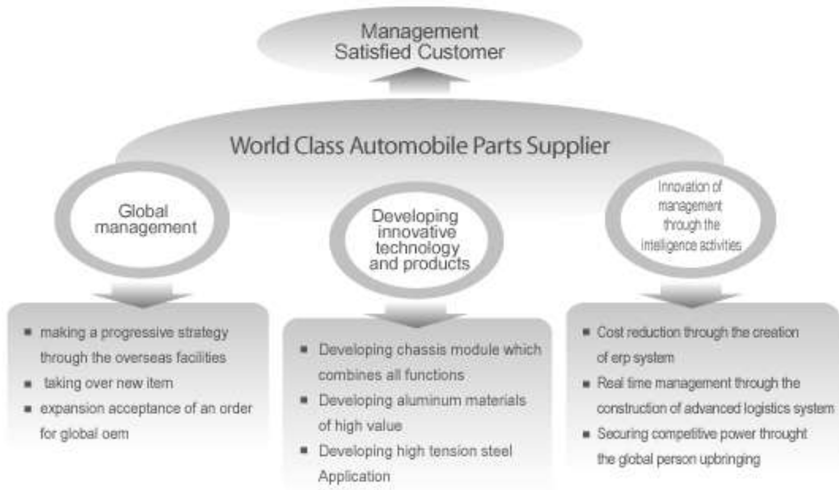

Figure 3: Hwashin's goals for 2008 and beyond

Having its own Research and Development Center, Hwashin is committed to developing high-performance, ecologically-friendly technologies for use in future automobiles. It has also maintained an international presence, with branch offices located in Chenai, India, Beijing, China, and Greenville, Alabama, US.

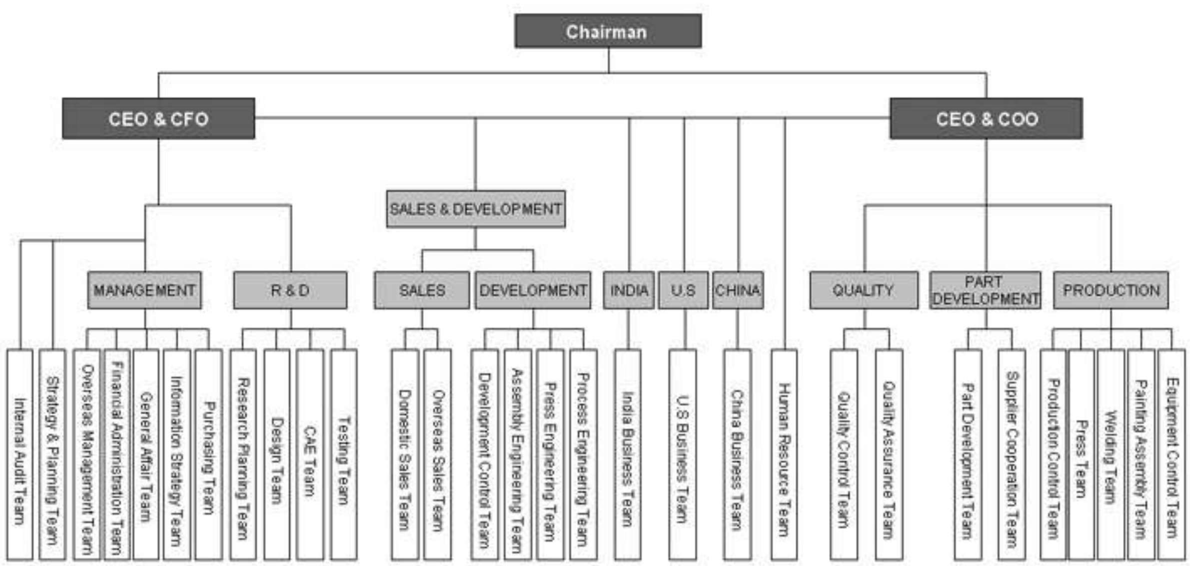

Figure 4: The organizational structure of Hwashin and its various branch offices

In addition to the Front Lower Arm, Hwashin has specialized in the making of small automobile component parts found in the chassis, body, and power train of cars. These include parts such as the engine oil pan, pedal system, and fuel tank. Although many of the parts

Hwashin makes go straight to Hyundai cars, some of its components are also sold to international automobile companies such as Honda, Toyota, and Chevy (Hwashin, 2008).

2.2. Design of Experiments

Design of Experiments (DOE) is a statistical technique introduced by Sir R.A. Fisher in England in the early 1920s. His primary goal was to determine the optimum water, rain, sunshine and soil conditions needed to produce the best crop. Using the DOE technique, Fisher was able to lay out all combination of the factors included in experiment study. The conditions were created using a matrix, which allowed each factor an equal number of test conditions. Methods for analyzing the results of such experiments were also introduced. When the number of combinations possible became too large, schemes were devised to carry out a fraction of the total possibilities such that all factors would be evenly present. Fisher devised the first method that made it possible to analyze the effect of more than one factor at a time.

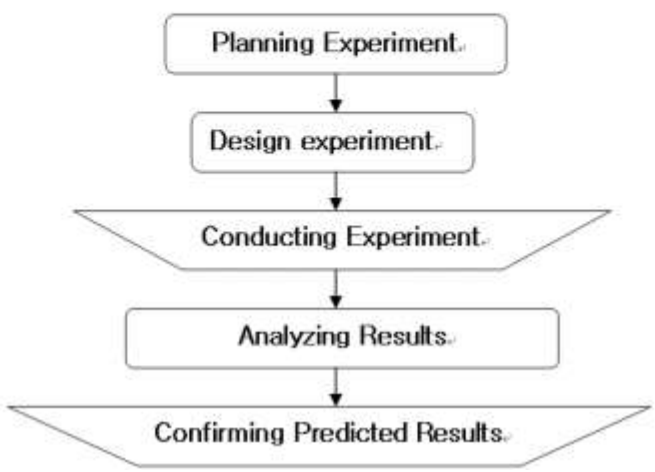

Figure 5: The thought process of DOE.

After Fisher introduced the technique and demonstrated its use in agricultural experiments, much more research and development followed. Unfortunately, most of the work remained in the academic environment. Although the need to study multivariable effects is widespread in the industrial environment, not many industries other than a few segments of the chemical and fertilizer industries have applied the DOE technique in their production processes. In fact, as academic knowledge grew, the further it got from a method that industry could absorb and apply. The more sophisticated the theory supporting DOE became, the less appealing it looked to practicing engineers.

2.2.1. The Taguchi Method

There are many methods in order for design of experiments. However, the industry standard for many practicing engineers is the Taguchi method. This method simplifies the complicated theory into a standardized procedure. The purpose of any experiment with more than one variable affecting various outcomes is to find trends or functions. The first design choice for an experiment is how detailed the trend analysis will be. This is controlled by how many values are given to each parameter. We chose to have three values for each parameter, because we this is a sufficient for a basic trend analysis between our parameters and our performance measures. The second choice is the complexity of our experiment. This affects the total number of cases to be run. Again for a basic trend analysis, the least complexity is sufficient. There is a parameter that measures complexity, *m*, seen in *Error! Reference source not found.*. The lowest possible value for *m* is 2.

$$3^m\left(3^{(3^m-1)/2}\right)$$

Equation 1: The Taguchi Method.

Once these two decisions were made, a simple procedure was followed giving us the final DOE table that we used for our project. *Error! Reference source not found.* is used only for experiments which contain three values for each parameter. If we had chosen a different number of parameter values then we would have used a different equation. The first section of the equation, *3m*, is used for generating the correct number of cases for the experiment. Because m = 2, we must have nine cases. The second part of the equation, $(3m - 1) / 2$ is used to ensure that there is enough complexity in experiment to allow for all parameters. For this type of experiment, you can have up to four unique parameters. Since we only had three parameters, this confirms that our experiment has enough complexity. The Taguchi Method also supplies a matrix for each experiment to be performed with a set number of case values and complexity. We looked up the table for three parameter values and a complexity of two seen in ***Table 1***.Finally the exact values for each parameter could be chosen and inserted into ***Table 1***.

Case	P_1	P_2	P_3
1	A	A	A
2	A	B	B
3	A	C	C
4	B	A	B
5	B	B	C
6	B	C	A
7	C	A	C
8	C	B	A
9	C	C	B

Table 1: The derived DOE table.

There were originally five variables to consider for our optimization plan: R1, R2, Height, Thickness, and Material Composition. However, due to the initial standardization of material composition at FB 540 steel and the subsequent discovery that R2 and H could not remain isolated and independent, our team combined R2 and H. The final DOE Table (***Table 2***) only consisted of three variables: R1, R2/H, and T. The color code in ***Table 2*** shows the approach our team utilized in dividing and assigning the cases.

R1 [mm]	H(R2) [mm]	T [mm]	Mass [kg]
80	3(17)	3.6	2.542
80	8.5(11.5)	4	2.806
80	14(6)	3.2	2.286
75	3(17)	4	2.802
75	8.5(11.5)	3.2	2.282
75	14(6)	3.6	2.55
70	3(17)	3.2	2.278
70	8.5(11.5)	3.6	2.546
70	14(6)	4	2.81

Table 2: Final Design of Experiments Table

Case 1 was a combined effort between all WPI and KNU students in order to standardize the method and approach to attack the problem. The blue cases were done by Group 1 (Derek and Ha-Sik) while the orange cases were done by Group 2 (Francis and Sun Xin). The groups were divided into one KNU student and one WPI student to try to foster a joint-team effort.

2.3. Solid Modeling

Solid modeling is a crucial step in the design process. It allows the user to create a three-dimensional picture of their design without physically creating a prototype or spending excessive time drawing sketches by hand. The solid modeling software used for this project was called CATIA. This was the software package of choice by KNU as well as Hwashin. It was created by Dassault Systemes who also created other engineering software programs such as SolidWorks.

The basic process of solid modeling in CATIA involves drawing a two-dimensional sketch and then extruding or removing material in a third direction. An example of this can be seen in ***Figure 6***. There are also various secondary functions that are also very helpful. The fillet replaces a sharp edge with a rounded corner. As seen in ***Figure 7***, this function can be varied across a given edge. This function is especially useful in removing stress concentrations.

Another important function for our project was the shell function. ***Figure 8***. This hollows out a three-dimensional block into a thin shell of a specified thickness. This modeling function is particularly useful in our project since our part is created from plate metal. Surface modeling is also an effective way to reduce the weight of the part without changing its basic shape.

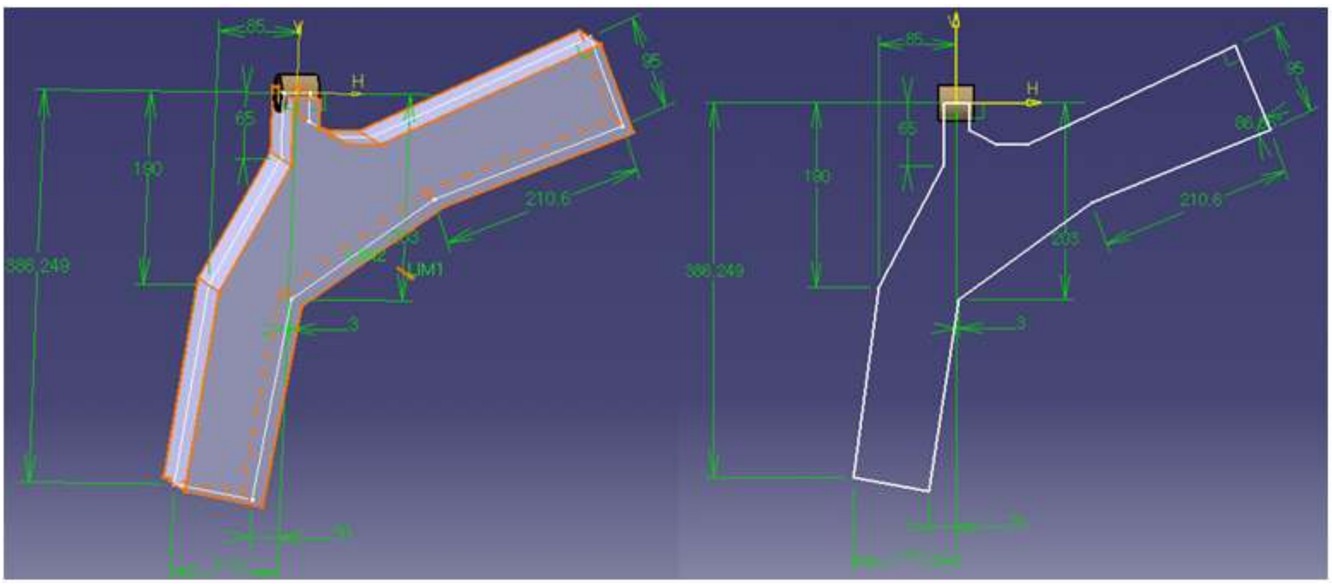

Figure 6: Sketch and Extrusion

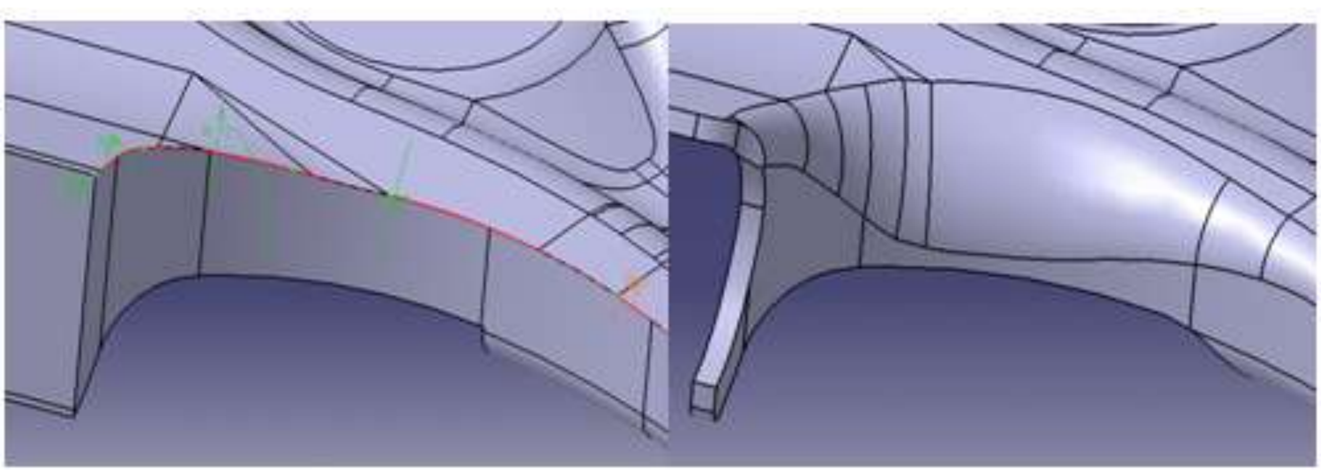

Figure 7: Variable Fillet Radius

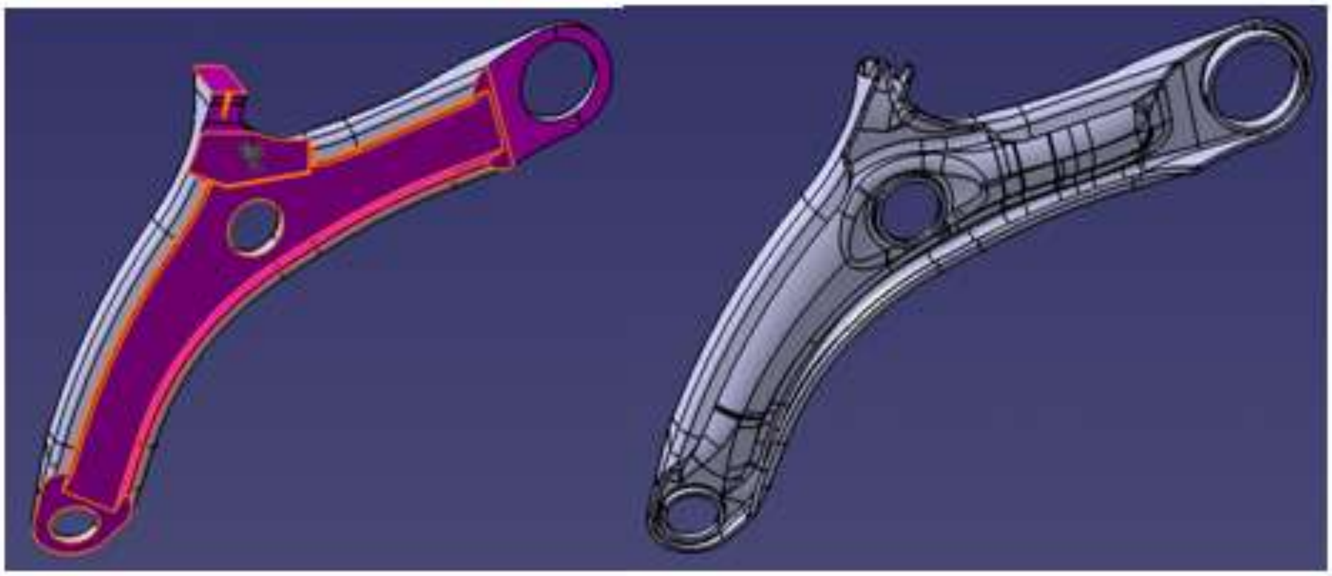

Figure 8: Shell Function

2.4. Finite Element Method

After the geometry of the part is modeled, it must be analyzed for stress analysis. Simple shapes are easily calculated through traditional methods on pen and paper. Parts which have a complex shape are not easily calculated in the traditional manner and a different approach must be taken, The Finite Element Method. This approach cuts the complex part into many small pieces, which must have a simple shape. Because each element has a simple shape, this process has easy and accurate calculations for each element. However, it can take a very long time to calculate by hand if there are very many elements. Because the process contains many repeated but simple calculations, this process is best suited to be calculated by a computer rather than by hand.

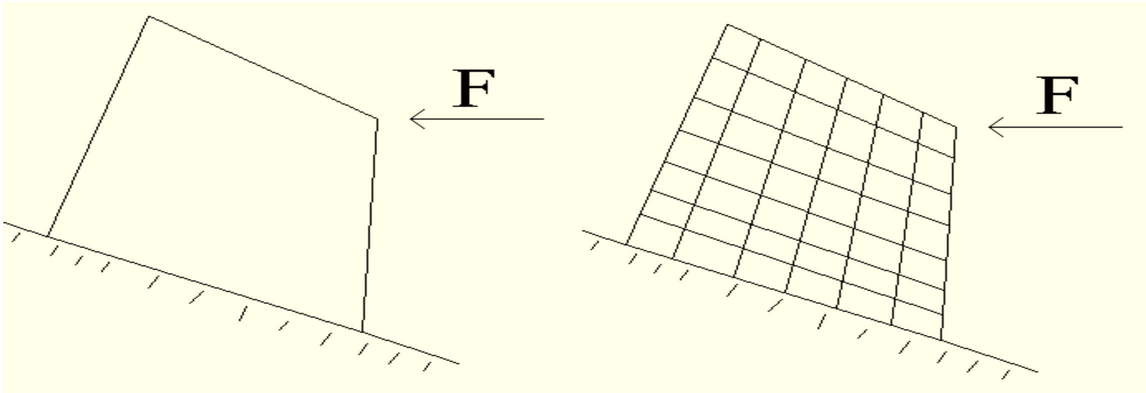

Figure 9: The Finite Element Method principle.

The accuracy of the finite element method is dependent upon the number of elements used as well the element size. However these two characteristics are interconnected. If a part is cut into more pieces, the size of each piece will be smaller. The general relationship between element size and accuracy is an inverted trend. As the element size decreases, the accuracy increases. This process is often more accurate than our traditional approximations for simple shapes. In calculating the stress or deflection in a beam by hand the problem is simplified. The width of the beam is not taken into consideration. However the Finite Element Method does not neglect this consideration and is more accurate by a few percent.

The ways in which a part is cut into elements is called the mesh. The most common element shapes are triangles and quadrilaterals for two dimensional shapes. The meshes for our part included mostly quadrilaterals and some triangles seen in ***Figure 10***.

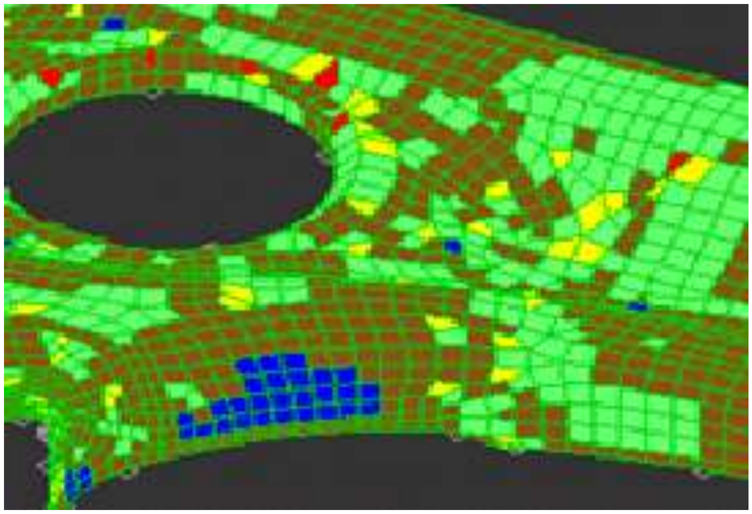
Figure 10: Variation of the mesh quality.

In complex shapes there is little symmetry, so the part cannot be cut into perfect squares or triangles. However there are quantitative measures that define the quality of a mesh. Largely obtuse angles in triangles or squares are considered to be poor quality. The second major characteristic is the length of the side of an element with respect to the average element size. If one side of an element is much shorter or longer than the average element size, then that element would be considered poor quality. Good mesh quality will ensure the accuracy of the analysis so that a larger element size can be used, thus reducing the processing time.

Often times, for very complex shapes, a special mesh creation program is needed. The program we used was called Hypermesh. This program automatically partitions the part when it is imported from CATIA or another solid modeling program. The partitions make it easier to create a smooth mesh. Without these partitions or even with too few partitions the generated elements would be deformed. ABAQUS is capable of generating a mesh for a complex shape, however it requires the partitions be applied manually rather than that the automated and user friendly interface of Hypermesh.

Once the mesh and geometry have been properly produced the remaining of the finite element method follows the same system as a normal stress analysis problem. This includes inputting any necessary material properties, loading conditions, and boundary constraints. Finally the simulation is run in an FEM software program such as ABAQUS and the simulation result is analyzed.

3. Methodology

The Front Lower Arm is a part that starts off as a single sheet of metal and is pressed to conform to its final shape (Park, 2008). As previously mentioned, it is responsible for connecting the wheel to the rest of the automobile. Our team was notified that the Front Lower Arm experienced the highest levels of stress when the automobile abruptly stops or accelerates. By utilizing CATIA for CAD surface modeling and ABAQUS as the program to run Finite Element Analyses, we hoped to optimize the Front Lower Arm to the best of our ability.

Although we were familiar with SolidWorks when dealing with Computer-Aided-Design (CAD) software and were introduced to ANSYS for Finite Element Analysis, our team discovered that CATIA and ABAQUS were the standard for CAD design and FEA respectively at both Kyungpook National University and Hwashin.

3.1. CATIA

As previously mentioned, CATIA was the software used to create surface model changes to each case. Initial attempts to change the given parameters proved difficult mainly due to the complex shape of the Front Lower Arm. After receiving guidance on how the Front Lower Arm was initially created in CATIA from an engineer at Hwashin (Bae, 2008), our team was able to understand the different features we needed to manipulate in order to modify the parameters for each case.

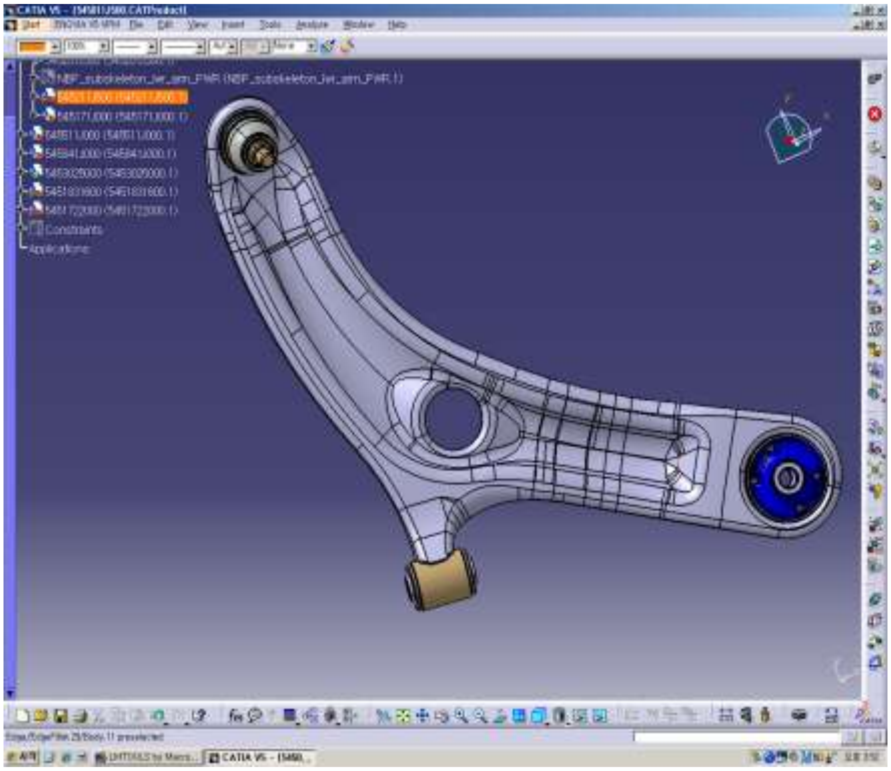

Figure 11: The Front Lower Arm in CATIA as given to us by Hwashin

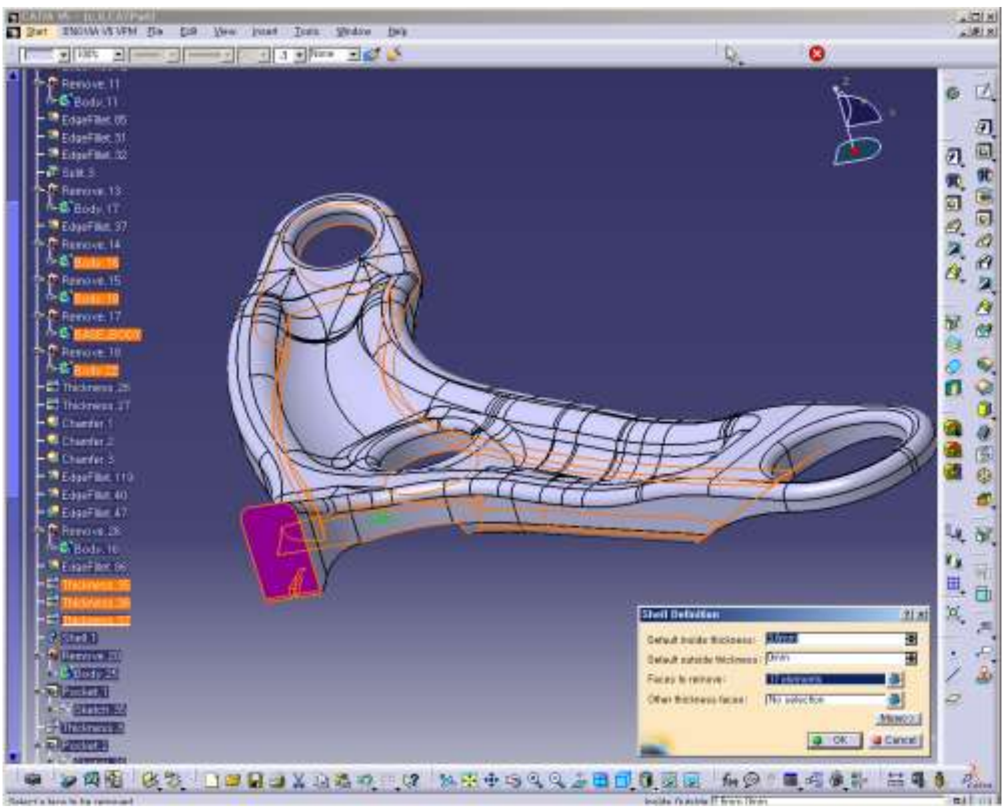

Figure 12: Modifying Shell 1 to obtain values for the mass of each case model.

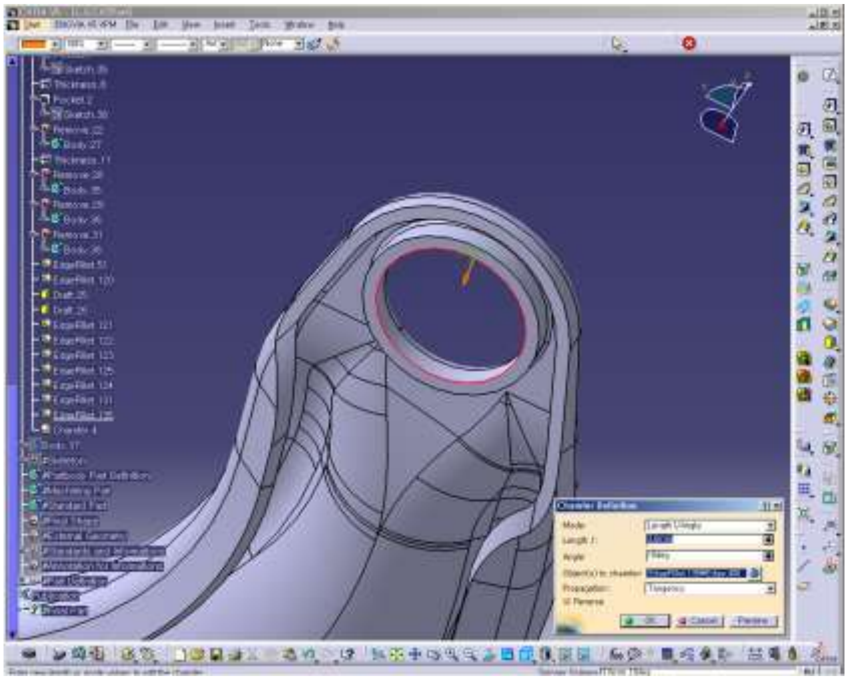

Figure 13: Modifying Chamber 4.

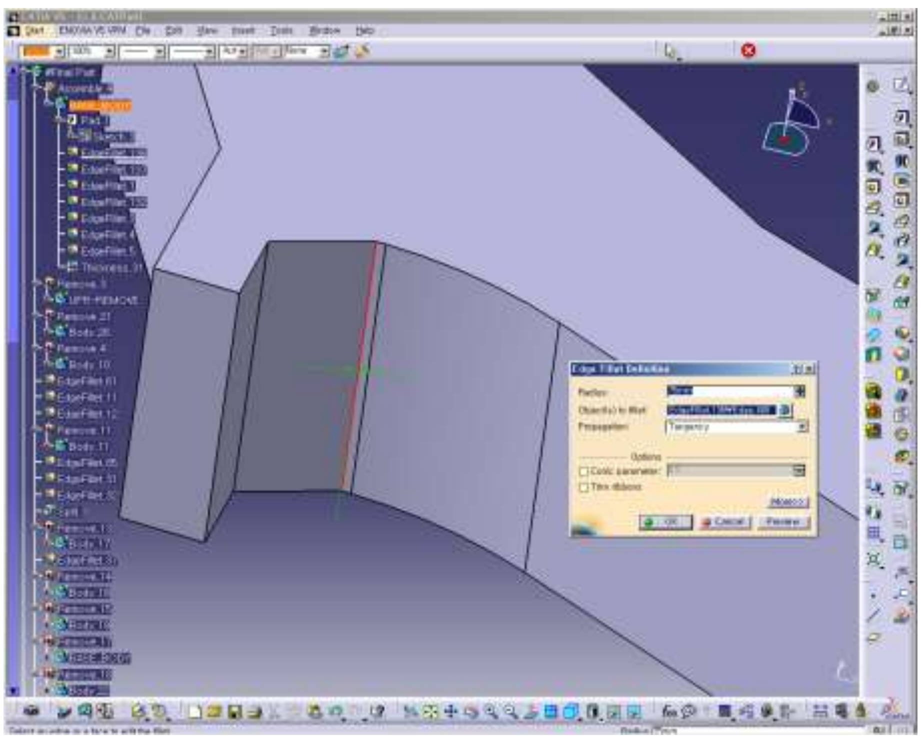

Figure 14: Modifying Edge Fillet 137 to manipulate values for R1.

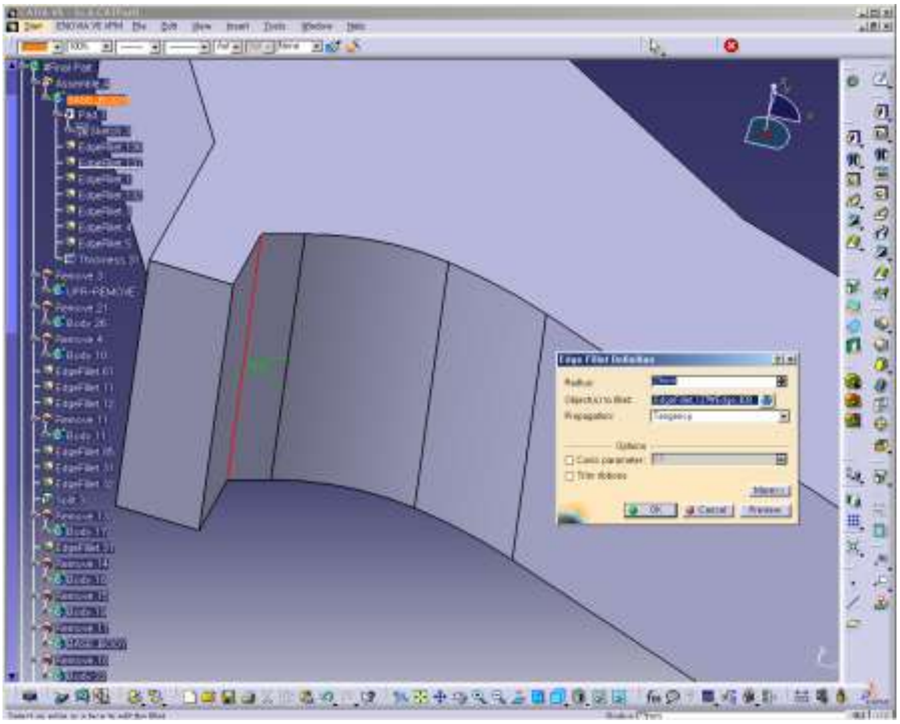

Figure 15: Modifying Edge Fillet 1 to set RM at 23mm.

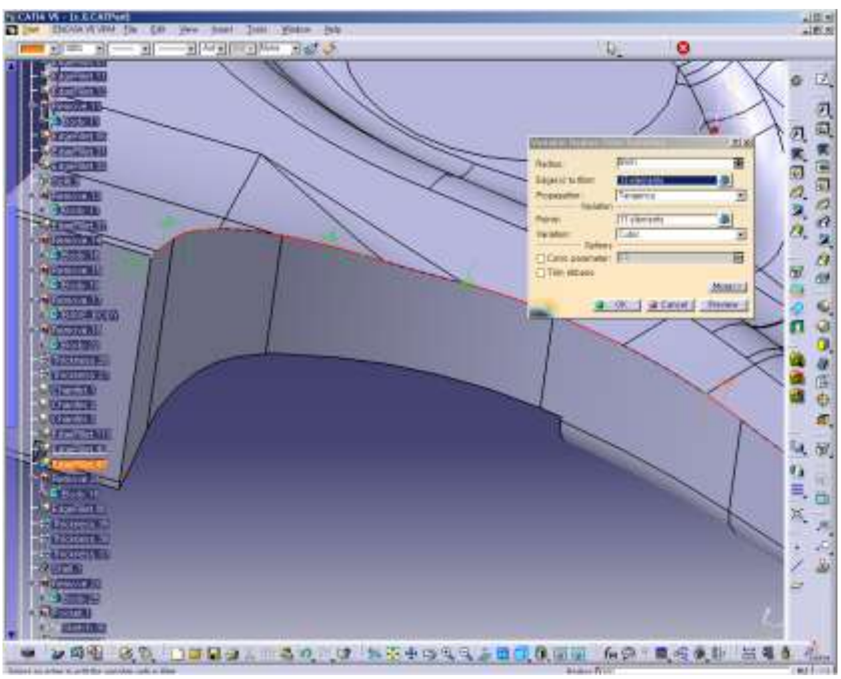

Figure 16: Modifying Edge Fillet 47 to manipulate values of R2/H.

3.2. Hypermesh

Due to the complexity of the Front Lower Arm component, ABAQUS could not produce a mesh for the component when it was imported. If an attempt was made to import the CATIA file directly into ABAQUS, it usually resulted in ABAQUS crashing. Instead, our team utilized Hypermesh, one of the many programs available through Altair Hyperworks.

Hypermesh was an important tool as it allowed our team to generate meshes for each case model and enable itself to be imported into ABAQUS for analysis. Because our FEM analysis was performed as a shell, we created a "mid-surface" and deleted the remaining outer 3D component, allowing us to focus on the mid-surface alone. We standardized the element size at 4mm for the entire Front Lower Arm with the exception of the Critical Area, which was standardized to an element size of 3mm.

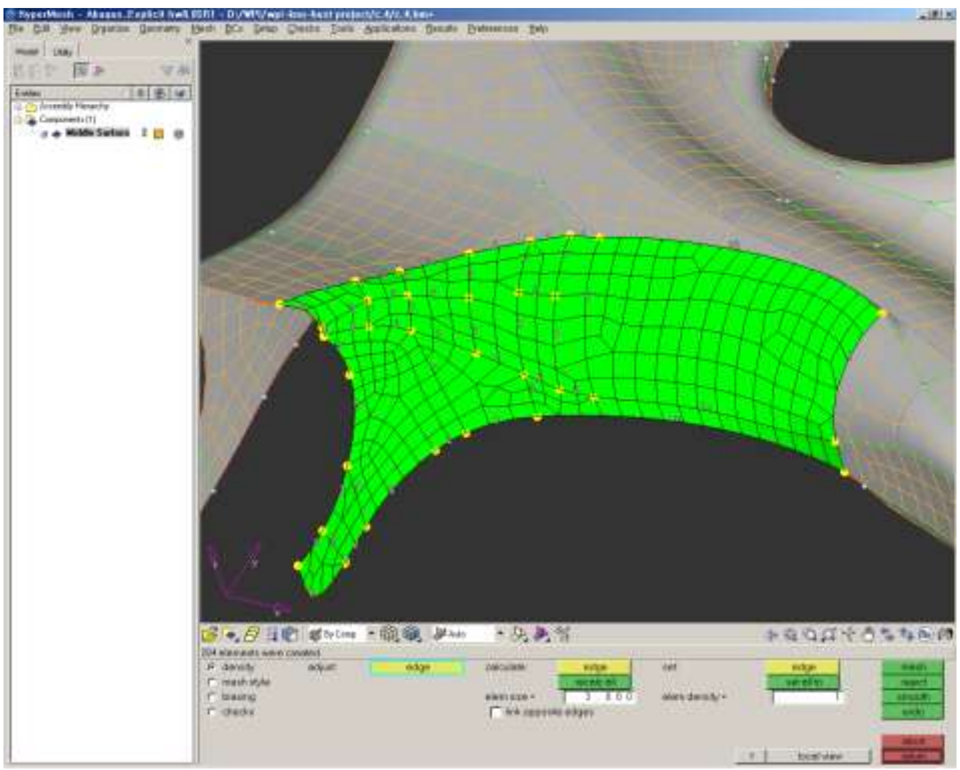

Figure 17: The Critical Area is shown highlighted with green elements.

Although the resulting mesh was sufficient enough to be imported into ABAQUS on its own, our team modified the mesh by creating or deleting design lines in order to accelerate the computing time required for each case model in ABAQUS. Specifically, we deleted sharp, angular surfaces and lines that created very small or tightly constrained areas. Unless edited, a case model could end up with very poor mesh quality; a mesh that would take unnecessarily longer for ABAQUS to compute due to uneven size and shape of the mesh elements.

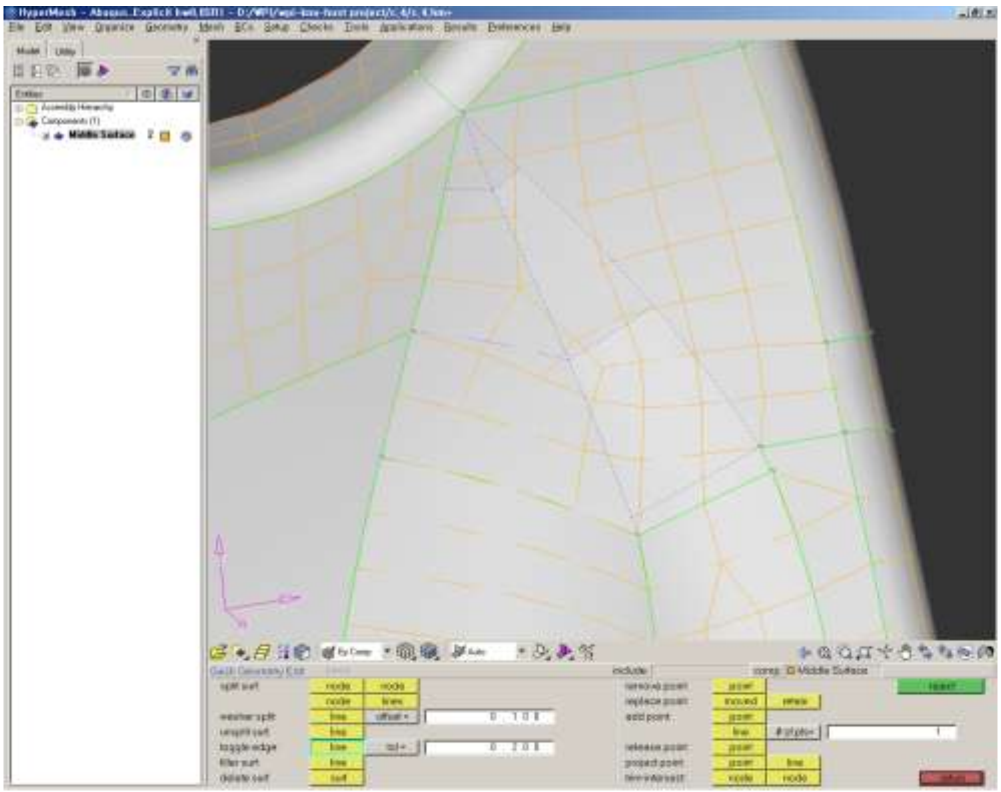

Figure 18: Original features (in green) and deleted features (in blue).

However, it was careful not to delete too many features or else the feature would not generate a mesh properly, resulting in gaps in the mesh. A typical finalized model is depicted as follows:

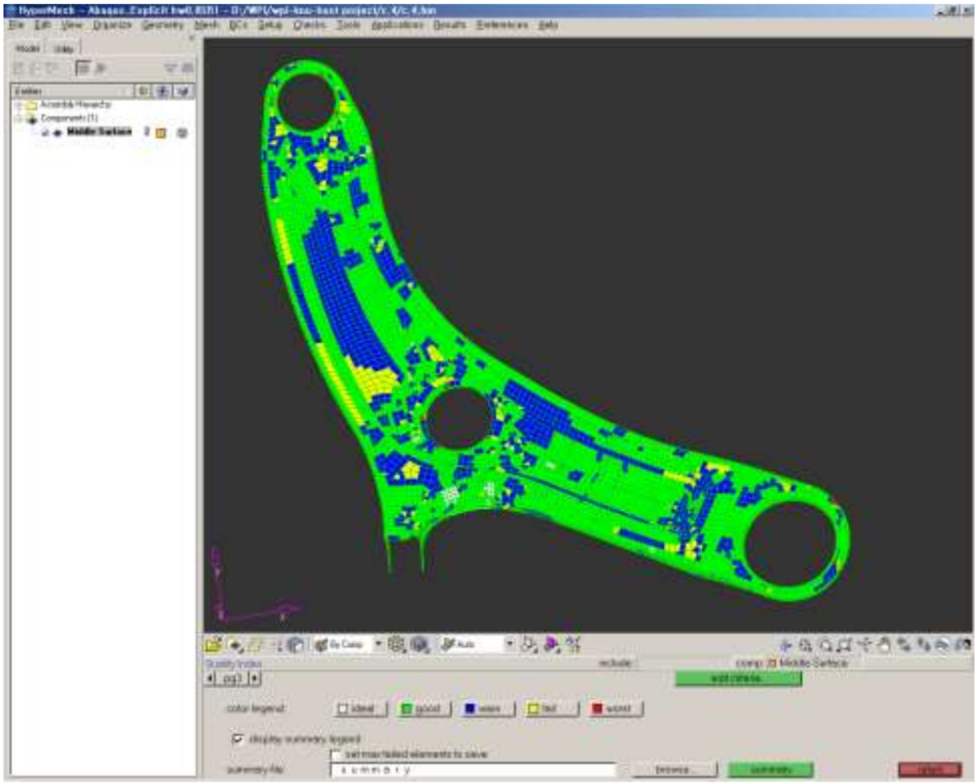

Figure 19: An example of a case model with complete mesh.

We paid particular attention to the quality of the mesh in the Critical Area, making sure that the elements were either “ideal,” “good,” or “warn.” We would also scan the entire part to see if any improvements could be made on the mesh quality of the entire part itself. Once we were satisfied with mesh quality, the file was saved as an exported file compatible with ABAQUS.

3.3. ABAQUS

The actual simulations of each case with initial boundary conditions were done through ABAQUS Computer Aided Engineering software. ABAQUS was the program our team was most unfamiliar with. We spent much of our initial weeks developing proficiency on its capabilities and understanding its interface. Much of the assistance was provided by Myung-Su, who took the time to give us tutorials on ABAQUS as well as troubleshoot when the program encountered errors.

Each case file had its Hypermesh file imported from Hypermesh into ABAQUS. Following the consecutive module sequence, each case file had its material composition defined at FB 540 steel and its initial boundary and loading conditions applied at its respective sections as prescribed by Hwashin. The initial boundary and loading conditions are shown below in ***Table 3***.

Initial Boundary Conditions		
A	B	G
U1	free	free
U2	free	U2
U3	U3	U3

Units: (kgf)

	FX	FY	FZ
A	-147.0	-525.0	36.0
B	320.0	165.0	-3.0
G	-173.0	360.0	-33.0

Table 3: The initial boundary and loading conditions.

Ultimately, we used a general static analysis for each case. We were also able to control the Thickness parameter through ABAQUS by inserting the value in "Shell Thickness" in the Section Manager under the Property Module. ***Figure 20*** below shows the actual dialogue box as seen in the Section Manager.

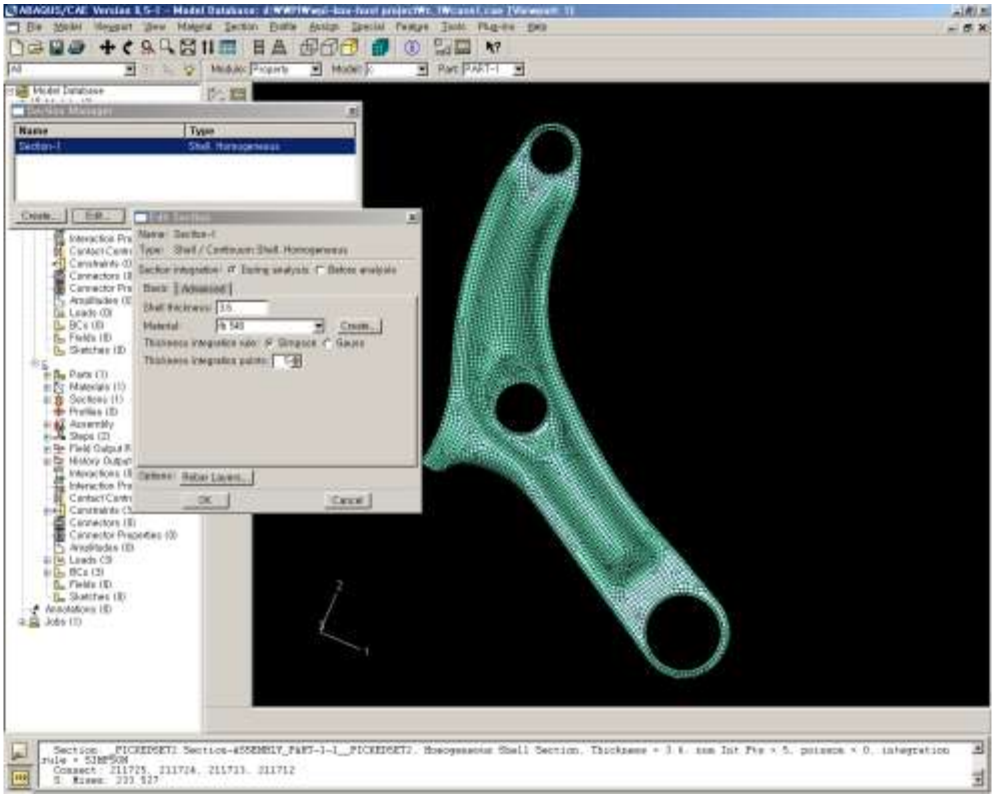

Figure 20: Inserting the Thickness parameter.

Once all preprocessing was complete, the job case was submitted for analysis, which typically took around five minutes to process. The end result was accessible through the Visualization Module and we could see the contour change over time. A query tool allowed us to determine the stress level of each element in the Critical Area.

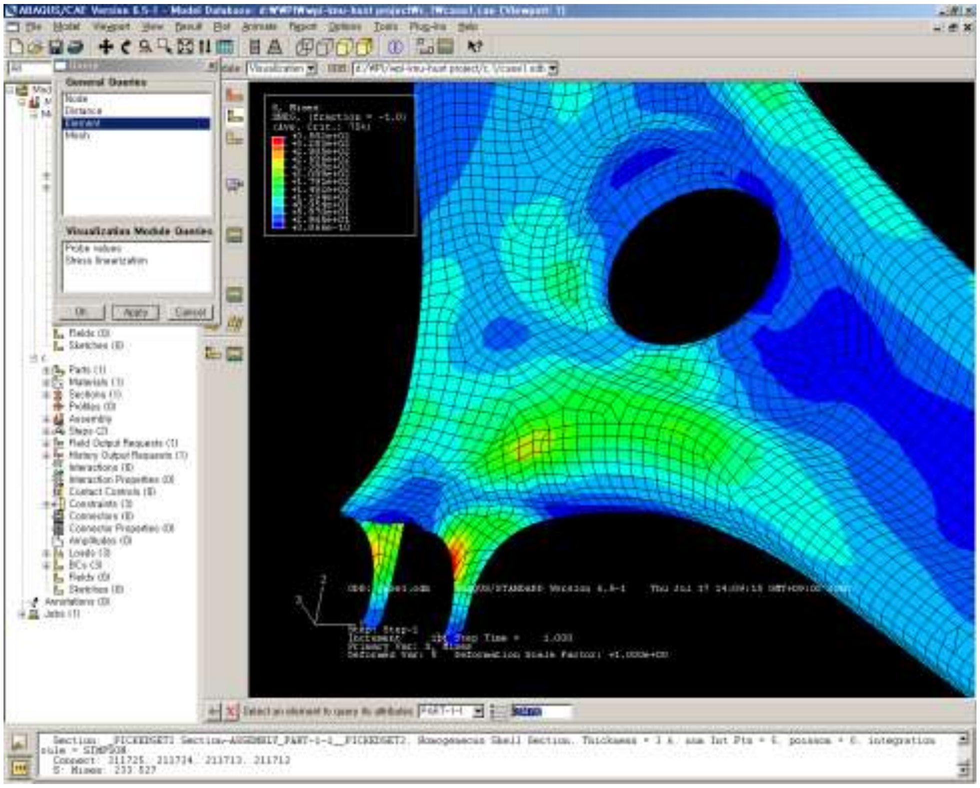

Figure 21: Depicting the Critical Area and its stress for each element.

4. Results

At the end of all nine of our cases, we began a basic trend analysis. We graphed each parameter versus our desired performance measures in a scatter plot. As seen in Figure 22 the relationship between R1 and stress is linear. As R1 increases stress increases. Seen in Figure 23 the relationship between H and stress is non-linear. For high values of height there is high stress, for mid and low values for height there is low stress. Seen in Figure 24 the relationship between thickness and stress is linear. As thickness increases stress decreases. Seen in Figure 26 and Figure 27 there is no relationship between R1 and mass as well as H and mass. And finally seen in Figure 29 the relationship between thickness and mass is linear. As thickness increases mass increases.

The next phase of our analysis was to find the best choices from these nine cases. We decided it would be easiest to graph our two performance measures against one another in another scatterplot. Because we wish to minimize both stress and mass the values closest to the origin will be the best. However some simple statistical analysis can also be used to eliminate some of the cases. If one case is equal or worse than another case in all performance measures, then it can be eliminated. Taking a look at Figure 29 one can see that cases 2, 4, and 9 are heavier and have higher stress than case 8, so these can be eliminated. Cases 1 and 6 have equal mass and higher stress than case 8, so these can also be eliminated. Finally cases 3 and 5 have equal mass and higher stress than case 7, so these can be eliminated. This leaves only two viable cases to be considered for a final design, Case 8, which has the lowest stress, and Case 7, which is lighter than Case 8 but has slightly higher stress.

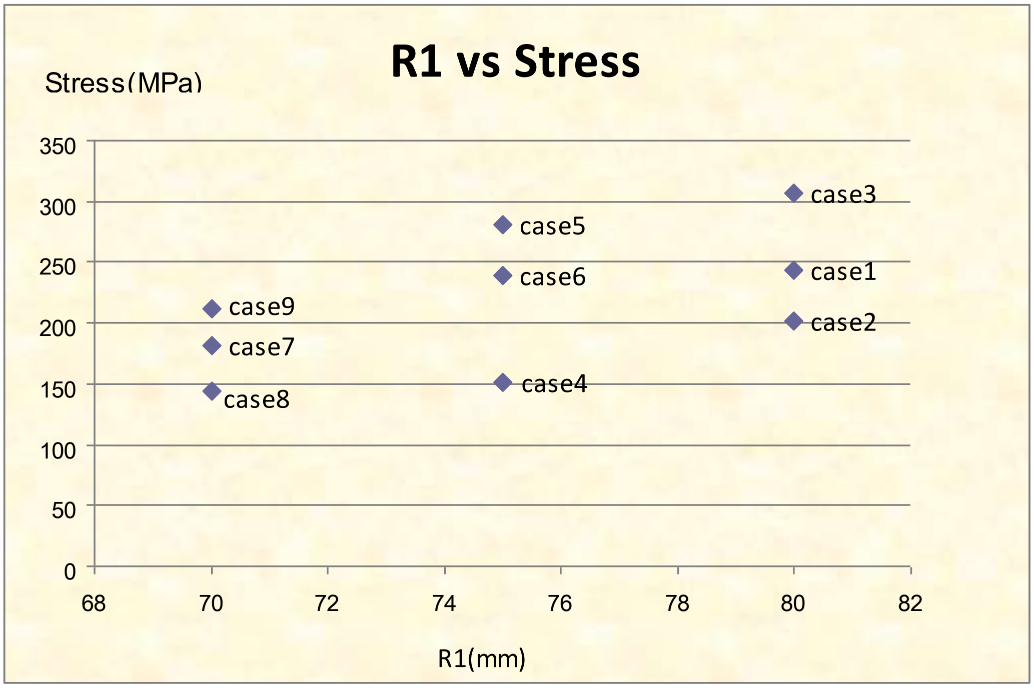

Figure 22: Trends: R1 vs Stress

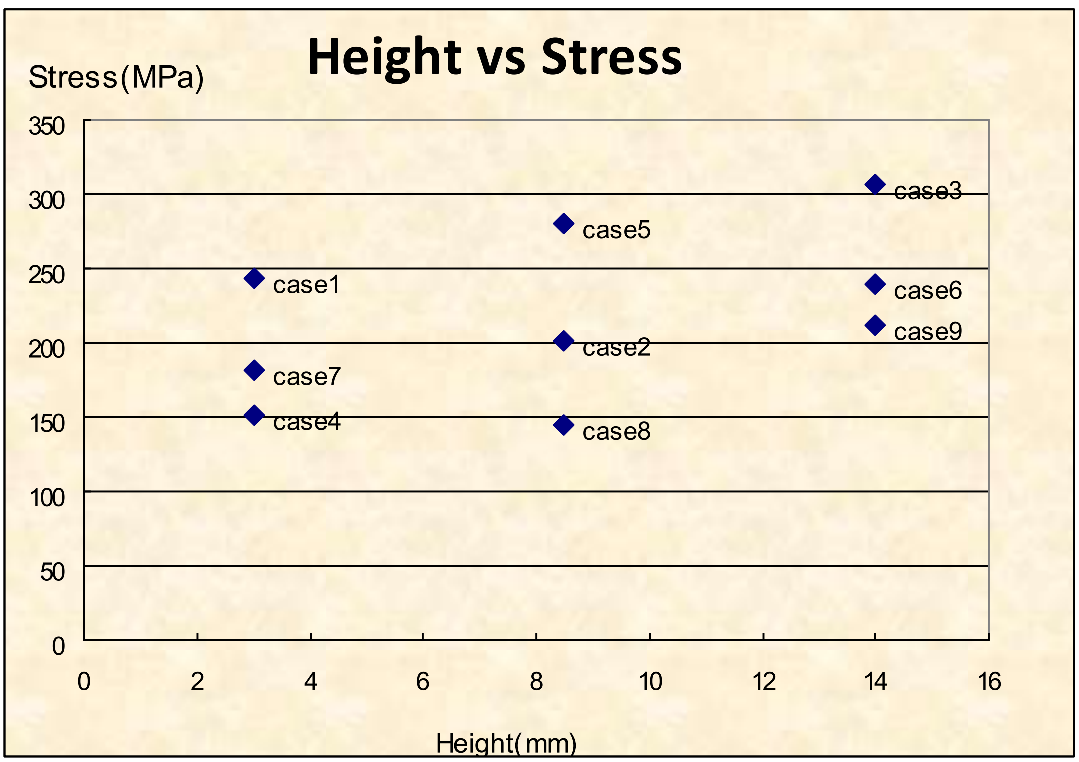

Figure 23: Trends - Height vs Stress

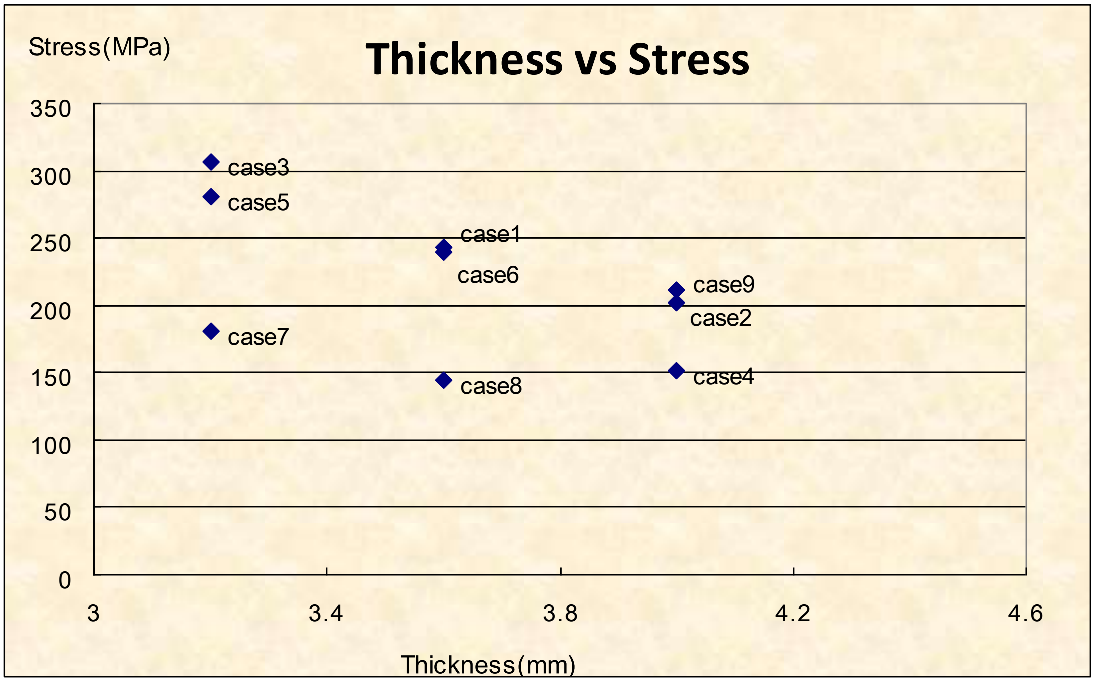

Figure 24: Trends - Thickness vs Stress

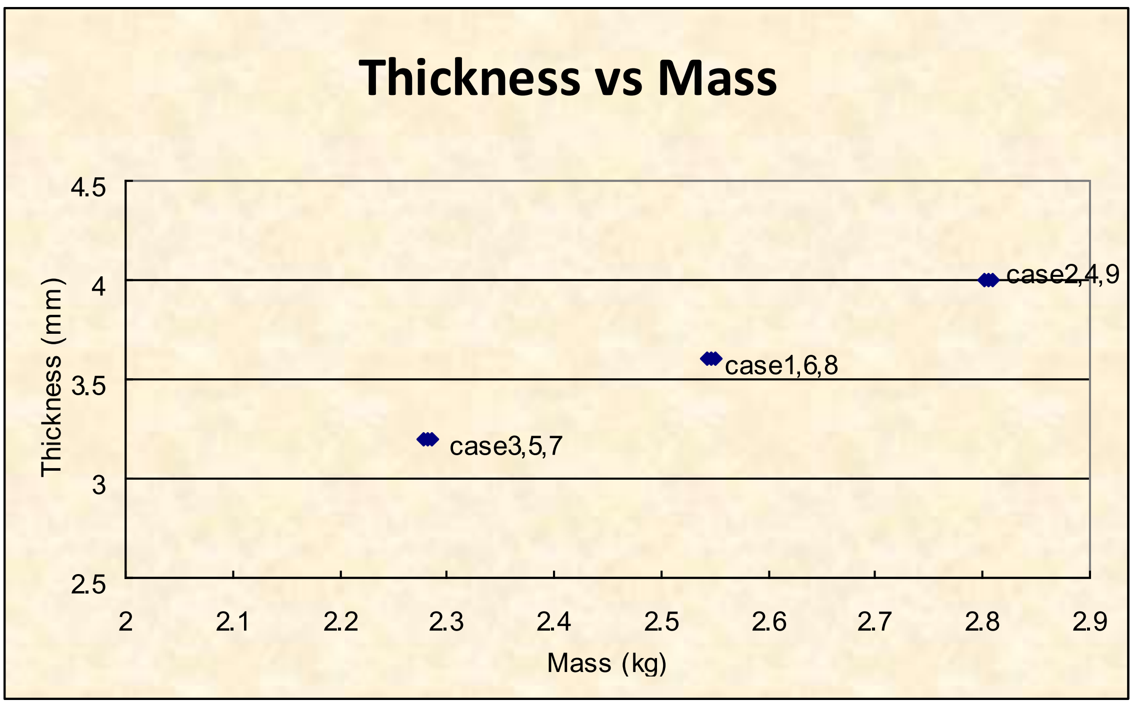

Figure 25: Trends - Thickness vs Mass

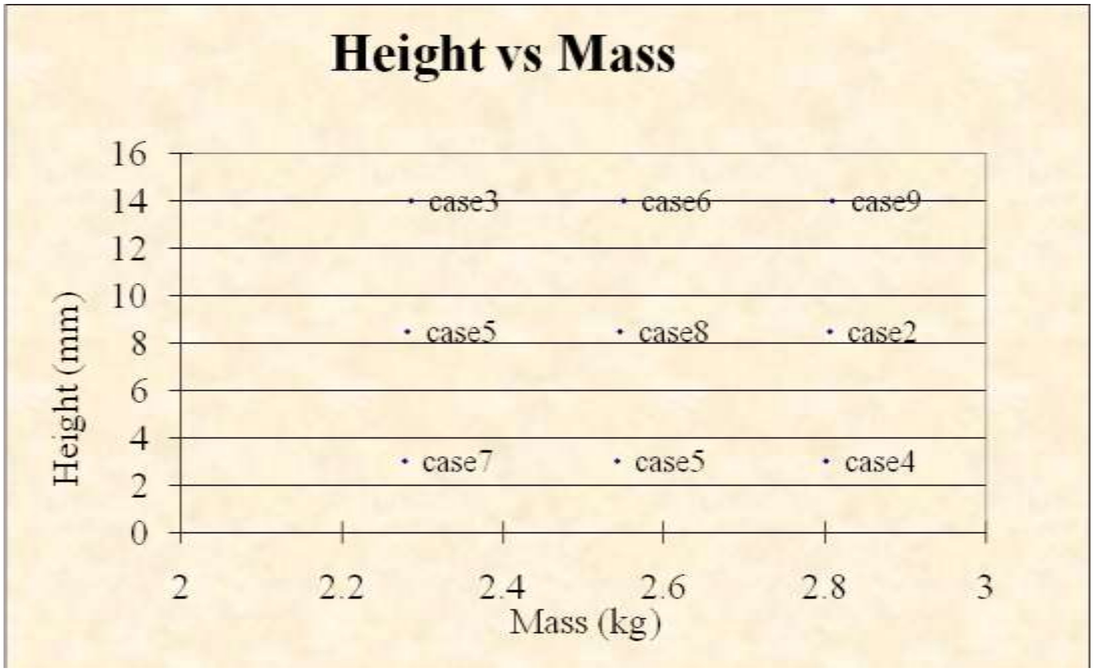

Figure 26: Trends - Height vs Mass

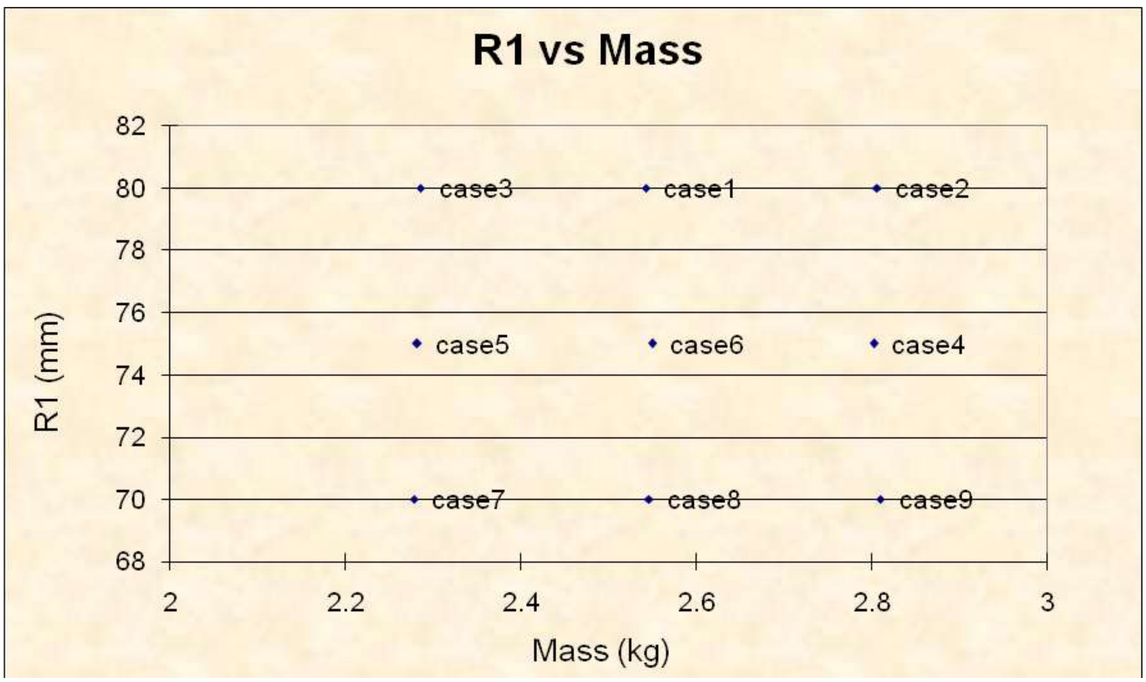

Figure 27: Trends - R1 vs Mass

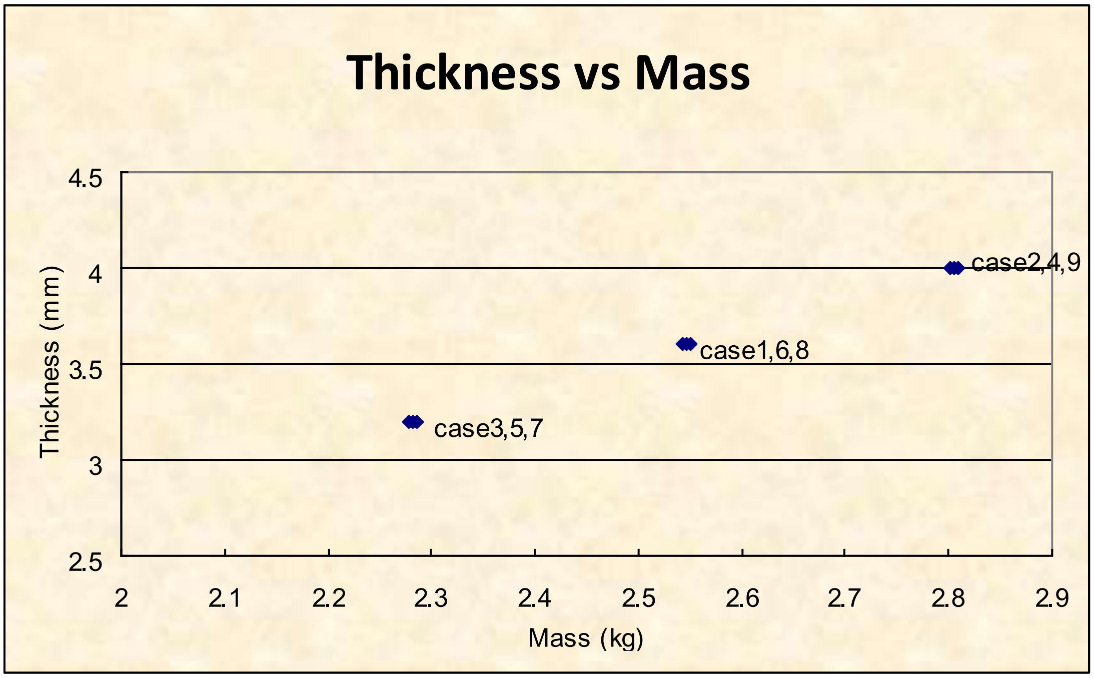

Figure 28: Trends - Thickness vs Mass

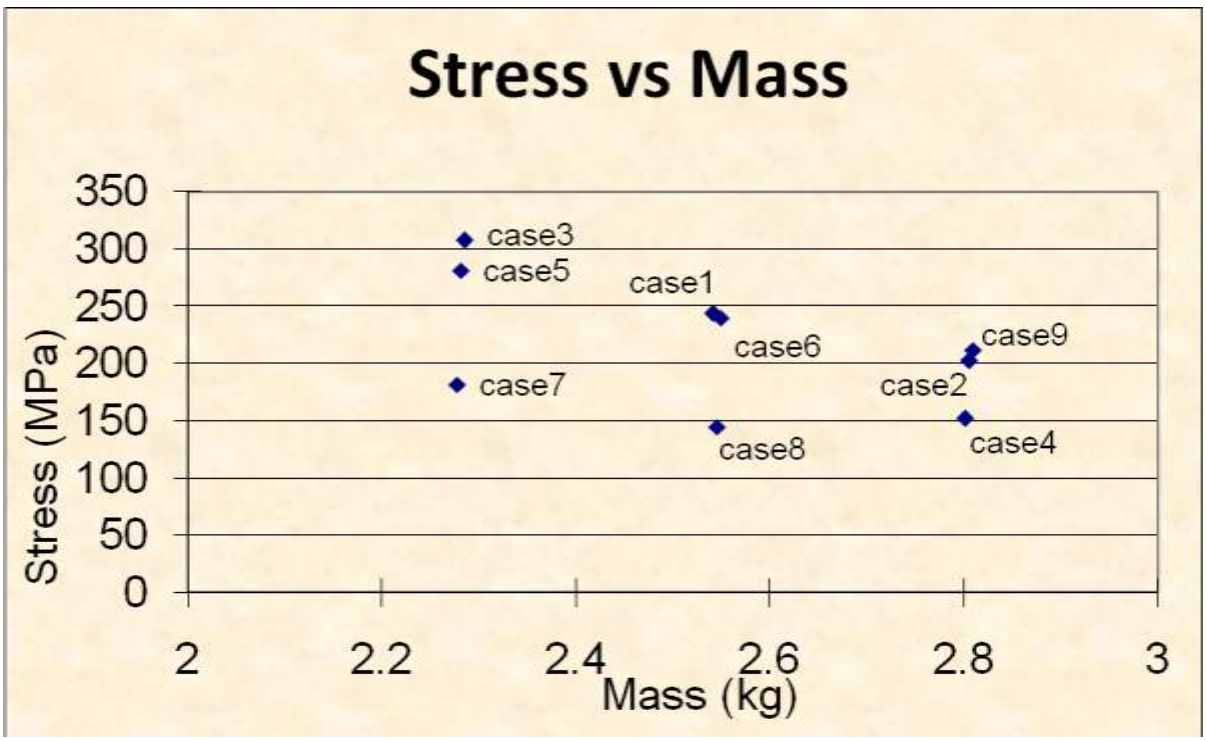

Figure 29: Performance Measure Analysis

Case #	Stress (MPa)	Mass (kg)
Case 1	243.63	2.542
Case 2	201.97	2.806
Case 3	307.037	2.286
Case 4	151.929	2.802
Case 5	280.722	2.282
Case 6	239.605	2.55
Case 7	181.266	2.278
Case 8	144.501	2.546
Case 9	211.501	2.81

Table 4: Comparing stress levels to mass for all nine cases

5. Future Recommendations

Our team strongly believes there is still much work to be done in regards to this project. Our project's goals were very simple and precise and only covered a small part in the optimization of the Front Lower Arm. Hwashin tasked us with modifying a part they were very familiar with to see how it would react under certain conditions and to make the part with as least mass as possible in the process. We achieved this goal but it was at the expense of simplicity and predictability.

In order to truly optimize the Front Lower Arm our team recommends that Hwashin look into other aspects of optimization, such as manufacturing process, cost, compatibility with other automotive parts, and additional simulations of varying boundary and loading conditions. One of the biggest observations our team noticed was the heavy stress concentration particularly found near the A-section. We notified Hwashin of this trend in all nine cases but they insisted we focus on the Critical Area instead.

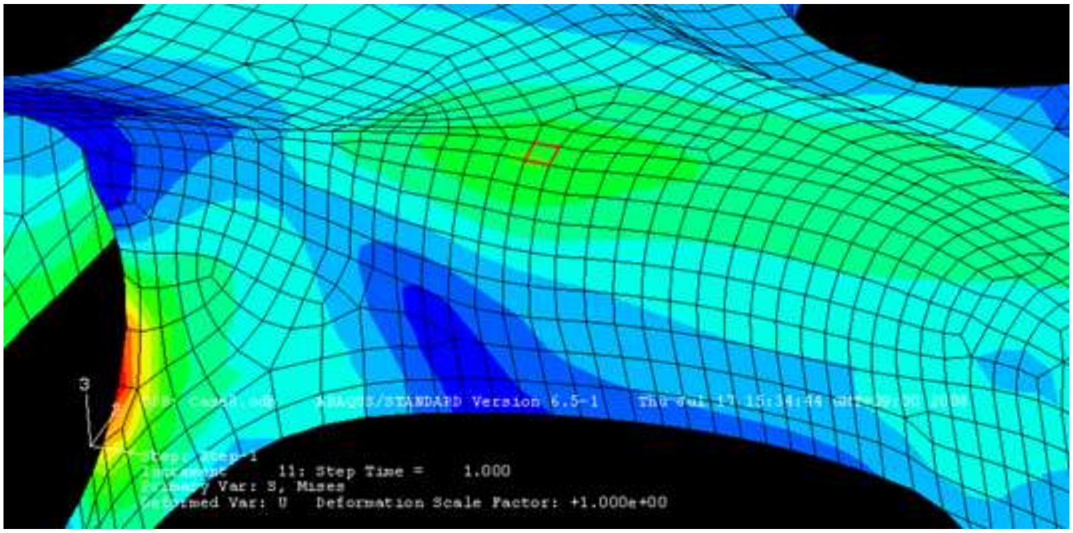

Figure 30: A-section and Critical Area

We recommend Hwashin look into the A-section in reference to future projects if they have not already begun to do so. And it is our sincere hope that the engineers and scientists at Hwashin carefully examine our findings to supplement their own research and trying to find the ideal solution to the problem posed by the Front Lower Arm.

6. Conclusion

Time was the biggest constraint we faced during our project. The time it took to learn and familiarize ourselves with the different programs left us with very limited time to actually implement the project. Given enough time, our team would have tried to implement other ideas to the project such as better refining the mesh and standardizing mesh quality in the Critical Area. Our Teaching Assistants, Myung-Su and Moon-Ki, aided us greatly in adapting to a different set of programs that we had not previously seen before as well as introducing us to new ideas and concepts, such as implementing Design of Experiments through the Taguchi Method.

The language barrier also posed a significant challenge, as the Teaching Assistants spoke very little English and our two partners knew almost none at all. Fortunately, Francis knew how to speak some Korean and was able to act as a translator in most situations.

Communication between our team and Hwashin Company could also have been improved. Unlike traditional WPI projects, our team worked at KNU and only visited Hwashin Company twice, leaving us with email and phone as the only means of communication between the two parties. We were able to meet with Mr. Park and Mr. Mune on both occasions, who still took the time to answer our questions and try to make the project clearer even though both had busy schedules. Additionally, Mr. Park was also able to stop by for both the mid-term and final presentation.

We hope that our project will be the first of many between KNU, WPI, and Hwashin. Even though Hwashin may seem like a small company, its international presence and its motivation to make the parts they make the best as possible make Hwashin an ideal company to continue completing projects for. Our project itself may create future ones someday if Hwashin decides to implement our suggestions.

The project itself opened our minds to unfamiliar ideas and concepts and introduced us to new places and environments. We learned that it is this ability to adapt to uncertain and unpredictable conditions that define a good engineer from a bad one, and that in engineering, nothing ever gets accomplished if you attempt to work on your own. These lessons were the most important rewards we received from completing our MQP and one we will never forget.

7. Works Cited

Bae, K.-U. (2008, July 1). Front Lower Arm Creation and Modification. (S. Francis, & D. Silsby, Interviewers)

Fisher, R. A. (1951). *The Design of Experiments* (6th ed.). New York: Hafner Publishing Company.

Frost and Sullivan. (2005, May 10). *Country Industry Forecast - The South Korean Automotive Industry*. Retrieved July 24, 2008, from Frost and Sullivan: http://www.frost.com/prod/servlet/report-homepage.pag?repid=4532-01-00-00-00

Kang, B., Sin, H.-C., & Kim, J. (2007). OPTIMAL SHAPE DESIGN OF THE FRONT LOWER WHEEL LOWER CONTROL ARM CONSIDERING DYNAMIC EFFECTS. *International Journal of Automotive Technology , 8* (3), 309-317.

Roy, R. K. (2001). *Design of Experiments Using the Taguchi Approach.* New York, New York, United States of America: John Wiley & Sons, Inc.