Modeling And Development Of Active Bumper System To Reduce Shocks For A Vehicle By Using FEA

[1]D.Srinivasulu, [2]M.Haritha
[1]Student, [2]Assistant Professor
QIS College of Engineering & Technology Ongole

Abstract - In this paper to investigation on modeling of a front bumper system with emphasis on lightweight and low cost, is a collaboration between Chalmers University of Technology and Volvo Cars. The goal of the project is to reduce the cost and weight of an front bumper system, also called Crash Management System (CMS) for the latest Volvo XC40 using Design to Profit approach. The current CMS is made out of aluminum in 6000, 7000 series and has a total weight of 9,114 kg. By defining the current CMS components, the project group divided the CMS into four different categories to be investigated; Front bumper beam, Crash box, Tow and Reinforcement. By glance, the beam was divided into two cells and the crash boxes. The dimensions for the inner and the outer walls combined with the weight were changed. Several concepts were evaluated and the remaining ones, after filtering, were simulated in a CAE program, Thin lin. From the simulations the concepts were compared with current one through crash test simulations. The simulation comparison was based on forces ANSYS stop time created during a crash. After many simulation iterations a successful concept was found. The new developed crashbox weighs 8,3 kg which is a weight reduction of 8,93%, saves approximately 7 MSEK/year and has a shorter displacement during a crash. The new crash box achieved good results and further simulations with the whole CMS are of interest.

Key words - Front bumper beam, Crash box, Tow and Crash Management System

1.INTRODUCTION

Volvo is a unique car company, where everything starts with people. Since 1927 Swedish culture and focus on humanity have all helped to shape Volvo's philosophy, to help people make their life easier and more enjoyable whilst also protecting them form harm. The company's statement is: "Cars are driven by people, the guiding principle behind everything we make at Volvo therefore is and must remain safety". This emphasis on safety, quality and people, has been a feature of the company's cars ever since. As has for passion of pioneering, automotive and environmental innovation of distinctive Swedish design. This rich heritage continues to inform what Volvo does today and shapes its future. Volvo cars might never had happened if there was not a meeting between two friends, that shared a passion for cars and a vision to start a Swedish car company, in a Stockholm restaurant 1924. After that meeting Volvo's founders, Assar Gabrielsson a engineer specialist and Gustav Larson a talented engineer with a background on engine design began to make plans for making the finest Swedish car possible.

Figure 1: Vehicle with Chassis and Bumper Locations Labeled

2.LITERATURE REIEW

Wiedenman et.al [1] studied the complexity of modern technology continues to increase, the time it takes to design and verify systems which integrate these complex technologies is increasing rapidly. Johnson et.al [2] discussed, the project aims at compressing the product development and deployment timeline for complex military systems. META focuses on developing a model-based system engineering framework to enable architectural analysis of systems during the conceptual design phase, allowing the user to make cost-saving, safety-enhancing decisions from an early point in time. The report discusses the development of the component library, design space exploration method, and failure identification. Kurtoglu et.al [3] Identification of failure points is one of the keys to successful design, and is important in attempts to shorten the design cycle. Kurtoglu published a paper in 2010 detailing a simulation-based failure analysis tool which analyzes failures and their propagation to allow system level architectural designers to make decisions based on risk on a functional level before specific components are selected. Baja et.al [4]. An initial system under consideration for these studies is a 1/5th scale model of an actual racing truck, the Baja 5sc Super Sport remote control vehicle made by HPI Racing. Based on the exhaustive literature will detail the modeling, various types of analysis, and results from the finite element studies for the two parts being considered. The ANSYS program will be used for all finite element analysis [ANSYS]. These studies, although not extremely complex, provide good examples of studies that would actually be done in industry, and may factor in to the iterative design loop of the vehicle. Upon conclusion, it will be determined how using these results could help to optimize design from a top down level.

3.MODELING OF THE MAIN CHASSIS AND BUMPER

3.1 MAIN CHASSIS

The main chassis component is the most important structural member of the vehicle. Located in the center of the vehicle, the main chassis houses several key power train components, and also serves as a connection between the front and rear bulkheads of the vehicle. The chassis, like most structural members, is subject to many loads, both structural and thermal. It provides a good part for analysis as these loads are often predictable and quantifiable. The main chassis was first modeled in the CAD tool, with the resulting geometry shown below. This geometry can then be easily imported into the FEA tool and edited as necessary as shown in Fig.3.1.

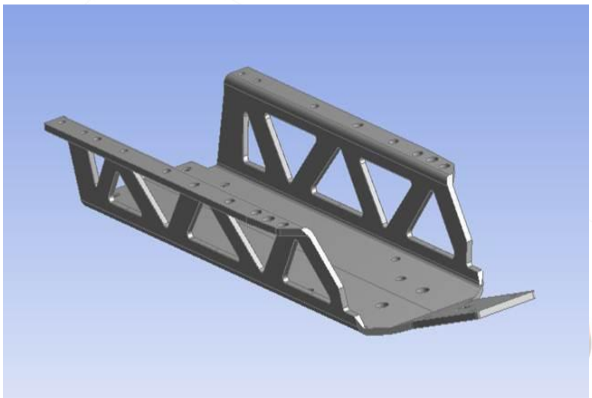

Fig. 3:1 Main Chassis CAD Geometry

3.2 FRONT BUMPER

The next part to be analyzed was the front bumper, another important structural member located at the front of the vehicle designed to protect the vehicle from damage due to collision. The bumper is unique in that it does not have much static load, and should, ideally, never have to take a load from a collision. However, in the case of a collision, the bumper must deform such that it absorbs most of the force without damaging the rest of the vehicle.

As with the main chassis, the first step in a finite element analysis is to model the geometry. For the front bumper, the part was modeled with the CAD tool and imported into the geometry section of the FEA tool. The model of the part is shown below.Fig.3.2.

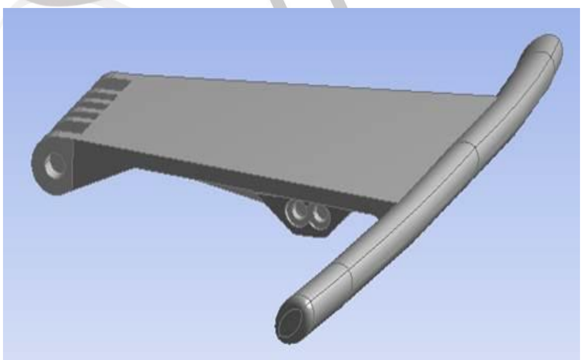

Fig.3.2: Front Bumper CAD Geometry

4.RESULTS AND DISCUSSIONS

4.1 STATIC STRUCTURAL ANALYSIS OF CHASSIS

The main chassis part that is under consideration is aluminum, and for the purposes of this study, the “Aluminum Alloy” material properties will be used, with a density of $\rho = 2770$ kg/m^3, Young’s Modulus of $E = 71$ GPa, and Poisson’s Ratio $v = 0.33$.

Next, the loads and supports were added to the model. The sum of the forces on the body has to be equal to zero for a static object. Therefore, the sum of the supports acting in directions opposite of the applied loads has to be equal to the sum of the applied loads in order to have a statically stable system. In this case, all of the loading on the chassis acts downward on the base plate of the chassis, and the chassis is supported at the two raised flaps where it connects to other structural members. The support was modeled as a fixed support on the two underneath surfaces of the flaps. The loads were slightly harder to characterize, but were also very important to the integrity of the study, as incorrect loading can greatly affect the results in a negative way. The first was a 150 N force distributed over the surface of the base of the chassis to account mainly for engine weight as well as other connections and parts. The second was a 50 N force acting at the four holes in the necked portion of the chassis, acting in the negative z direction (downward). The two forces were both conservatively high, and accounted for the static load of the vehicle under normal operating conditions. A figure showing the chassis with the fixed support and applied forces is below as shown in Fig.4.1

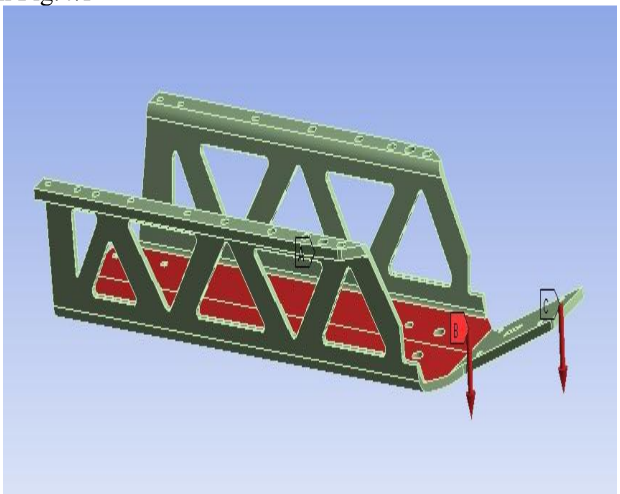

Fig 4.1: Chassis Loads and Supports

A Fig.4.2 showing the resulting stress distribution is below. Important things to note are the maximum stress value of roughly 67 MPa, occurring at the edge of the region where the necking in the chassis occurs. This stress concentration was to be expected, as the load on the neck acting at a distance from the fixed supports creates a moment arm causing a bending stress on the neck region. The bending stress causes a tensile stress on the top face of the chassis and a compressive stress on the bottom face. However, maximum stress must always be considered relative to the ultimate tensile strength of the material. Aluminum's tensile strength is around 300 MPa, so even at the point of maximum stress, the part still has a factor of safety of 4.2.

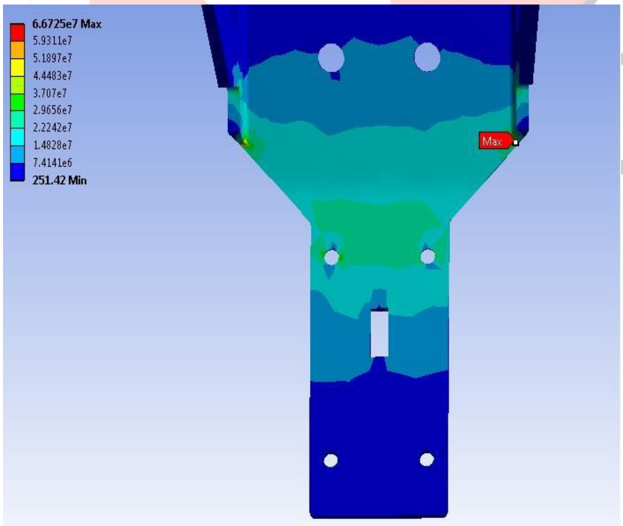

Fig. 4.2: Chassis Stress (MPa)

The other result to consider was the deformation of the chassis as shown in Fig.4.3. Excessive deformation could have very negative effects on the overall structure of the vehicle. The study showed a maximum deformation of only 1.1 mm at the very edge of the neck of the chassis. This small amount of deformation is negligible, and will have no effect on the structural stability of the vehicle.

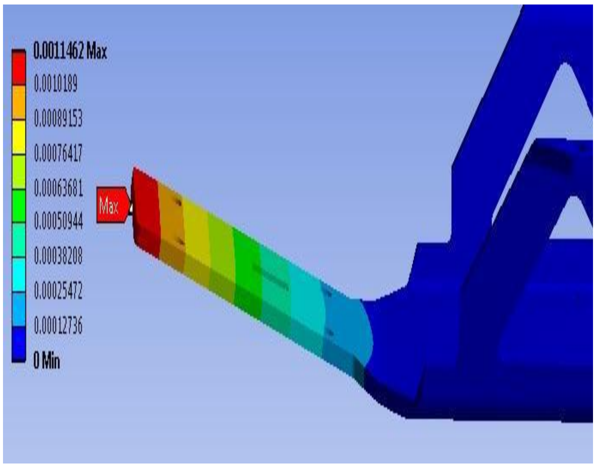

Fig. 4.3: Chassis Deformation (meters)

4.2 VIBRATION ANALYSIS OF CHASSIS

For a modal analysis study, the only conditions that need to be applied by the user are the supports. The solver is then able to determine the natural frequencies of the structure. In this case, a fixed support was applied under the two horizontal structural flaps, as in the static structural case. However, unlike the static structural case, where a load was applied to the neck portion where the chassis attaches to the front bulkhead, a support, rather than a load, was applied in this case. This is because the bulkhead applies the frequency to the chassis, so the two moves relatively to each other, and either may be considered fixed to study the dynamics of the other.

After applying and verifying the necessary support conditions, the problem was solved, and the first 4 natural frequencies were found to be 1168.2 Hz, 1640.8 Hz, 1712.6 Hz, and 1911.7 Hz. Images showing the maximum deformation of the chassis at each of these frequencies can be found. To check the relevancy of these results, the frequencies that the chassis will be exposed to had to be determined.

The chassis is subject to two main vibration loads during vehicle operation. The first is the frequency applied by the terrain on the vehicle. For example, if the vehicle is traveling on gravel, the applied frequency could be as much as 30 to 50 Hz. This frequency, although small, could potentially have impact. The other applied frequency is due to the engine. The engine used in this vehicle is a Fuelie 26S CC engine which has a maximum rpm of 15,000. This corresponds to an induced frequency calculated using the simple conversion shown below

$$\frac{15000\ rev}{min} * \frac{1\ cycle}{1\ rev} * \frac{1\ minute}{60\ seconds} = 250\ Hz$$

The two applied frequencies are therefore roughly 40 Hz and 250 Hz. The modal study in ANSYS gave a minimal natural frequency of 1168 Hz, more than four times the maximum applied frequency. These results suggest that the chassis is designed such that the resonant frequency is not a factor, implying a good design. Resonant frequency can be a concern on some parts which are thinner or have a smaller stiffness, and should therefore always be considered. There are several ways to redesign a component to change its natural frequency, including changing the thickness of structural elements or adding support members to the component to change the stiffness.

4.3 THERMAL ANAKYSIS OF CHASSIS

The third and final study performed on the chassis was a thermal study. Thermal studies are important in scenarios involving large amounts of heat being generated that need to be dissipated. The important case to consider with a thermal study, as with any study, is the worst-case scenario. In the case of this vehicle, the engine produces a majority of the heat due to the engine combustion and frictional losses of the internal components of the engine. The engine sits directly on the chassis, so it can be assumed that a portion of the heat generated will flow to the chassis through conduction. From there, the heat is dissipated out through the chassis to other structural members, as well as to the air moving past the vehicle in the form of convection.

The first step in modeling this thermal scenario was to determine how much heat energy the engine was producing. From the manufacturer's data Baja et.al [6], the engine is capable of producing 2.9 horsepower of power, or 2160 Watts. This means that the engine shaft output is 2160 W. However, much more energy had to be consumed in order to overcome the losses of the engine, as no engine has an efficiency of 100 percent. The engine efficiency suggests that 30 percent of the energy produced is actually transmitted to the shaft as mechanical energy, meaning the other 70 percent is lost as heat. Therefore, for every 2160 W of mechanical energy produced, 5040 W of heat is produced. Of this 5040 W, only about 10 percent is actually transferred to the chassis, due to high rates of convection. This calculation is shown below.

$$2.9\ hp * \frac{745.7\ W}{1\ hp} = 2160\ W * \frac{70}{30} = 5040\ W * 0.1 = 504\ W$$

Therefore, a conservative estimate of 500 W of heat energy is transferred to the chassis during maximum operating conditions. Although conservative, this load provides a good worst case scenario.

The heat transfer coefficient (h) and the ambient temperature must be known. These can be assumed to be h = 150 W/m^2K and Tamb = 22°C, based off data found in heat transfer tables for moving air Kays et.al [7].A steady state thermal analysis was used, as the heat load will not fluctuate from the maximum possible load. The two external conditions that need to be applied are the heat flow into the chassis from the engine, and the convection of heat out of the chassis to the surrounding air. As discussed above, a 500 W heat load was applied to the base face of the chassis, and convection with a heat transfer coefficient of 150 W/m^2K was applied to all faces of the body. The desired solution information of the temperature profile of the entire body was requested, and the problem was ready for solving.

The heat transfer problem was solved, and the resulting temperature profile is shown Fig.4.4 below, with temperatures given in degrees Celsius. The maximum temperature was found to be 78°C in the middle of the base of the chassis, with the minimum temperature of 28°C located at the edge of the neck portion of the chassis.

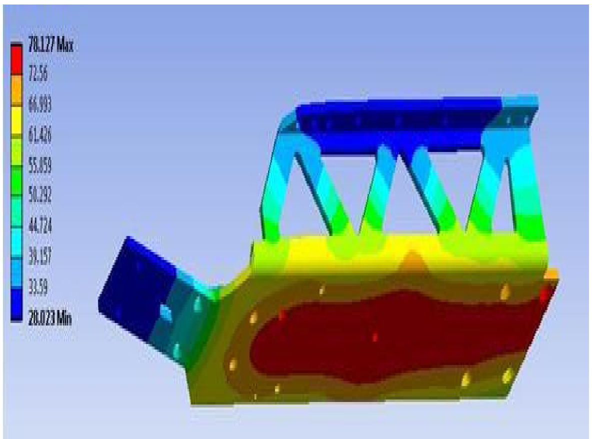

Fig.4.4: Chassis Temperature Profile (°C)

4.4 IMPACT ANALYSIS OF BUMPER

The first study to be considered was an impact study, designed to see how the bumper would react to different collision scenarios. This study was modeled using a static structural analysis in the ANSYS finite element analysis tool.

The Second step was to assign a material to the geometry. The modified polyethylene was selected for the entire body using the material properties listed above. The next step was to mesh the part to ensure that the mesh was fine enough for a realistic analysis. The mesh relevance was increased to 100, and a refinement was added on the front edge of the bumper to handle the higher stress concentrations.

The third step was to assign all of the loading and support constraints to the model. The supports in this case were fairly straightforward. The front bumper is rigidly connected to the other structural members by fasteners through holes along three different parallel axes. A fixed support was added on the faces of each of these holes to rigidly fix the bumper to the rest of the vehicle.

The loads for this model were slightly more complicated. Two types of collisions were considered. The first is a "fender bender" type scenario. For this case, the vehicle would be traveling around 3 m/s, a typical speed for slow vehicle operation. The front bumper of the vehicle would then come into contact with an object (wall, tree, etc.), decelerating the vehicle to 0 m/s in roughly 0.2 seconds. The resulting calculated acceleration can then be used to determine the force exerted to achieve this acceleration for the 20 kg vehicle. The acceleration and force were calculated using the following equations, in which *a* is acceleration, *v2* and *v1* are final and initial velocities respectively, *t* is time of collision or impact, *f* is force, and *m* is mass.

$$a = \frac{v_2 - v_1}{t} = \frac{0\frac{m}{s} - 3\frac{m}{s}}{0.2\ s} = -15\frac{m}{s^2}$$

$$F = m * a = 20\ kg * -15\frac{m}{s^2} = 300\ N\ \text{opposite direction of motion}$$

The front bumper experiences a force of 300 N giving it a negative acceleration of 15 m/s2 in a typical slow-moving collision for this vehicle. The second case to consider is a worst-case scenario type condition, in which the vehicle is traveling at its maximum speed of 20 m/s and collides with an object, forcing it to come to rest in only 0.1 seconds. The resulting acceleration and force calculations are shown below.

$$a = \frac{v_2 - v_1}{t} = \frac{0\frac{m}{s} - 20\frac{m}{s}}{0.1\ s} = -200\frac{m}{s^2}$$

$$F = m * a = 20\ kg * -200\frac{m}{s^2} = 4000\ N\ \text{opposite direction of motion}$$

The maximum deformation of the bumper was found to be 5.13 mm, occurring at the two outer tips of the bumper. The maximum stress was found to be 23 MPa, occurring where the bumper is attached to the structural connection. This was to be expected, as the sharp corner should create a slight stress concentration. The corresponding factor of safety at this point was 1.09, the lowest across the part as shown in Fig 4.5 and Fig.4.6

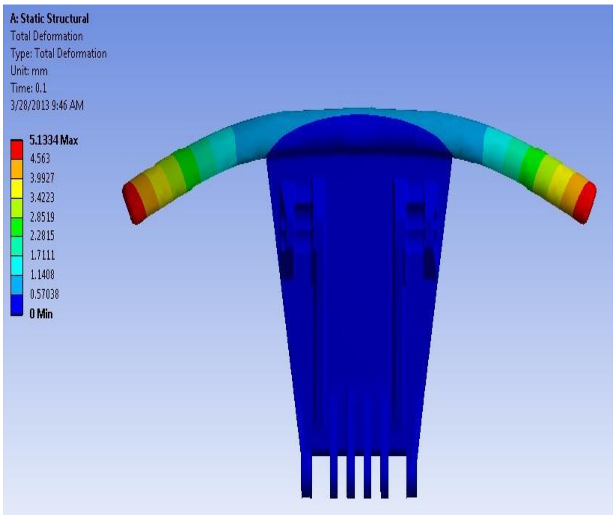

Fig.4.5: Low Speed Case Bumper Deformation (mm)

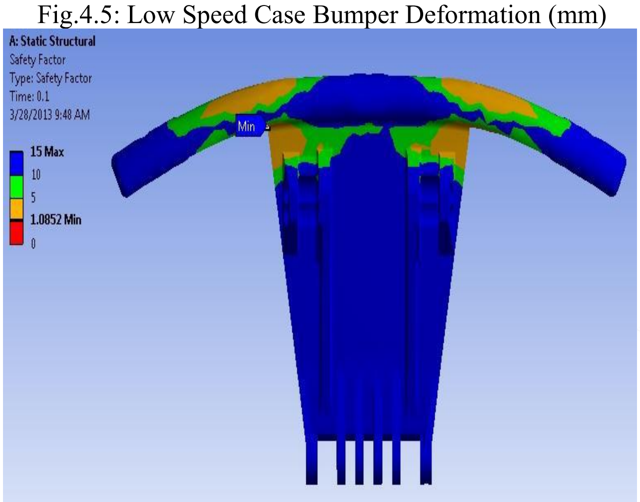

Fig.4.6: Low Speed Case Bumper Factor of Safety

The maximum deformation and minimum factor of safety were the two most important pieces of data. The maximum deformation was found to be 67.4 mm, or nearly 7 cm, and the maximum stress was 302 MPa, giving a minimum factor of safety of 0.08. This is clearly a much greater stress than the previous case, and the low factor of safety suggests that this part will certainly fail. The deformation becomes less of a meaningful result therefore, because this plastic part will fracture before it deforms this much. From these results, we can conclude that the bumper will break if the vehicle hits an immovable object at maximum speed. The bumper should absorb much of the load, so although it would need to be replaced, the rest of the vehicle should be still operable. Figures of the deformation and factor of safety for the high speed case are shown in Figs 4.7 & Fig.4.8below.

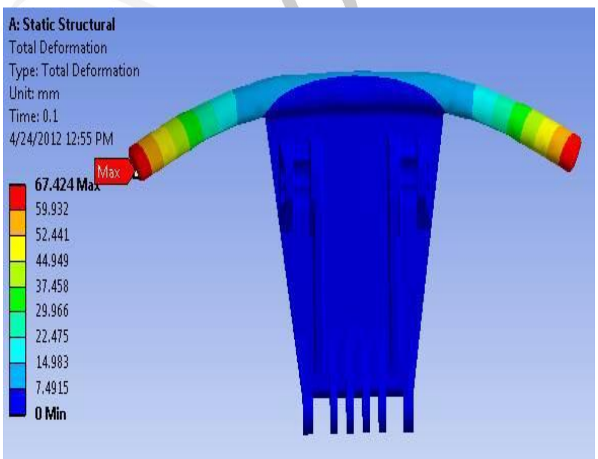

Fig.4.7: Worst Case Scenario Bumper Deformation (mm)

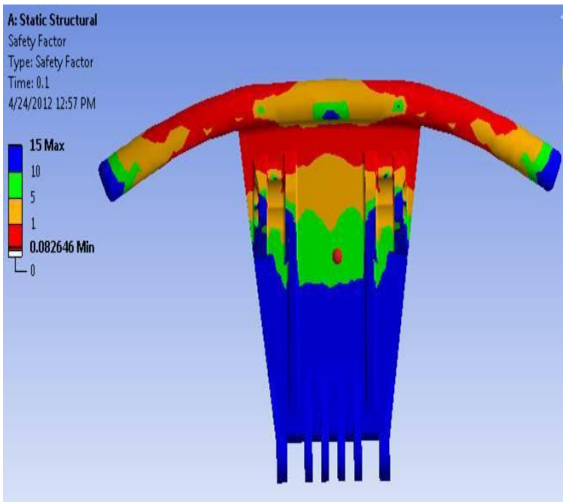

Fig.4.8: Worst Case Scenario Factor of Safety

After studying and attempting to draw conclusions from these results, the next step was to question how reliable these results are. The biggest weakness with this model was the surface across which the collision force was applied. In the original model, the force acts on the entire surface area of the front bumper, as shown in Fig.4.9 below.

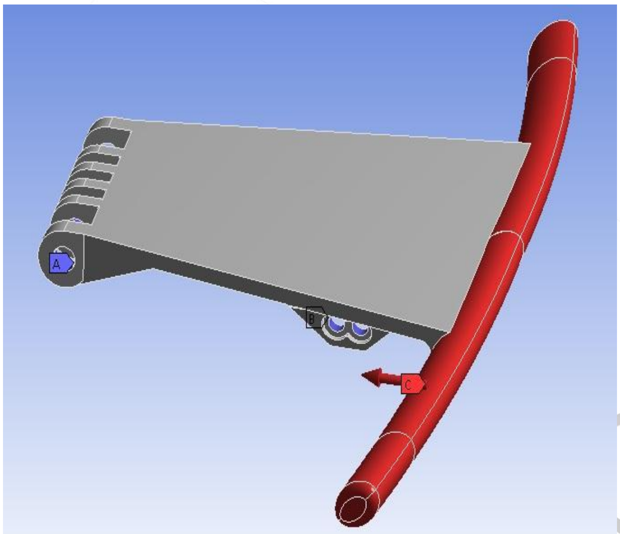

Fig.4.9: Surface Area of Force Applied to Bumper

In reality, a collision force will almost never act this uniformly. Instead, the load is more likely to occur at one point. If the collision is head on and acts at the center of the bumper, the results are likely to be similar. However, a point load at the tip of the bumper would cause much larger shear and moment forces. The model was changed to replace the distributed load with a point load, as shown Fig.4.10 below.

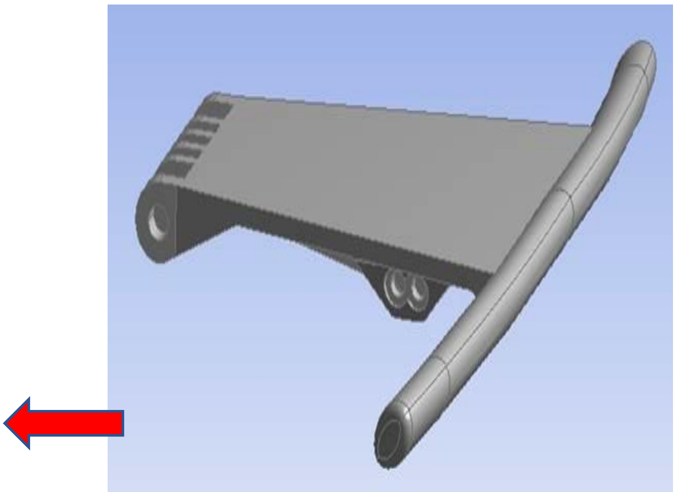

Fig.4.10: Front Bumper with Point Load

When the low speed case was repeated with the load applied as a point load at one end of the bumper, the results show that the bumper would in fact be likely to fail. A figure showing the factor of safety for this case, with all of the red areas having factors of safety of less than one, is shown in Fig 4.11below.

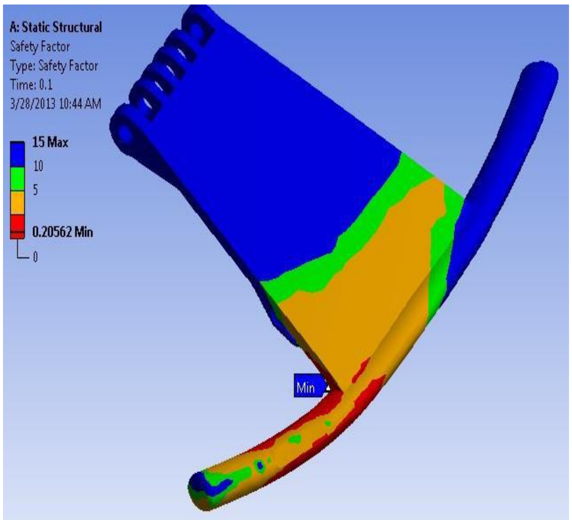

Fig.4.11: Bumper Factor of Safety Results Updated for Impact Point

5.CONCLUSIONS

Out of the three study types (structural, modal, and thermal), modal is the simplest, and requires the least user inputs. From the studies, it can be concluded that a modal study is a quick, cheap, and effective way to ballpark the natural frequency of a part to see if further analysis must be done. Because of the simplicity of the study, incorrect inputs are less of an issue, and a computer may be capable of automatically performing this analysis with confidence placed in the reliably of the results.

The other two studies (structural and thermal), require much more approximations, estimations of loads, and information from outside sources. A human may use intuition and outside research, as well as a basic understanding of the science involved, to solve the problem as close to accurate as possible. A software program may also be able to make such approximations from tables of experimental data. However, as evidenced by these studies, these results are only as accurate as the approximations that produced them. Being able to make design decisions based off analysis results requires analysing more than just a maximum or minimum value, but also locations of data points on a component relative to different geometry features, and an understanding of the accuracy and nature of the results.

At a higher level, the literature studied suggests that one of the modern-day challenges of engineering is solving the problem of automated system level mechanical design. The challenge, and problem that the AVM project is attempting to address, is the difference between isolated component behaviour and component behaviour in a system. The studies performed in this chapter have explored the challenges with making assumptions and approximations in finite element studies, and the importance of putting results into perspective. The studies provided have shown how finite element analysis tools can be used for redesign. The studies have also shown how modularity in components may be achieved, leading to higher level studies performed earlier in the design process, saving both time and money in the design cycle. However, care must be taken to continue to use finite element tools as tools to help guide design decisions rather than as calculators with results automatically taken as fact.

REFERENCES

[1] Adaptive Vehicle Make, DARPA, http://www.darpa.mil/Our_Work/TTO/Programs/Adaptive_Vehicle_Make__%28AVM%29.aspx
[2] ANSYS, Swanson, J. A., ANSYS-Engineering Analysis Systems User's Manual, Swanson Analysis Systems, Inc., Johnson Rd., P.O. Box 65, Houston, PA, 15342.
[3] Archer, J. S., "Consistent Matrix Formulations for Structural Analysis Using Finite-Element Techniques," Journal of the American Institute of Aeronautics and Astronautics, Vol. 3, No. 10, pp. 1910-1918, 1965.
[4] Argyris, J. H., and Kelsey, S., Energy Theorems and Structural Analysis, Butterworths, London, 1960 [collection of papers published in Aircraft Engineering in 1954 and 1955].
[5] Argyris, J. H., "Recent Advances in Matrix Methods of Structural Analysis", Progress in Aeronautical Science, Vol. 4, Pergamon Press, New York, 1964.
[6] Arpaci, Vedat S., and Vedat S. Arpaci. Conduction heat transfer. Reading: Addison-Wesley, 1966.
[7] Baja 5sc, Super Sport, HPI Racing, # 105735 http://www.hpiracing.com/en/kit/105735
[8] Bergquist Company, The, 18930 W. 78th Street, Chanhassen, MN, 55317, http://www.bergquistcompany.com
[9] Clough, R. W., "The Finite Element Method in Plane Stress Analysis", Proceedings, American Society of Civil Engineers, 2nd Conference on Electronic Computation, Pittsburgh, PA, pp. 345-378, Sept. 1960.
[10] Clough, R. W., and Rashid, Y., "Finite Element Analysis of Axisymmetric Solids", Journal of the Engineering Mechanics Division, Proceedings of the American Society of Civil Engineers, Vol. 91, pp. 71-85, Feb. 1965.

[11] "Computer Misuse – Are We Dealing with a Time Bomb? Who is to Blame and What Are We Doing about It? A Panel Discussion", in Forensic Engineering, Proceedings of the First Congress, K. L. Rens (ed.), American Society of Civil Engineers, Reston, VA, 1997, pp. 285-336.
[12] Conover, Dave. "Verification and Validation of FEA Simulations". Integration of Simulation Technology into the Engineering Curriculum. Cornell University. 2008