International Journal of Emerging Technology and Advanced Engineering
Website: www.ijetae.com (ISSN 2250-2459, ISO 9001:2008 Certified Journal, Volume 7, Issue 8, August 2017)

Design and Weight Optimization of Fuel Tank Mounting Bracket for HCV

Dipali T. Bendarkar[1], Prof. Dr. A. D. Desai[2], Prof. A. B. Bhane[3]
[1]PG Student at Shree Ramchandra College of engineering, [2]Professor at Shree Ramchandra College of Engineering, Pune
[3]Asst. Professor at Shree Ramchandra College of Engineering.

***Abstract*-Automobile sector is one of the largest branch of Mechanical Engineering industry. It consumes a lot of fuel while transporting goods and people from one place to other by road. Reducing automobile weight for better economy is the challenge industry faces right now. This work is aimed at on design and weight optimization of HCV fuel tank mounting bracket. Also to find alternative design and to conduct parametric study. It is designed using Taguchi Matrix and the structural and modal Finite Element Analysis is performed using ANSYS software. Numbers of iterations are performed to find out the best possible shape for weight optimization of the bracket. From the optimized module it is find out that 14% weight reduction is obtained.**

***Keywords*-- Mounting bracket, Analysis, FEA, Optimization.**

I. Introduction

Diesel fuel tanks for the truck industry are generally built for the same applications as those for automotive uses but with larger capacity. Brackets are used to hold or support the fuel tank while being mounted on the chassis. Same time tension in the strap is used to keep the tank in position with some stiffness. Main considerations in design of a diesel fuel tank are deciding placement, choosing shape and calculating the required volume. Side mounting is the most common placement of diesel tanks for trucks. This is typically accomplished by using the brackets, straps or a combination of both for the purpose of attaching the fuel tank to the truck frame. Shape is generally decided by the need for maximum capacity and the demand for a stylish look.

A. Design of Fuel Tank

Commonly diesel tank designs are cylindrical, rectangular and D-Style tanks. Cylindrical designs are usually created for their visual appeal while the rectangular tank is mostly used for maximum fuel volume for a available space. The D-Tank, as its name implies, is actually a hybrid of the cylindrical and rectangular designs. It offers the curved visual appeal of a cylindrical tank with significantly more fuel volume. Replacing a cylindrical fuel tank with a D-Tank can result in 46% additional fuel capacity. While calculating volume requirements, one would begin by assessing the available space. Once width, height and length restrictions have been decided, the easiest method of determining volume is using truck tank volume calculator.

Although basic mathematics can be applied to calculate the volume of a cylinder, calculating that of a rectangular tank is more complex due to the rounded corners. Designers should take into consideration the loss of space due to the radius of rounded corners.

B. Construction of Fuel Tank

Material selection is one of the key considerations in producing fuel tanks. Three most common materials utilized in the manufacture of fuel tanks are aluminum, steel and stainless steel. Regardless of the choice of material, the quality of the selection must be such as to allow that material to be malleable enough to be bent, rolled and stamped into formation. Aluminum alloy is a popular choice for fuel tanks as it contains adequate magnesium content to allow the material to be pliable enough to meet the need of the manufacturing process. The selection of steel and stainless steel should be that of prime grade material. An important consideration in manufacturing is choosing material suitable for stamping and bending. The material should be ductile enough so that it can be bent and formed yet thick enough to provide strength and to accept a weld. This is especially true for tanks of a design that require sharp bends. Fuel system tank support system is shown in the image below which is used to support, and hold the fuel tank of the truck in its position. Bracket takes the entire load that is of fuel and tank while being hard mounted on the chassis, while tension in the strap is used to keep the tank in position with some stiffness. Without strap completely hard mounting the bracket will increase the load on the assembly due to gravitational force hugely.

Fig. 1. Fuel tank bracket with straps.

International Journal of Emerging Technology and Advanced Engineering
Website: www.ijetae.com (ISSN 2250-2459, ISO 9001:2008 Certified Journal, Volume 7, Issue 8, August 2017)

II. Literature Survey

Various people have worked for minimizing the cost of material by reducing material with optimum design, or using different types of material. Authors Gajendra G. [1] works on finite element analysis of engine bracket of car and natural frequency will be determined. Engine bracket has been designed as a framework to support engine. Matteo Bruggi [2] observe that stress constrained topology optimization is an effective tool to investigate layouts that are fully feasible with respect to the strength of the material or any prescribed requirement involving the stress field. Hemanth R. [3] reduced the material cost by 15% in two wheeler products. This has been attempted using a job plan approach from Value Analysis technique. He replaced fuel tank adaptor by copper brazing there by 10% material cost was reduced and mounting brackets count was reduced from 3 to 1 number, there by 8% of material cost was reduced. Cumulatively 18% of material cost was reduced with annual savings of around Rs. 50 lakhs to the organization. Kennerly Digges [4] conduct Eighteen full-scale tests were performed on six alternatives to the sidesaddle tanks on 1973 to 1987 GM C/K pickup trucks. The critical test configuration was an 80.0 km/h (50 mph) lateral impact from a Chevrolet Caprice. Two alternatives tank systems were selected for further test and evaluation. The evaluation included FMVSS 301frontal, rear and lateral type tests conducted at higher severity than required by FMVSS 301.Himeki. H et.al. [5] was analyzed fatigue behavior of the high-dendity polyethylene applied to fuel tanks, under low-level cyclic loading that simulated fuel tank pressure changes. The correlation between fatigue life and stress, temperature and frequency(the major influencing factors) was expressed quantitatively using fatigue test data for test pieces. This formulation was then verified in fatigue test conducted on plastic fuel tanks. The validity of this equation for predicting the fatigue life of plastic fuel tanks was thus confirmed.

III. Mathematical Calculations

Design of fuel tank

In this study we have chosen the TATA LPK 2518 fuel tank bracket for design and optimization. This has fuel capacity of 225 liters. Choosing a appropriate design for fuel tank is the first step towards selection of best design of supporting brackets. Standard inner diameter of fuel tank for truck is 28 inches. By converting in to metric unit, diameter of tank selected will be 457.2 mm. Following table number I is showing the result of mathematical in which serial number 4 result is optimize.

TABLE 1.
Mathematical Calculations.

Sr. No	Total Mass (kg)	Load (N)	L1 (mm)	σ (allow stress)	d (depth) (mm)	b (width) (mm)	ρ (kg/mm^3)
1	235	768.45	460	185	3	318.4	0.00000785
2	235	768.45	460	185	5	114.6	0.00000785
3	235	768.45	460	185	6	79.61	0.00000785
4	235	768.45	460	185	7	58.4	0.00000785
5	235	768.45	460	185	8	44.7	0.00000785
6	235	768.45	460	185	9	35.3	0.00000785
7	235	768.45	460	185	10	28.6	0.00000785

IV. Modelling And Analysis

CAD model for fuel tank bracket:

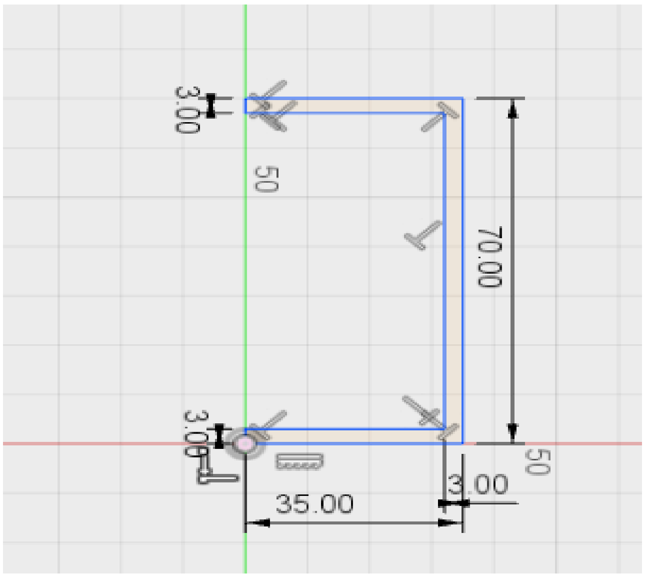

Fig .2. Cross section of the bars used for modeling

A. Modal Analysis

Modal analysis is performed to find out the vibrational frequencies and the mode shapes of the component at the given boundary condition. Fuel tank bracket is directly placed on the chassis and bolted to it so all the chassis vibrations directly transferred to the bracket. So it is important to understand the vibrational frequencies of the bracket and nature of its mode shape, and it should also be verified that modification in the design doesn't adversely affects the results of the bracket vibrational analysis.

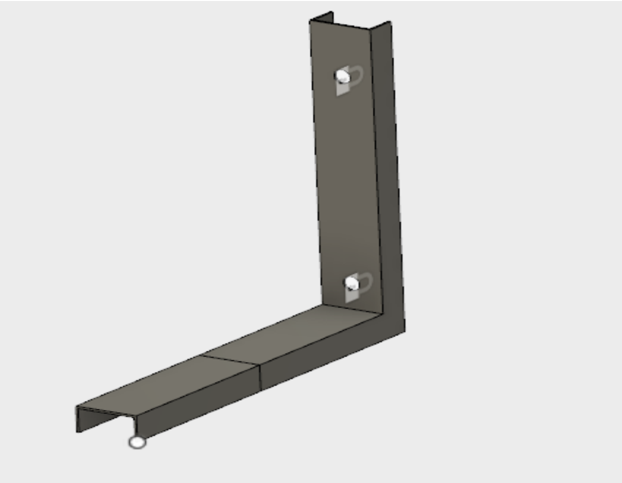

Fig.3. L CAD model of the Bracket

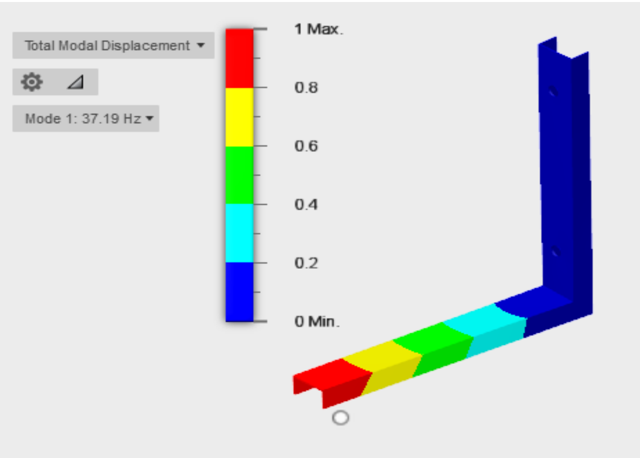

Fig.4. Mode shape plot at first frequency 37.2 Hz

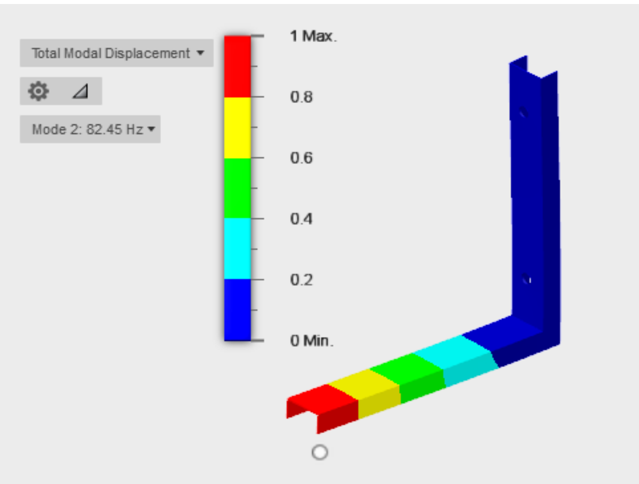

Fig.5. Mode shape plot at second natural frequency 82.45 Hz.

Figures 4 and 5 shows the first two mode shapes and their natural frequencies for the baseline model of L bracket for the support truck fuel tank under study.

B. Static Analysis

For static analysis, the load per bracket is to be considered and in the present case, with the use of 3 brackets and a total load of 225 kg, the load per bracket will be 75 kg (735.75N) downword.

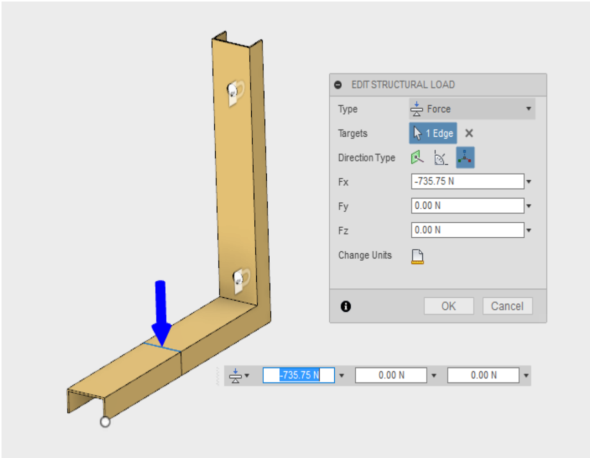

Fig.6. Loading condition and boundary condition of the bracket.

Fixed in all direction boundary condition is applied to the bolt holes of the bracket which are on the vertical flank shown by the lock symbols in Fig.6 with these boundary conditions following results are observed for stress, deformation and strain as shown in Fig.7 below.

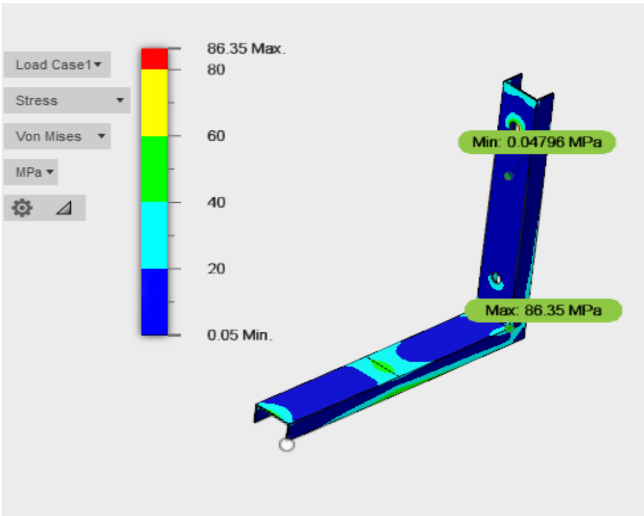

Fig.7. von Mises stress plot at static analysis.

High stress of 86.35 MPa is observed at the corner. It is within the acceptable stress limit of the steel used for the design.

International Journal of Emerging Technology and Advanced Engineering
Website: www.ijetae.com (ISSN 2250-2459, ISO 9001:2008 Certified Journal, Volume 7, Issue 8, August 2017)

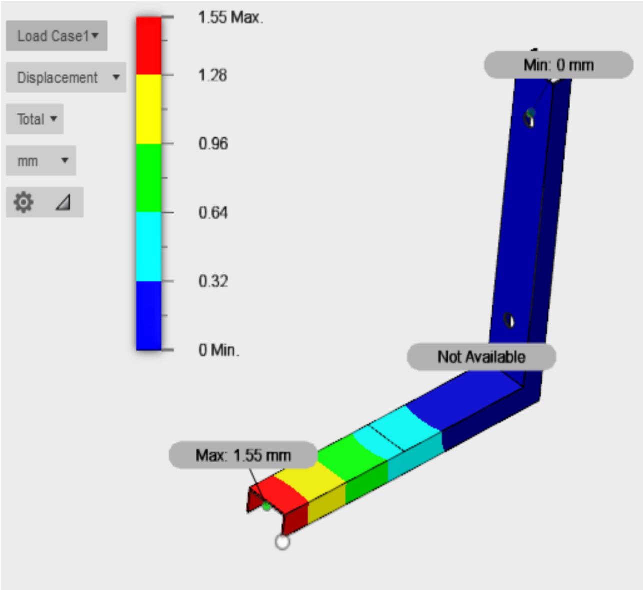

Fig.8. Deformation plot at static analysis (mm)

Total deformation of 2 mm is observed at the end of the overhanging horizontal flank of the bracket. Strain plot Fig. 9, shows that total maximum strain of 0.00158 is observed at the corner of the bracket. Further optimization module is completed for the bracket to find out which is insignificant to the load transfer.

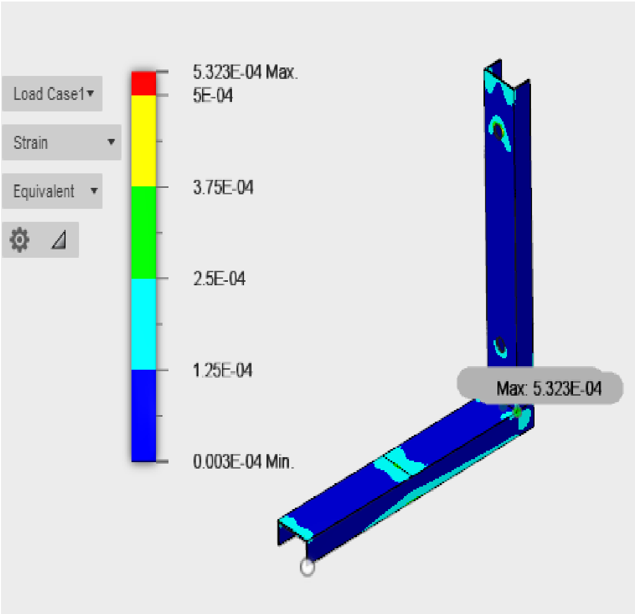

Fig.9. strain plot at the static analysis of bracket

C.Shape Optimization

With the same boundary conditions as static analysis are used for component and optimization module is obtained with FEA-FUSION 360 Software. This plot in Fig. 9 shows us the material which is important for load carrying capacity of the design. Part in blue shows insignificant part while red is the most important part of the component. We can perform optimization iterations by removing the materials from blue zone.

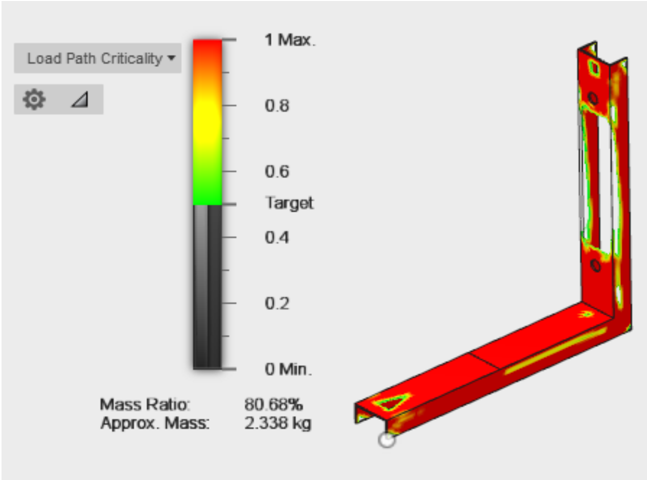

Fig.10.Load path criticality path plot

Decision of the material removal iterations is made based on requirement of the adjacent components. Weight of the baseline model is 2.89 Kg.

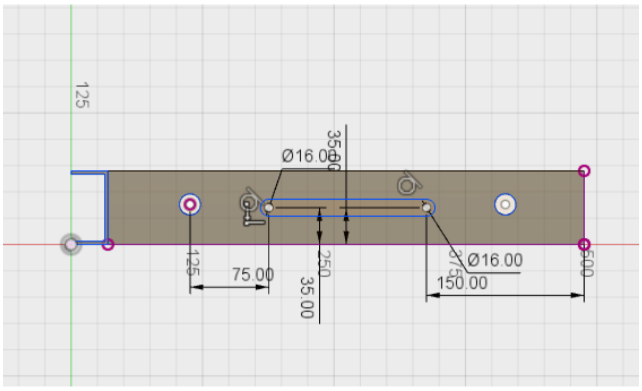

Fig.11. Sketch for Modifications in Iteration 1

Modifications shown in the image above are done for iteration 1 according to the load criticality path diagram.

International Journal of Emerging Technology and Advanced Engineering
Website: www.ijetae.com (ISSN 2250-2459, ISO 9001:2008 Certified Journal, Volume 7, Issue 8, August 2017)

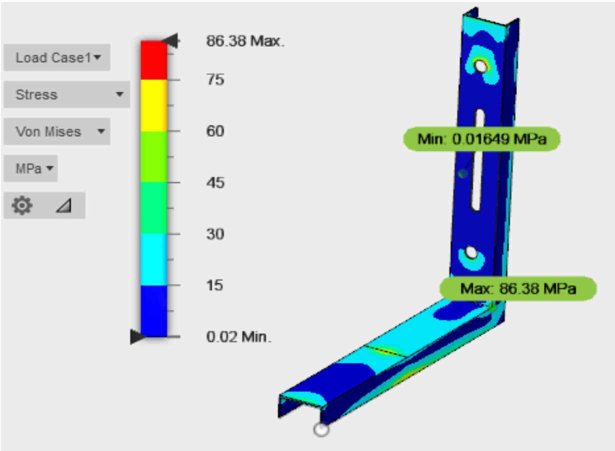

Fig.12.Iteration 1 Stress plot (Mpa).

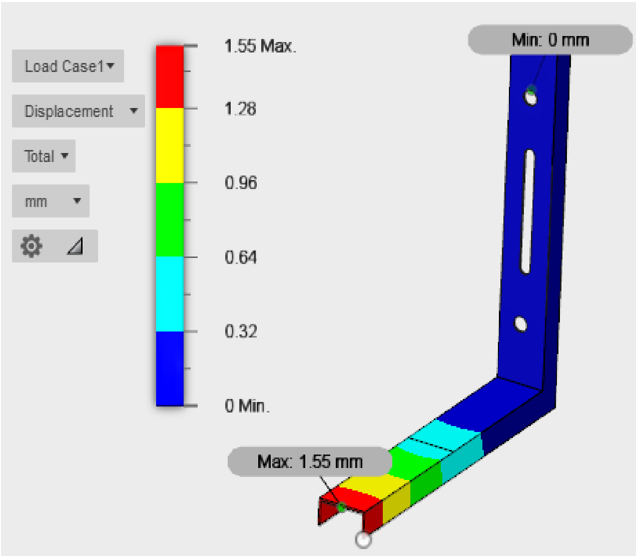

Fig.13. Iteration 1 deformation plot

Mass of the iteration 1 design is 2837 grams shows in Fig.13 which is only 60 grams less than the original design. Again shape optimization module is used on the iteration 1 design and analysis is performed to find out the material removal scope in the iteration 1 model. Results for the load criticality path are shown in the image Fig. 14.

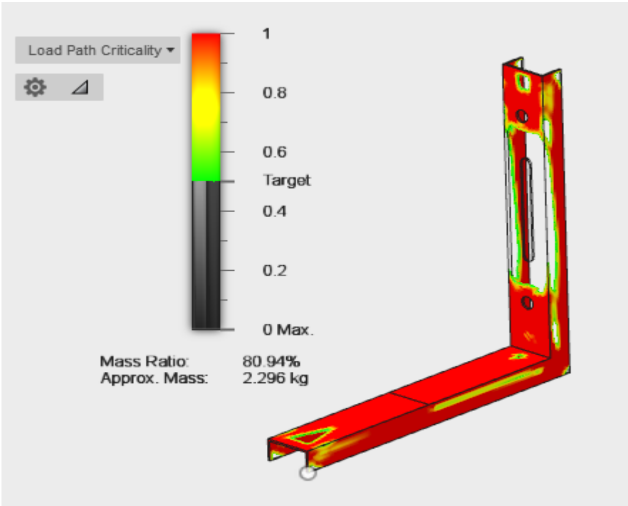

Fig.14. Load Criticality path for Iteration 1

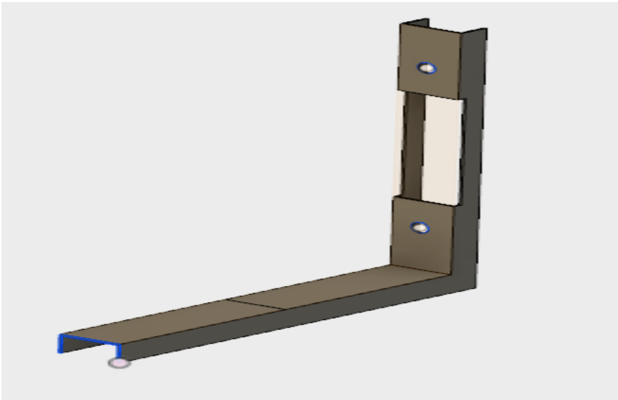

Fig.15. CAD model for Iteration 2 design

Iteration 2 model is created again based on the load criticality path of the optimization module from baseline analysis. Above in Fig. 15 shown the sketch and model for iteration2 design.

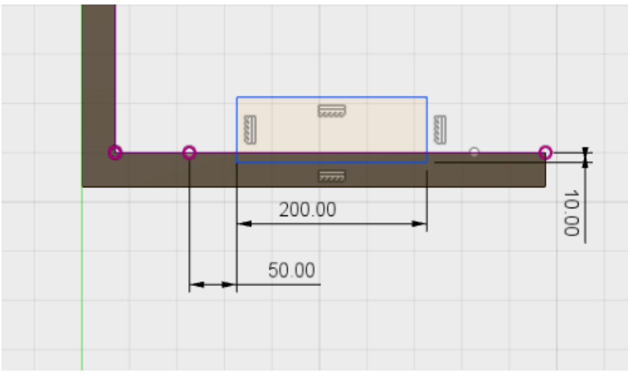

Fig.16.Sketch for modification of L bracket Iteration 2 design

International Journal of Emerging Technology and Advanced Engineering
Website: www.ijetae.com (ISSN 2250-2459, ISO 9001:2008 Certified Journal, Volume 7, Issue 8, August 2017)

Modifications shown in the image above are done for iteration 2 according to the load criticality path diagram.

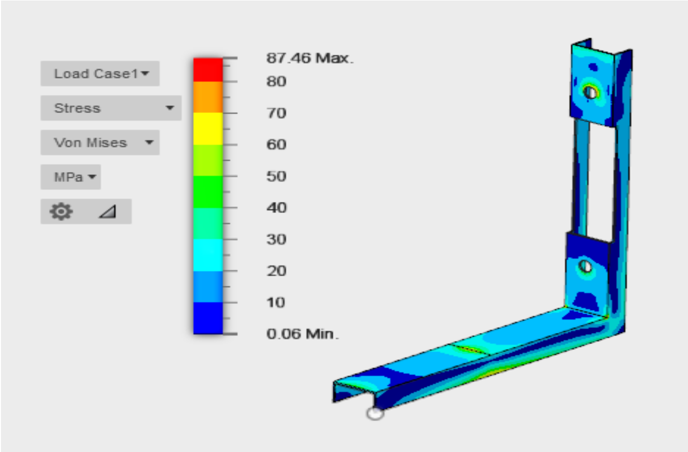

Fig.17. Von Mises stress plot Iteration 2

In Fig.17 Maximum von mises stress plot observed at the corner is 87.5 MPa which is still within he acceptance limit of the bracket.

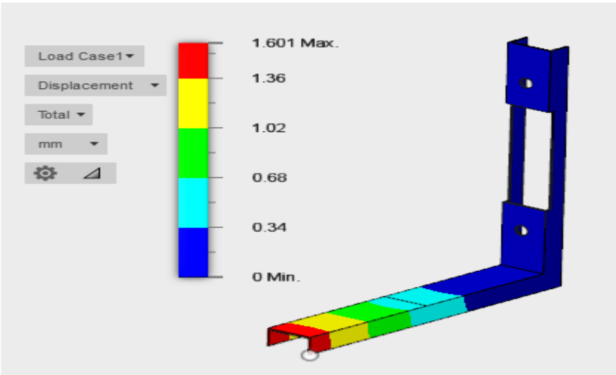

Fig.18. Total deformation plot Iteration 2

Not much change is observed in FEA results of stress and deformation in iteration 2 when compared with previous iteration. While weight is reduced due to slot at the supporting bracket. Weight of the iteration 2 model is observed as 2502 grams. Total of 395 grams weight reduction in the model is achieved.

Modal analysis is also performed on the iteration 2 model to verify the impact on the frequencies.

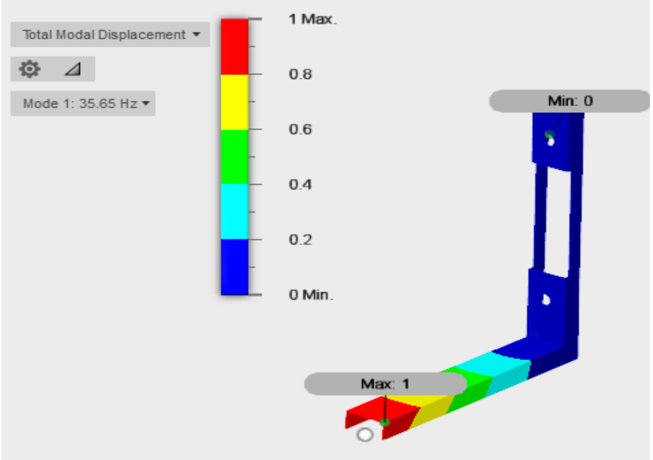

Fig.19. Mode shape plot iteration 2 model mode 1

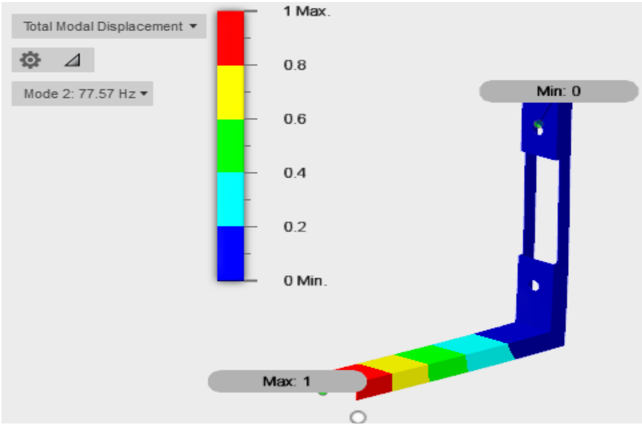

Fig.20. Mode shape plot iteration 2 model mode 2

Iteration 2 model has first natural frequency as 35.65 Hz and second natural frequency as 77.57 Hz with modes those of similar as baseline model.

With similar load fatigue analysis is ran on the model using ANSYS software with same boundary conditions and assuming the fully reversible loads. Baseline and final iteration are ran for fatigue analysis. Results for the same are given below.

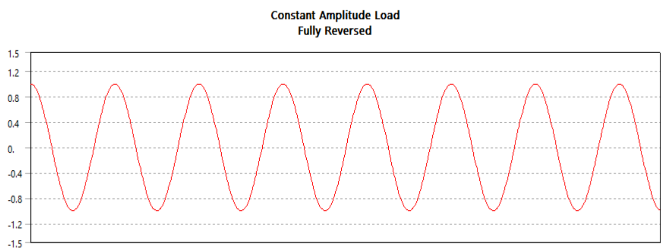

Fig.21. Loading for fatigue analysis

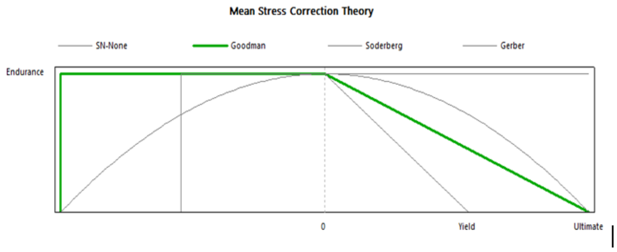

Fig.22. Mean correction Good Plot

Above two conditions are applied with standard boundary conditions which were applied in static analysis of loading and fixing, again shown in the images below for ANSYS software.

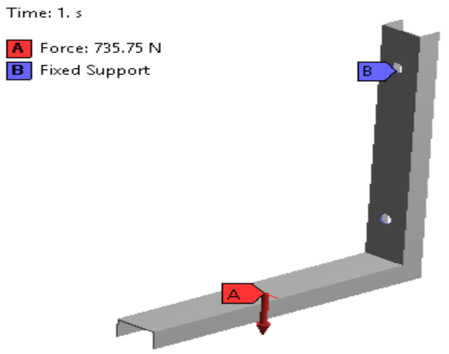

Fig.23. Loading of 735.75 N on the side and fixed support at holes

Fatigue analysis is performed with these loading conditions are results for the analysis are as given below for the baseline geometry.

Minimum life of 1.58 10^6 cycles is observed that is also at the corners due to stress concentration. Baseline model is safe in fatigue.

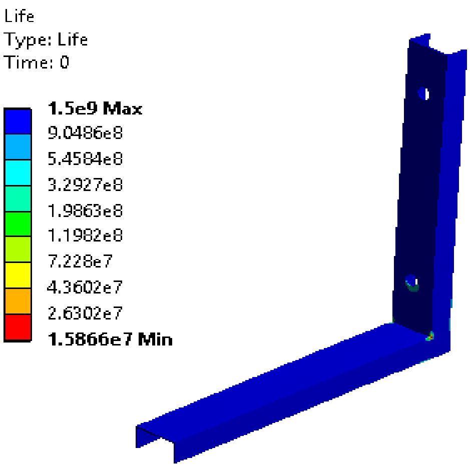

Fig.24. Life plot for fatigue of baseline

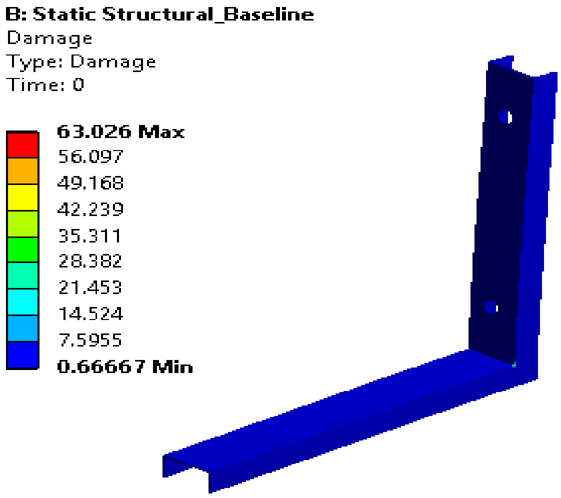

Fig.25. Damage plot of baseline

At almost all the places damage is below 1 which means component has infinite life in fatigue under given loading conditions. At the corners it shows high damage due to singularity of stress concentration.

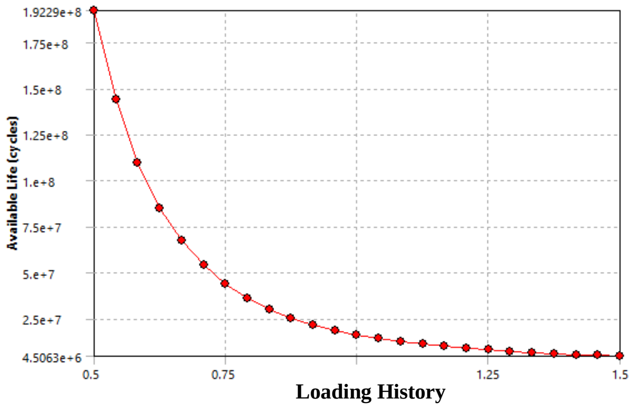

Fig.26. Fatigue sensitivity graph for baseline

Iteration 3 results for the fatigue analysis are shown below.

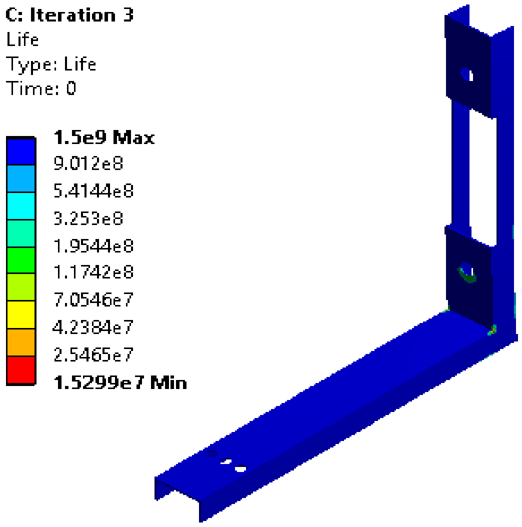

Fig.27. Life plot at iteration, 3

Life plot at iteration 3 shows minimum of 1.52 X 10^7 cycles of life.

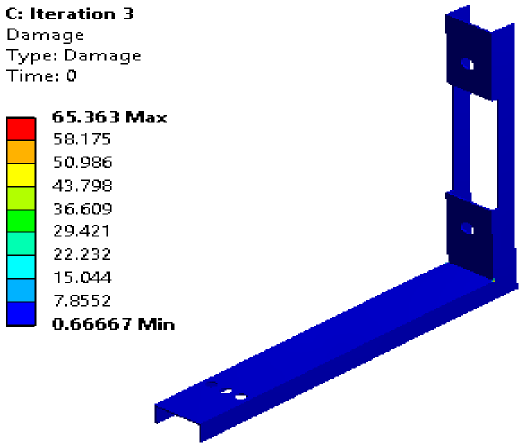

Fig.28.Damage plot iteration, 3

Iteration 3 model damage plot shows results similar to baseline damage plot, damage is below 1 at almost all the part of component.

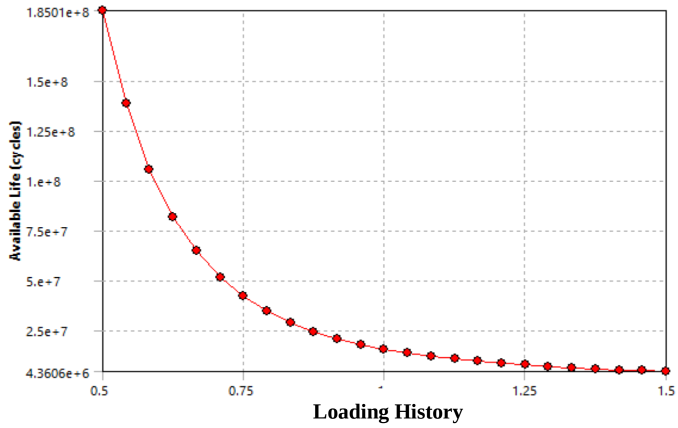

Fig.29. Fatigue sensitivity plot iteration 3

V. Experimental Validation Plan

The basic model of the fuel tank mounting bracket currently in use is as shown in the below image. The manufacturing of the optimized bracket will be done on the basis of the results obtained from the FEA.

Fig.30. Basic model of the fuel tank mounting bracket

V. Results And Discussion

A summary of FEA result is shown in the Table 2 below modification of the support bracket for fuel tank of TATA 2518.

TABLE 2.
Summary of FEA result.

Design	Stress (MPa)	Frequency 1	Frequenc y 2	Weight (kg)	Weight Reductio n
Baseline	86.35	37.2	82.5	2.898	0
Iteration 1	86.38	37.2	82.2	2.837	2.1%
Iteration 2	87.46	35.65	77.6	2.502	13.7%

Minimum fatigue life of the component in the fatigue analysis performed using ANSYS is observed as 1.58X 10^7 cycles for baseline model and 1.52 X 10^7 cycles for the iteration 3 model.

VII. CONCLUSION

Fuel tank supporting bracket for TATA 2518 truck was designed and modeled using plane C section channels. Baseline model was created using modeling software and finite element analysis was performed to find out the maximum stresses and deformation for the analysis. Baseline stresses are found to be well within the acceptance criteria of design. Modal analysis was also performed on the baseline model to find out first two modal frequencies and mode shape plots. Optimization FEA module was used for finding out possible material removal regions and approximately 14 % materials was reduced using different design iteration taking the guidelines from load criticality path. Life of iteration 3 component is almost as good as baseline. Life of the component is not affected by much due to geometric changes and material removal according to fatigue analysis performed using ANSYS.

REFERENCES

[1] Gajendra G. 2015, “Design and Optimization of HTV Fuel Tank Assembly by Finite Element Analysis, IRJET,2015, PP: 43-50.

[2] Matteo Bruggi 2015, “Topology optimization with mixed finite elements on finite elements on regular grids” ELSEVIER, pp:133-153.

[3] R. Hemanth 2013, “Reducing Material Cost in Fabricated Parts for Two Wheelers Through Value Analysis Technique” M.S.R. pp43-50.

[4] Kennerly Digges 2003, “Research to Evaluate Safety Technologies for Vulnerable Fuel Tanks”, pp:1-14.

[5] Hiroaki Himeki 2006, “ Fatigue Behavior Analysis and Durability Evaluation of Plastic Fuel Tank” SAE pp:1-7.

[6] S. Shojaee 2012, “ Structural Topology Optimization using an enhanced level set method” ScientiaIranica, pp: 1157-1167.

[7] James I Chang 2006, “A Study of storage tank accidents” ELSEVIER, pp:51-59.

[8] Jordan C. et.al. 1990, “ Fuel Tank Heat Shield Design”, SAE, pp:1-15.

[9] Rakheja S. et.al. 1989, “Influence of Tank Design Factors on the Rollover Threshold of the Partially Filled Tank Vehicles”, SAE, pp:1-15.

[10] T.B. Jones 1962, “Minimum Weight Design of Tank Structures”, SAE, pp:3-16.

[11] Shivakumar M.M et.al. 2012, “Fatigue Life Estimation of Chassis Frame FESM Bracket for Commercial Vehicle”, IJSR, pp:441-447.

[12] Mr.Mohd et.al. 2015, “Design and Analysis of Fuel Tank A-Review” IJRITCC, pp:38-39.

[13] H. V. Chavan, 2015,”Design and Analysis of Plastic Fuel Tank for Three Wheeler CNG Passenger Vehicle”, IJRITCC, pp:3976-3981.

[14] Paul S. Griffin et.al. 2003, “Design and Manufacture of a Lightweight Fuel Assembly”, AIAA, pp:1-20.

[15] Rajamani Rajagouder et.al. 2016, “A study of Liquid Sloshig in an Automotiove Fuel Tank under Uniform Acceleration”, Egineering Joural, pp:71-85.