Finite Element Analysis of Fuel Tank Mounting Bracket

Abhishek P [1], Hardeep S M [2], Sarvocch G [3], Anil L[4]

[1]CDGI, Indore
[2]IIST, Indore
[3,4]IICAE, Indore

Abstract - The fuel tank is used in automobile industries to carry fuel and supply fuel to the engine. The aim of this research work is to optimize the design of fuel tank mounting bracket for commercial vehicles. The project includes the geometry and finite element modeling of fuel tank mounting bracket design. Geometrical modeling was done using Pro-E; finite element modeling using Hypermesh software and analysis was done using Optistruct solver. This project deals with the static analysis of bracket assembly. This project provides a methodology for analysis of an assembly consisting of components made of metal components.

Key Words: **FEA, Fuel Tank, Hypermesh , Optistruct**

INTRODUCTION

It is said that in early days buoyancy in vehicle sales in India is heartening for the industry and economy. It caused to sever competition in Automobile industry, Market rate challenges to automobile industries. So it becomes a necessary strategy for each and every company to accept this challenge and to make a room in the automobile sector. Cost of the vehicle is at the core point of view. So it's become an important objective of each company is that to pay their attention on the market price rate and should attempt to minimize the price of the vehicle. It found that each and every part of engine, vehicle body and its necessary accessories and equipment's, are the assessors and they are directly affecting the total price of the vehicle. There is a chance to minimize the cost of the vehicle through its designing aspects this overall assessment taking into consideration and it is decided that there is a chance and these needs of reassessment of three wheeler vehicle in connection to reduce its price. The fuel tank mounting brackets are responsible for carrying the fuel tank hence its design is very critical for safety point of view. Fig- 1 shows typical fuel tank and mounting bracket of truck.

Fig -1: fuel tank and mounting bracket of truck

Umesh et al have studied about the finite element analysis of fuel tank mounting bracket [1]. Fuel tank mounting bracket has been designed as a framework to support Fuel Tank. The main concern is for static stress for bracket which may lead to structural failure if resulting vibration and stress are excessive. Tatsuo et al. also performed the similar type of study [2]. The method and structure for mounting fuel tank improves the efficiency of assembling work, certainty of piping and workability can be attempted. The fuel tank is placed on a tank supporting frame and tightened to said frame by a belt. For pipe and seal installation pipe and leak preventing seals are assembled with the tank. For frame fixing frame are fixed on a car body and tank is fixed on the frame.

This arrangement reduces the unnecessary time hour and labor to install each fuel one by one on the car body. The study performed by Pavan B. et al., explains the process of optimization of natural frequency of engine bracket by finite element analysis by use of different lightweight materials [3]. The strategy of increasing lightweight material in vehicle has proven to be successful method of achieving fuel economy and environmental concepts. Evaluation of engine mount bracket assembly was performed using FEA and modal analysis technique from the result it was found that bracket manufactured with Mg alloy gives optimized frequency. Dhillon and group studied similar phenomena [4]. In an automotive vehicle the engine rest on bracket which are connected to the main frame or chassis of the body. The engine mount plays an important role in vehicle. Correct geometry and positioning of the mount bracket gives a good ride quality and performance. This paper discusses the modeling, finite element analysis, modal analysis and mass optimization of engine bracket foe FSAE car.

Since the FSAE car are high performance vehicle brackets tends to undergo continuous vibration so fatigue strength and durability calculations also have been done to ensure engine safety. Michael Davis studied about the design of high speed vehicle fuel tank [5]. The design of fuel tank for high speed vehicle is not a simple task. There are numerous physical factors that need to be considered while designing an effective fuel tank. This paper discuss about some different models that have been proposed for sloshing in a fuel tank and it also examine the overall geometry of the fuel tank to gauge the volume of fuel remaining in the fuel tank.

METHOD

At first the geometry of the bracket is prepared. The overall purpose of fuel tank bracket is to support the fuel tank and sustain the loads by fuel tank as well as chassis from tires due to uneven road surfaces. The key areas for modification are identified. The main task in this study is to design the bracket with sufficient strength and optimizing it for various design modification. The 3-Dimensional model of fuel tank bracket is prepared. Different design modification is done and analysis is carried out using finite element analysis software named Hypermesh and Optistruct.

Fig- 2 shows the computer aided design of base model fuel tank mount bracket and fig- 3 shows the meshed model for the bracket. This bracket has been assigned to various design modification by adding stiffeners to base design. FINITE ELEMENT ANALYSIS is carried out by using Hypermesh and Optistruct.

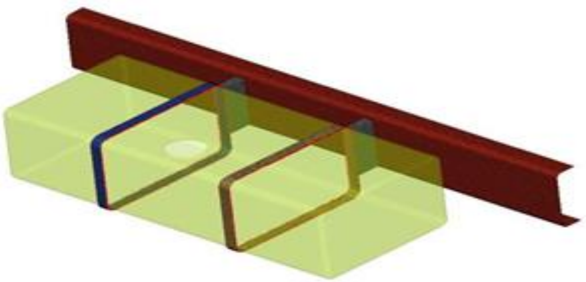


Fig -2: Meshed Model of Fuel Tank Assembly

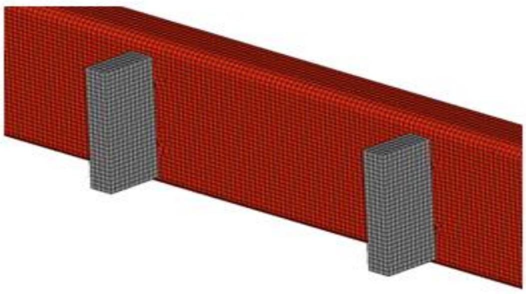


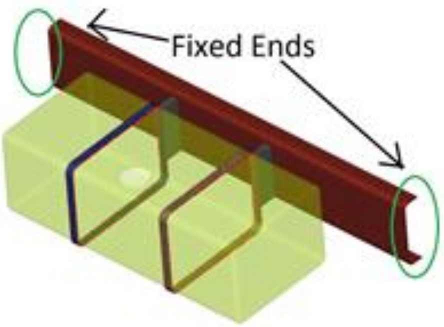
Fig -3 : Meshed Model of Fuel Tank Bracket

The steel mechanical properties were used for this analysis as shown in Table 1.

Table -1 Material description (Steel BSK46)

Parameters	Values
Density	7.8E-9 Tons/mm3
Elastic Modulus	2.1E5 MPa
Poisson's ratio	0.3
Yield Stress	255 MPa

The bracket is fixed on all degree of freedom from the mounting position as shown in fig- 4.



In case of speed barkers, maximum 3g (gravitational load) can be experienced by the fuel tank. Similar load was applied in the fuel tank assemply to simulate the real world situation as shown in fig- 5.

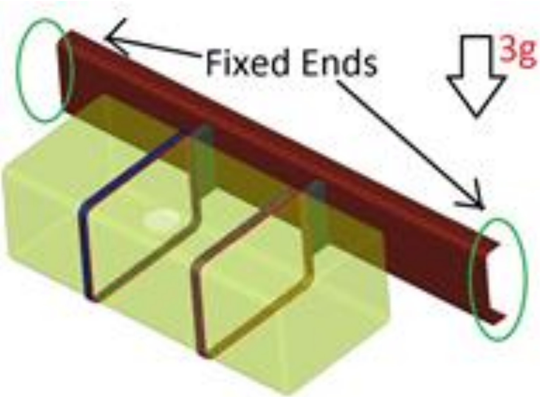


Fig 5: Loading Conditions

RESULTS

Fig- 6 shows the overall displacement of the baseline case. The maximum displacement was measured as 2.7mm.

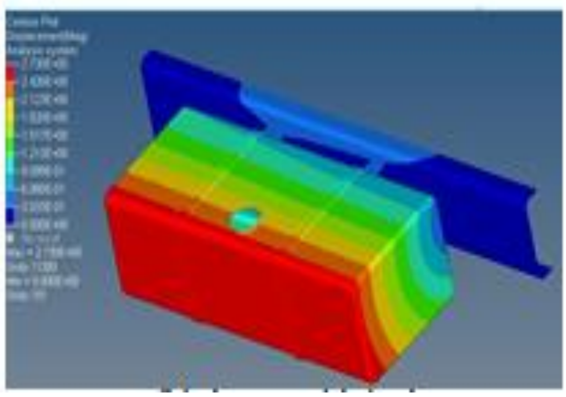


Fig 6: overall displacement of the baseline case

Fig- 7 shows the overall stress of the baseline case. The maximum stresses was measured as 20 MPa.

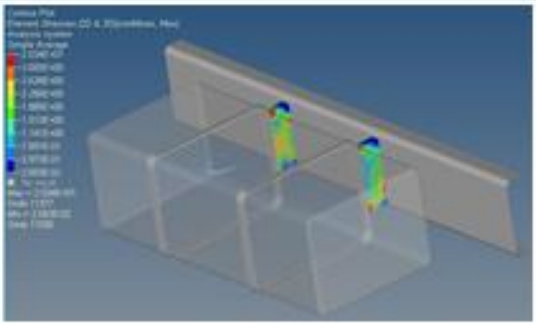


Fig 7: overall stresses of the baseline case

Fig- 8 shows the overall displacement of the modified design case. The maximum displacement was measured as 2.74mm

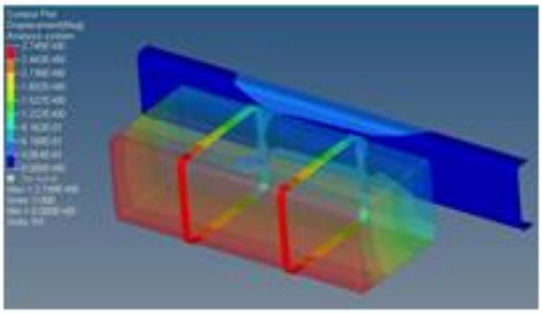


Fig- 8: overall displacement of the modified case

Fig- 9 shows the overall stress of the modified design case. The maximum stress was measured as

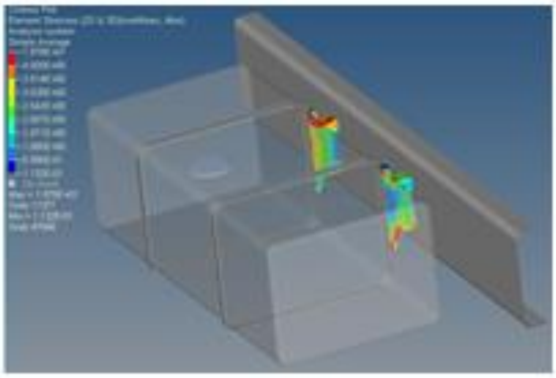


Fig- 9: overall stresses of the modified case

CONCLUSIONS AND DISCUSSION

Table 2 shows the results summary and calculated FOS (Factor of safety). The modified design is showing sufficient FOS i.e. greater than 1.

Table 2: Results Summary

	Maximum Stress	Yield Stress	FOS	Total Weight
Baseline Design	20	255	12.75	17.6 kg
Modified Design	19.5	255	13	13 kg

Overall 26 % weight saving was achieved without compromising the strength of the brackets. The complete assembly of fuel tank was designed, analyzed and optimized successfully.

REFERENCES

1.Umesh s ghorpade, Mahendra gaikwad " FEA and natural frequency optimization of engine bracket" IJMIE, Vol-2, Iss-3, 2012.

2.Tatsuo kasuga, Eisei higuchi, "method and structure for mounting a fuel tank" US5794979 A.

3.Pavan B. chaudhari, "comparison of Mg, Al and cast iron to obtain optimum frequency of engine bracket using FEA" IJERA, Vol-2, sep-oct 2012.

4.Jasvir Singh Dhillon, Priyanka Rao,"design of engine mount bracket for FSAE car using finite element analysis" IJERA, Vol-4, Iss-9(version 6), September 2014.

5.Michael Davis, Zhingxing fu, Qunhui Han, "optimizing fuel tank design for high speed vehicles "June 9-12 2009.