DESIGN AND OPTIMIZATION OF HTV FUEL TANK ASSEMBLY BY FINITE ELEMENT ANALYSIS

GAJENDRA G[1], PRAKASHA A M[2], DR NOOR AHMED R[3], DR.K.S.BADRINARAYAN[4]

[1]PG Scholar, Mechanical department, M S Engineering College, Bangalore, Karnataka, India
[2]Assistant prof, Mechanical department, M S Engineering College, Bangalore, Karnataka, India
[3]Professor and Hod, Mechanical department, M S Engineering College, Bangalore, Karnataka, India
[4]Principal, Mechanical department, M S Engineering College, Bangalore, Karnataka, India

Abstract : The present study is a methodology to improve the first natural frequency of a fuel tank brackets for heavy duty vehicles using FEA. To improve the performance of the fuel tank bracket series of design iteration was carried out by taking the account of base model structure. Normal modal analysis for base model is carried out to find the first natural frequency. Normal modal analysis was carried for all the design iteration to improve the first natural frequency of the fuel tank bracket. Static analysis is carried out for all modified designs to find out the Maximum displacement and von misses stress at critical location. Maximum principle stress and minimum principle stresses are carried out.

Keywords : Bracket stiffener, Fuel tank, Optimization, Modal Analysis, FEA

1. INTRODUCTION

The first heavy duty trucks were developed in United States in the late 1890s. During World War 1heavy duty truck played an important role in moving supplies at home and overseas. Fuel tanks in the heavy duty trucks were made with steel because of high strength and durability. Fuel tank is a safe container for flammable fluids.

The fuel system of automobile vehicles should perform within major safety parameters related to the importance of flammable substances such as diesel fuels which is extensively consumed worldwide.

Important consideration in designing a fuel tank are determining placement choosing the shape and determining the required volume. The fuel system of automobile chassis body system may undergo undesirable vibration due to disturbance from road and fuel tank system. In order to control the road induced vibration the fuel tank bracket should be stiff and damped. Fuel tank mounting is accomplished with use of brackets, straps or a combination of both for the purpose of attaching the fuel tank to the truck frame.

Let us consider an example of high speed vehicle boat, at high speeds the sloshing that occur in the tank can drastically affect center of gravity of the vehicle, depending on the size of the fuel tank severity of the sloshing can negatively affect a control system. The forces that act on the wall of the tank can also reduce the integrity of the tank. By considering these guidelines we are going to examine the overall geometry of a fuel tank and designing the most effective fuel tank for a given vehicle. This paper mainly focuses on finite element analysis of fuel tank bracket for optimizing natural frequency by use of different bracket stiffeners.

2. LITERATURE SURVEY

[1] This paper deals 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. The main concern is for vibration and fatigue of engine bracket which may lead to structural failure if resulting vibration and stress are excessive.

[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.

[3] This paper explains the process of optimization of natural frequency of engine bracket by finite element analysis by use of different lightweight materials. 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.

[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.

[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.

3. PROPOSED METHODOLOGY

At first the theoretical study of bracket is done. The overall purpose of fuel tank bracket is to support the fuel tank and sustain the vibrations caused 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 tune the natural frequency of bracket by optimizing it for various design modification. The 3-Dimensional model of fuel tank bracket is prepared. Different design modification are done and analysis is carried out using finite element analysis software named Ansys Inc. Best design modification is selected by comparing with various design modification brackets.

4. OBJECTIVE

To do static structural and modal analysis of fuel tank mounting bracket for different design modification and suggest best design for the bracket.

5. ANALYSIS OF FUEL TANK MOUNT BRACKET

Figure 1 shows the computer aided design of base model fuel tank mount bracket and figure 2 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 ANSYS software.

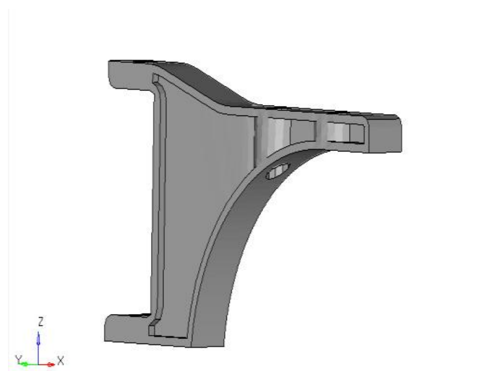

FIG-1 CAD model of fuel tank bracket

FIG-2 Meshed model of fuel tank bracket

Table -1 Material description

MECHANICAL PROPERTIES	Steel
YOUNGS MODULUS(E)	$2.1E^5$ N/mm^2
DENSITY(ρ)	7.89 Ton/mm^3
POISSON'S RATIO(ν)	0.3

6. FINITE ELEMENT ANALYSIS

Finite element analysis (FEA) is a powerful engineering tool that can solve many kinds of engineering problems to as high degree of precision as necessary. In essence, the finite element is a mathematical method for solving ordinary & partial differential equations

FEM is a computational technique used to obtain approximate solutions of boundary value problems in Engineering. This involves deciding what parts are important and what unnecessary detail can be omitted i.e. disregard any small geometric irregularities; consider load as concentrated, homogenized composite material properties. Then we choose the theory which best describes the behavior of the model such as the behavior best described by beam theory, plate-bending theory, plane elasticity, plane strain or plane stress formulations. When modeling something in FEA attention must be paid to what you are actually trying to achieve.

In this analysis, a complex region defining a continuum is discretized into simple geometric shapes called finite elements. The material properties and the governing relationships are considered over the elements and expressed in terms of unknown values at element corners called as nodes.

Figure 3 describes the boundary condition of the bracket for the analysis

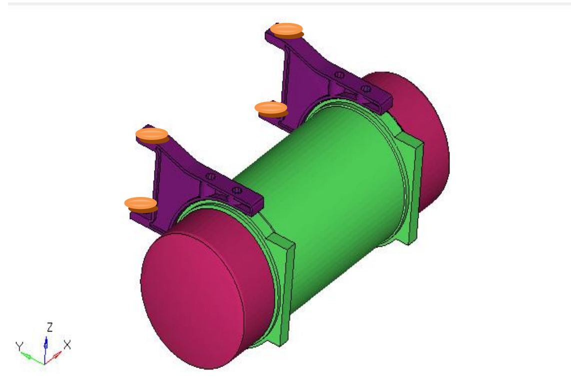

FIG -3 Boundary condition applied to the fuel tank bracket.

= All 6 DOF constrained over here

A load of 5G load is applied on braking along x axis, 5G load on turning along y axis and 10G load on humps along Z axis. These loads are applied for worst road condition. stress and displacement are calculated by using these data's

7.RESULTS AND DISCUSSION

A. STATIC ANALYSIS

Static analysis is performed with 5G, 5G, 10G and road load applied as body force in X, Y and Z directions. Maximum stress is developed for different design modification. below figure shows the maximum stresses in different design modification

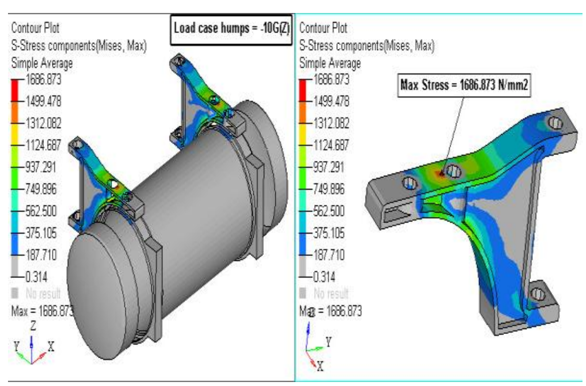

FIG-4 Maximum stress of base design

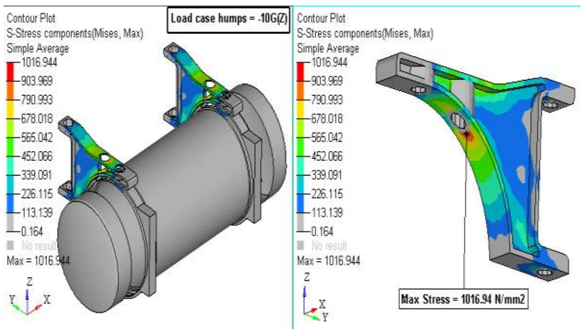

FIG-5 Maximum stress of without stiffener bracket

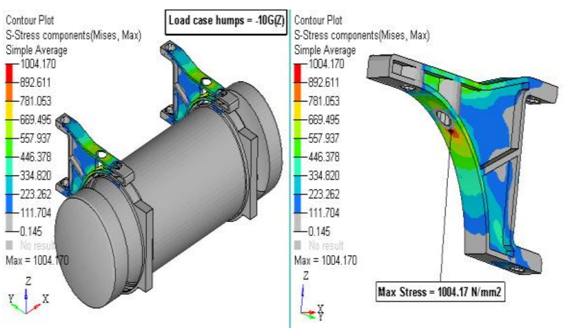

FIG-6 Maximum stress of single stiffener bracket

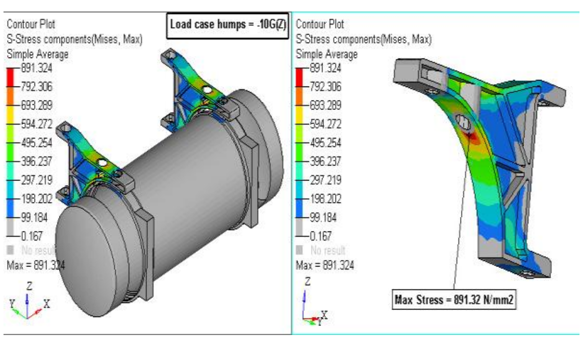

FIG-7 Maximum stress of multi stiffener bracket

from the static analysis it is observed that the multi stiffener bracket has less principal stress and it is best suited for bracket.

Table -2 stress results of different designs of fuel brackets for worst road condition

	Base design	Without stiffener	Single stiffener	Multi stiffener
x	1071.756 N/mm2	821.343 N/mm2	794.530 N/mm2	763.322 N/mm2
y	1644.386 N/mm2	841.108 N/mm2	833.441 N/mm2	793.771 N/mm2
z	1686.873 N/mm2	1016.94 N/mm2	1004.170 N/mm2	891.321 N/mm2

Table -3 Displacement results of different designs of fuel tank brackets

	Base design	Without stiffener	Single stiffener	Multi stiffener
x	0.406mm	0.265mm	0.265mm	0.257mm
y	1.210mm	0.692mm	0.690mm	0.675mm
z	1.329mm	0.787mm	0.787mm	0.763mm

B. MODAL ANALYSIS

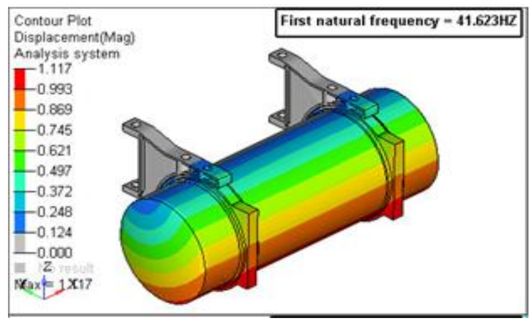

FIG -8 First natural frequency and mode shape of base design

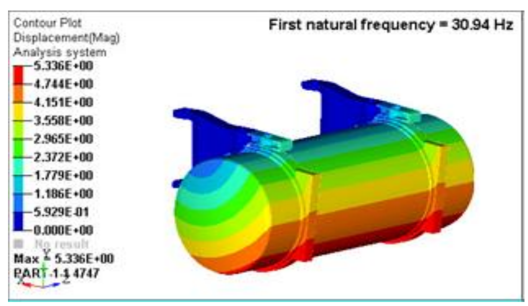

FIG -9 First natural frequency and mode shape of without stiffener bracket

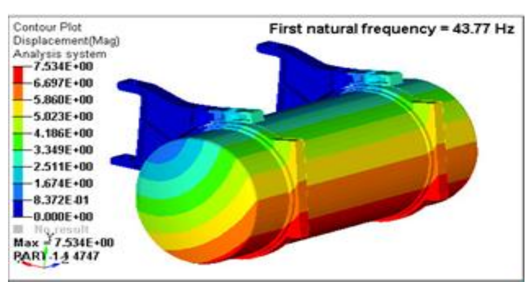

FIG -10 First natural frequency and mode shape of single stiffener bracket

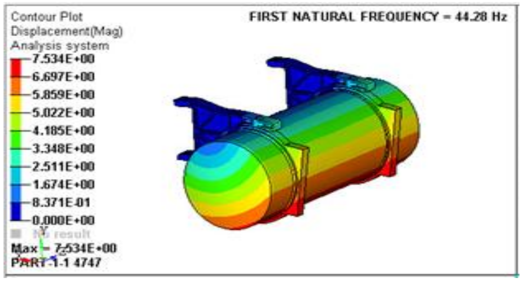

FIG -11 First natural frequency and mode shape of multi stiffener bracket

Table -4 First natural frequencies of different design modification

Design	First Natural frequency(HZ)
Base design	41.62
Without stiffener	31
Single stiffener	44
Multi stiffener	45

8.CONCLUSION

A successful effort has been made to predict the Eigen values and Eigen vectors of fuel tank mounting brackets assembly by numerical simulation. By comparing the results of all design iteration of normal mode analysis we came to the conclusion that the first natural frequency of multi stiffener fuel tank mounting bracket is 45Hz is good compare to other stiffeners.

9. SCOPE OF WORK

This paper explains the process of optimization of natural frequency of fuel tank mount bracket. Static analysis for the mounting brackets to evaluate the stress and displacement. Further scope is to use different materials or design, which further reduces weight.

10.REFERENCE

[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.

BIOGRAPHIES

Gajendra G is PG scholar (Machine Design) mechanical department, M S engineering college, Bangalore. He has two years of industrial experience. His area of interest is automotive design.

Prakasha A M is Assistant professor of mechanical department M S Engineering College, Bangalore. He has presented journal papers in international research journals. His areas of interest are composite materials and Automobile systems.

Dr Noor Ahmed R is a HOD of mechanical in M S Engineering College Bangalore. He is having 30 years of academics and research experience. His area of interest is composite materials.

Dr. K. S. Badarinarayan is principal of M S Engineering College, Bangalore. He having 27 years of academics and industrial experience including 6 years of research experience. His area of interest is Flexible manufacturing system scheduling.