ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Design and Optimization of HTV Fuel Tank Assembly by Finite Element Analysis

Ms.Baseera Banushaik
PG Student,
Department of Mechanical Engineering,
Malla Reddy College of Engineering,
Secunderabad.

Ms.I.Prasanna
Assistant Professor,
Department of Mechanical Engineering,
Malla Reddy College of Engineering,
Secunderabad.

ABSTRACT:

A fuel tank (or petrol tank) is a safe container for flammable fluids. The functions of a fuel tank are Storage of fuel, filling, and venting. A Fuel tank must provide a method for determining level of fuel in tank and Anticipate potentials for damage. 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.The objectives of the project are to develop structural modeling of fuel tank and bracket using CATIA, to perform Modal analysis of fuel tank and bracket by using Ansys 15 for finding natural frequency for basic design, to perform finite element analysis of fuel tank and bracket by using Ansys 15 for structural steel material for the basic design. In this project, design modifications on basic design to improve natural frequency and avoid high stress locations and analyse the maximum stress of fuel tank and bracket using ANSYS 15 software.

Keywords:

Tank, fuel tank Bracket, , Failure analysis, Catia, Elements, FEM, Ansys.

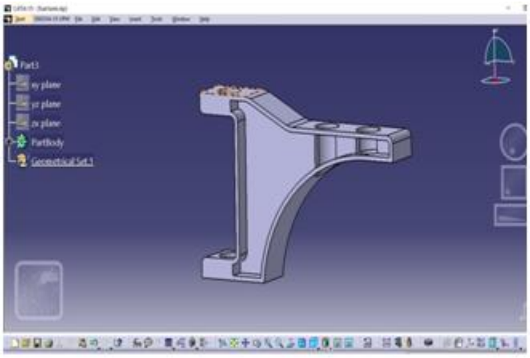
1. INTRODUCTION:

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.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

A fuel tank (or petrol tank) is a safe container for flammable fluids. Though any storage tank for fuel may be so called, the term is typically applied to part of an engine system in which the fuel is stored and propelled (fuel pump) or released (pressurized gas) into an engine. Fuel tanks range in size and complexity from the small plastic tank of a butane lighter to the multi-chambered cryogenic Space Shuttle external tank.

2. COMPLETE DESIGN OF A FUEL TANK MOUNTING BRACKET:



3. STATIC STRUCTURAL AND MODAL ANALYSIS:

The computer aided design of base model fuel tank mount bracket is shown in figure 3.1. 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. Static structural analysis was performed on the basic design and modified designs by applying proper loads and boundary conditions. Structural steel material is used in the analysis. Total deformation and von-mises stress results are plotted. As brackets are the areas of interest brackets are modeled accurately tank part is modeled as single solid part. The results on the brackets are only considered in the present analysis report.Modal analysis is performed on the structure to find the natural frequency of the structure. The model is exported as parasolid format from Catia and imported into Ansys workbench. The Geometry of the brackets for design modifications are shown in below figures.

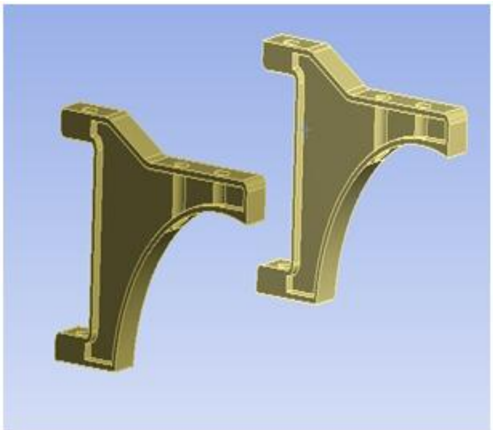


Figure: 3.1 Geometry of the Fuel tank brackets - Basic design

MESHING:

One of the most relevant steps in the Finite Element Analysis is the meshing. The speed and the accuracy of the results have a direct connection in how this part is done. The higher the numbers of nodes are the higher the accuracy of the results, however the speed of the simulation decreases.

HYPERMESH:

ALTAIR HYPERMESH is widely used for meshing for gas turbine rotor blades. It is almost used in all automobile-leading industries. For complex geometries it is best suited. The effective mesh generation is done. The main objective is to check all the element quality checking such as aspect ratio, warpage angle, skew angle, and jacobian. So tetra mesh and mapped mesh of gas turbine rotor blade is done. The other objective is to find out the stresses, deformation and temperature distribution and natural frequencies using structural, thermal and modal analysis. The material properties and loading conditions for gas turbine rotor blade are taken into consideration.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

MESHING OF FUEL TANK:

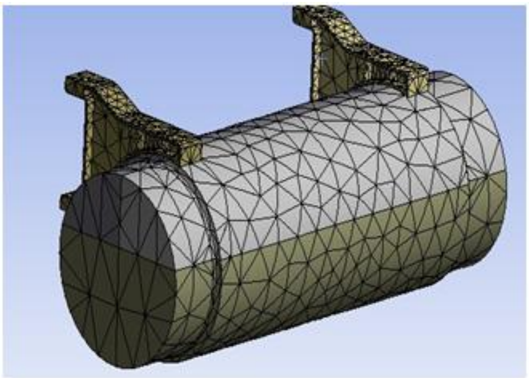


Figure: 3.2 Meshing of fuel tank in Tetrahedral

MESH DETAILS:

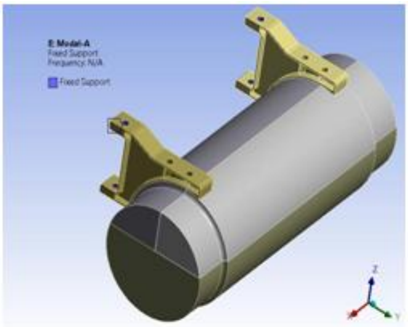
Statistics	
Nodes	11279
Elements	5733
Mesh Metric	None

ELEMENT TYPE:

Solid 187(Tetrahedral second order element)

3.3 LOADS & BOUNDARY CONDITIONS:

Bolt holes of the brackets are fixed in all degrees of freedom. Gravity load applied to the model. Loads & Boundary conditions applied to the model is shown in below figure.



4. RESULTS AND DISCUSSIONS

GEOMETRY 1: GEOMETRY FOR THE BASIC DESIGN

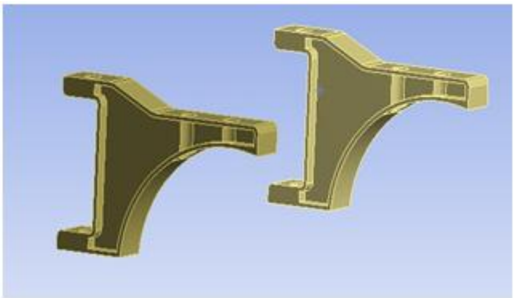


Figure: 4.1 Geometry for the basic design

TOTAL DEFORMATION:

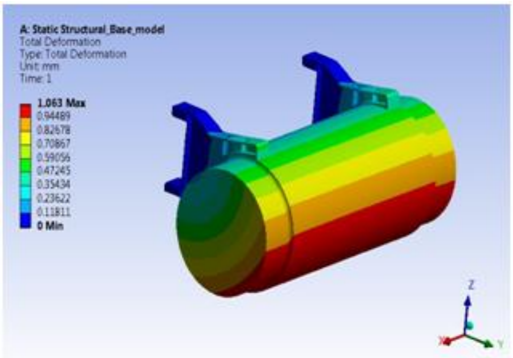
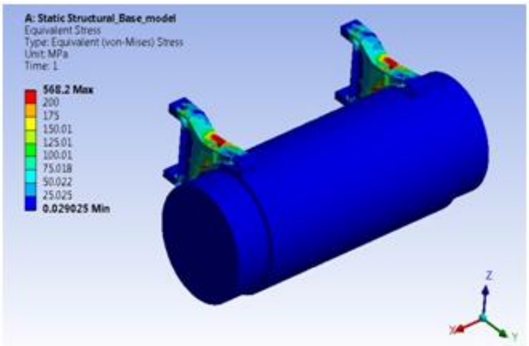


Figure: 4.2 Total deformation in the basic design

The total deformation observed in the structure is **1mm**. The result of deformation is as expected behavior and is accepted.

EQUIVALENT STRESS:



ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

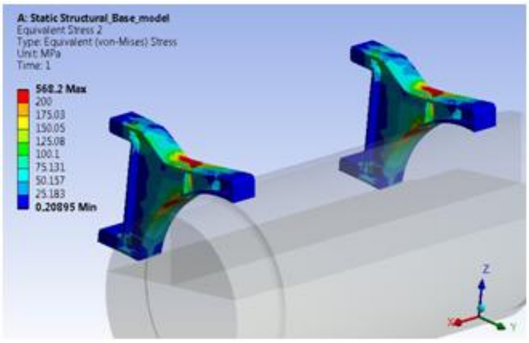


Figure: 4.3 Equivalent Stress brackets

The total equivalent (von mises) stress observed in the structure is **568MPa.**

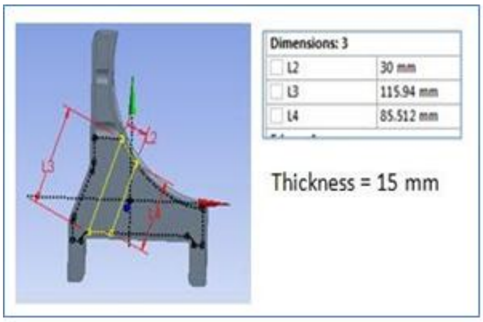
Tabular Data

	Mode	Frequency [Hz]
1	1.	70.183
2	2.	112.86
3	3.	371.6
4	4.	410.04
5	5.	568.46
6	6.	746.15

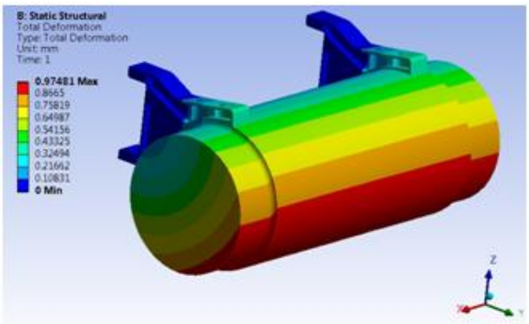
Basic Design

4.1.1 DESIGN MODIFICATION -1:

GEOMETRY:



TOTAL DEFORMATION:



The total deformation observed in the structure is **0.97mm**. The result of deformation is as expected behavior and is accepted.

EQUIVALENT STRESS:

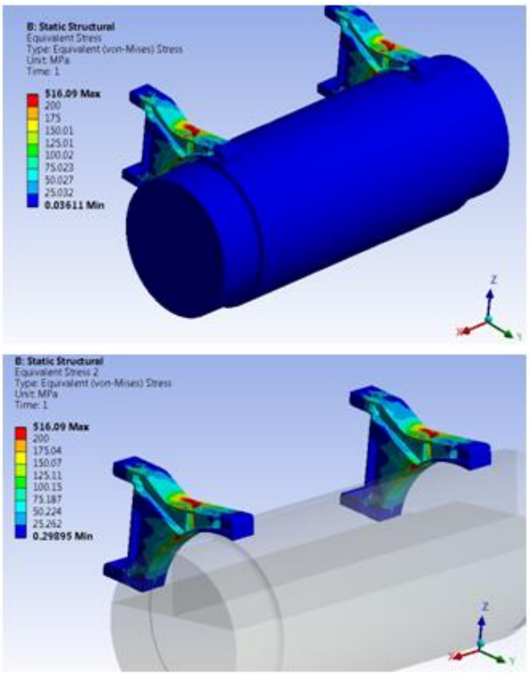


Figure:4.4 Equivalent Stress brackets

The total equivalent von mises stress observed in the structure is **516MPa.**

Tabular Data

	Mode	Frequency [Hz]
1	1.	71.745
2	2.	129.39
3	3.	379.73
4	4.	438.17
5	5.	605.27
6	6.	756.37

Design modification - 1

4.1.2 DESIGN MODIFICATION -2:

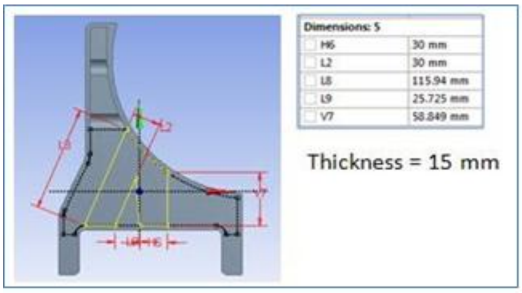
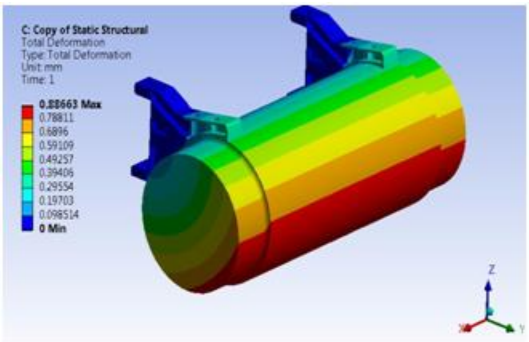


Figure: 4.5 Modified design-2 dimensions

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

TOTAL DEFORMATION:



EQUIVALENT STRESS:

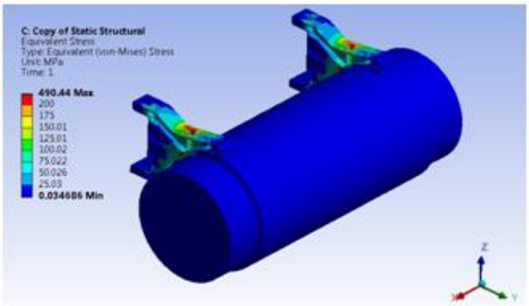


Figure: 4.6 Equivalent Stress structure

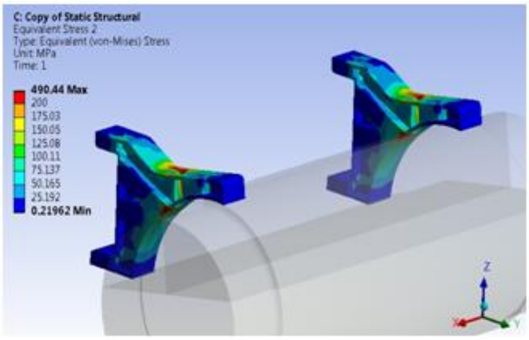


Figure: 4.7 Equivalent Stress brackets

The total equivalent von mises stress observed in the structure is **490MPa.**

Tabular Data

	Mode	Frequency [Hz]
1	1.	72.285
2	2.	132.14
3	3.	384.62
4	4.	440.04
5	5.	607.7
6	6.	763.73

Design modification - 2

4.1.3 DESIGN MODIFICATION -3:

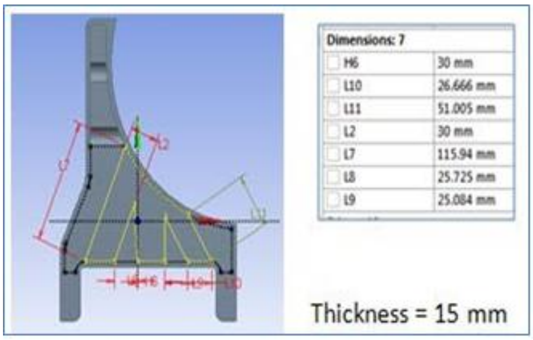


Figure: 4.8 Modified design-3 dimensions

TOTAL DEFORMATION:

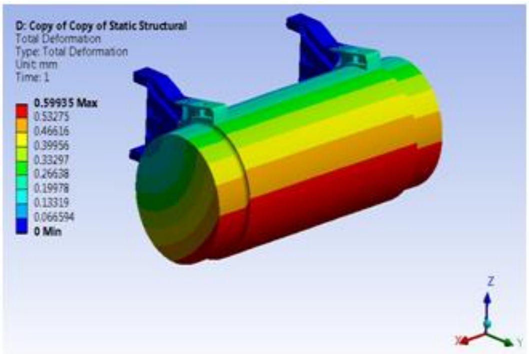


Figure: 4.9 Total deformation in the design modification -3

The total deformation observed in the structure **is 0.59mm.** The result of deformation is as expected behavior and is accepted.

EQUIVALENT STRESS:

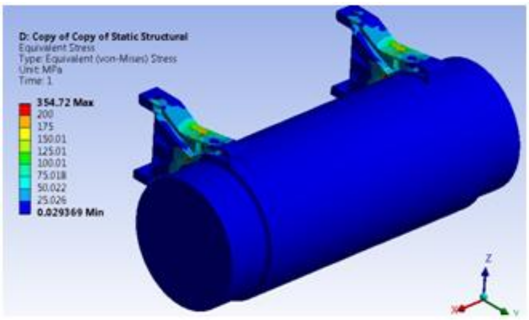


Figure: 4.10 Equivalent Stress structure

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

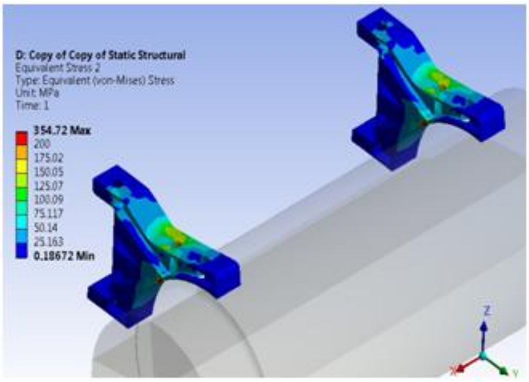


Figure: 4.11 Equivalent Stress brackets

The total equivalent von mises stress observed in the structure is **354MPa.**

Tabular Data

	Mode	Frequency [Hz
1	1.	73.284
2	2.	135.33
3	3.	398.51
4	4.	442.72
5	5.	611.98
6	6.	789.1

Design modification - 3

RESULTS SUMMARY:

The results for the basic design and design modifications are represented in below table.

Design	Deformation (mm)	Mises von stress (MPa)
Basic Design	1.06	568
Design Mode-1	0.97	516
Design Mode-2	0.88	490
Design Mode-3	0.59	354

Table1: Results Comparison of Geometry 1, Geometry 2, and Geometry 3.

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 iterations the following conclusions are made.
- static structural analysis we came to the conclusion that the brackets with multi stiffeners i.e. design modification 3 are giving good results of deformation of 0.59mm and von mises stress of 384MPa. Which are minimal values as compare with basic and other design modifications.
- From the Normal mode analysis that the first natural frequency of multi stiffener fuel tank mounting bracket is 73Hz is good compare to other stiffeners.

REFERENCES:

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

➢ Tatsuo kasuga, Eisei higuchi, “method and structure for mounting a fuel tank” US5794979 A.

➢ 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.

➢ 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.