International Journal of Advances in Engineering and Management (IJAEM)
Volume 4, Issue 1 Jan 2022, pp: 802-808 www.ijaem.net ISSN: 2395-5252

Modeling and Finite Element Analysis of Hydraulic Tank Mounting Bracket Structure for A Dump Truck

A S Jeyaseelan[1], M Sivakumar[2]
PG Student, Department of Mechanical Engineering, RVS Technical Campus – Coimbatore
, Associate Professor, Department of Mechanical Engineering, RVS Technical Campus - Coimbatore

Submitted: 15-01-2022 Revised: 23-01-2022 Accepted: 25-01-2022

ABSTRACT
A dump truck is used for transporting loose material for construction. A typical dump truck is equipped with a hydraulically operated open-box bed hinged at the rear, the front of which can be lifted up to allow the contents to be deposited on the ground behind the truck at the site of delivery. The tank is rigidly held in position with the help of front and rear brackets which are in turn attached to the rails of the main frame. The hydraulic fluid is stored in the hydraulic tank which is used to raise and lower the open-box bed. The positioning of the hydraulic tank using appropriate bracket mountings hence plays an important role in the dumping operation. The 3D modeling has been carried out and analysis has been done by software. Finally the Factor of safety of the component has been obtained and validate with FEM results.

Keywords: Finite element analysis, CAD modelling and bracket structure

I. INTRODUCTION

FEA is a powerful tool for numerical solution of wide range of engineering solutions and applications. Finite element analysis was first developed for use in the aerospace and nuclear industries where the safety of structures is critical. Today, the growth in usage of the method is directly attributable to the rapid advances in computer technology in recent years. As a result, commercial finite element packages exist that are capable of solving the most sophisticated problems, not just in structural analysis, but for a wide range of phenomena such as steady state and dynamic, model and temperature distributions.

Static analysis is the analysis that is carried out when the body is said to be in the condition of rest. The analysis that is carried out when the body is said to be in motion is called Dynamic analysis. A structural model created can be used to predict the behaviour of the real structure, under the action of external forces. The response is usually measured in terms of deflections and stresses. The response is static if the loads are steady. This is called static analysis.

When the loads vary with time it is called dynamic analysis. A dynamic force excites velocities and accelerations that produce appreciable variations of displacements and stresses. These are computed over time and the response history is called transient response analysis.

If the properties of the structure, such as stiffness remain constant during the entire analysis, the analysis is called linear. If these properties vary, the analysis is non-linear. Such variation can be due to large displacements in the structure, large scale yielding in the material or changes in boundary conditions.

Linear analysis deals with problems in which the structural response is linear. Therefore, if the applied forces are doubled, then, the displacements and internal stresses also double.

Static or steady state analyses are those where the solution is independent of time. Inertial forces are either ignored or neglected and so there is no requirement to calculate actual time derivatives. Linear static analyses are usually sufficient for situations where loads are known and the instance at which peak stress occurs is obvious. When performing a linear stress static analysis, static loads are applied to the model which has to be analyzed.

FEA is used in the recent days in almost all fields of engineering for different requirements. Using FEA various kinds of analysis can be carried out such as Structural analysis, Thermal analysis, Vibration analysis, Fluid flow analysis, Buckling analysis, Electromagnetic evaluations and so on.

The basic concept of FEM is that the structure to be analyzed is considered to be an assemblage of discrete pieces called ‘elements’ that are connected together at a finite number of points or nodes.

The finite element model is made up of different types of elements such as quadrilateral, triangular, rod type elements etc. In FEM the stiffness of each element is calculated and the global stiffness matrix is then calculated. Using the stiffness matrix we can define the load-deflection relationship. We now apply the loads and boundary conditions to obtain the deflections at various nodes.

2D drawings of individual parts of mountings, tank and frame rails have been considered to obtain the 3D model. The obtained 3D model has been exported to ANSYS and loads have been applied make the stress analysis. In addition to the above, manual calculations have been made, considering the brackets to be a beam fixed at both the ends, applying appropriate formulae. The value thus obtained is compared with the value obtained by the software and shown to be matching. From the modelling and analysis of the bracket mountings it is evident that the brackets are over-designed and shape optimization can be taken up as future scope of the project.

II. LITERATURE STUDY

Some of the study has been carried out to improve the primary regular recurrence of hydraulic and fuel tank sections for hardcore vehicles utilizing FEA. Ustaoglu, H.B, explained about the static analysis investigation is completed for every single changed configuration to discover the maximum relocation and von misses worry at the basic area. Most extreme rule pressure and the least standard worries are conveyed. The destinations of the venture are to create auxiliary displaying of tank and section utilizing cad tool, to do modal investigation of fuel tank and section by utilizing analysis tool for discovering normal recurrence for essential plan, to do limited component examination of fuel tank and section by utilizing analyzing tool for basic steel material for the fundamental structure. Wang, Y., Liew, J.Y.R, Lee, S. Ch, discussed about the structural performance of the tank structure mounting section for business vehicles. The task incorporates the geometry and limited component demonstrating of fuel tank mounting section plan. Abhishek etal, have discussed about the finite element analysis of fuel tank mounting bracket and its performance.

III. OBJECTIVE

The main objective of this paper is to create CAD model, FEA analysis and determine factor of safety to withstand hydraulic tank mounting bracket structure. The 2D drawings of each part of each bracket are create and converted to 3D using CATIA software. These parts are assembled to form the brackets. Thus the two brackets are done one by one. Then the drawings of the hoist structure and the rails of the frame are given which are again converted to 3D. Finally the whole assembly consisting of brackets, hoist and rails is done. Further to be finite element analysis for finding out the factor of safety.

IV. METHODOLOGY

A hydraulic system is one of the most important system of the dump truck as hydraulics is involved most of the systems. The hydraulic system mainly for hoisting, steering and brake oil cooling. Hence, the hydraulic tank is an important part of the dump truck as hydraulic pressure is applied to different parts by flow of oil from the tank itself. Here the tank has a capacity of 510 liters. SAE-10 hydraulic oil is filled up in the tank. A phosphorous coating is given to the tank so that there is no reaction between the oil and the tank material.

Bracket is a part or component or structure that is used to hold other parts in its position. Hydraulic tank mounting brackets are the brackets on which the hydraulic tank is mounted, that is, it is between these brackets that the hydraulic tank is held safely. There are two brackets, the front and the rear, which are bolted to the main frame of the truck. Hence, the design of these brackets is very important as the whole tank weight acts on these brackets and thus failure of these brackets should not take place.

Computer Aided Three dimensional Interactive Application, CATIA stands for CATIA. It is a Dessault System software which is user friendly, flexible and interactive. It is a mechanical design software, addressing advanced process centric design requirements of the mechanical industry. With its feature based design solutions, CATIA has proved to be highly productive for mechanical assemblies and drawing generation.

CATIA is the best solution capable of addressing the complete product development process from product concept specification through product in service in a fully integrated and associative manner. Mechanical design solutions provide tools to help you implement a sophisticated standard based architecture. This enables collaborative design and offers digital mockups and

International Journal of Advances in Engineering and Management (IJAEM)
Volume 4, Issue 1 Jan 2022, pp: 802-808 www.ijaem.net ISSN: 2395-5252

hybrid designs. Various types of designing activities can be carried out in this software such as Solid modeling, surface modeling, Wire frame modeling, sheet metal design, tooling design, core and cavity design, weld design and so on. Meshing can also be carried out using this software. Machining operations can also be carried out using this software.

Finite element structural analysis is a method of predicting the behaviour of a real structure under specified load and displacement conditions. Structural analysis is probably is the most common application of the FEM. The term structure applies to all the civil, mechanical, aeronautical and naval structures.

Structural analysis comprises the set of physical laws and mathematics required to study and predict the behaviour of structures. From a theoretical perspective the primary goal of structural analysis is the computation of deformations, internal forces and stresses. Structural analysis can be viewed as a method to prove or test the engineering design without directly testing it using prototypes or such similar methods.

There are various types of structural analysis such as stress and displacement analysis, vibration and frequency analysis, Thermal analysis, buckling analysis and so on. Of all these stress and displacement analysis is the most widely used analysis which is required in all fields of engineering.

Structural analysis can be carried out in two ways that is Static analysis and Dynamic analysis. Further they are classified as Linear and Non-linear static and dynamic analysis.

4.1 Cad Modeling

Created a CAD model of hydraulic tank and its mounting brackets, rail structure chassis and its bracket structure, assemble all the parts with fasteners. The chassis structure created and assemble with tank structure for finite element analysis.

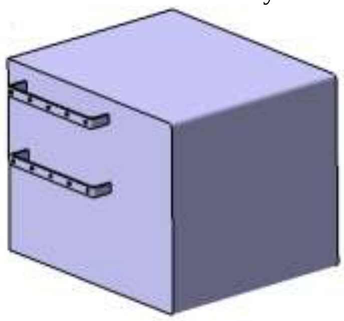

Fig 1 Hydraulic Tank

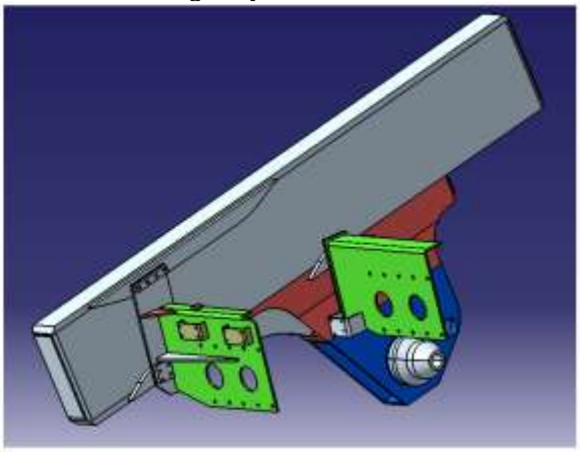

Fig 2 Frame Structure

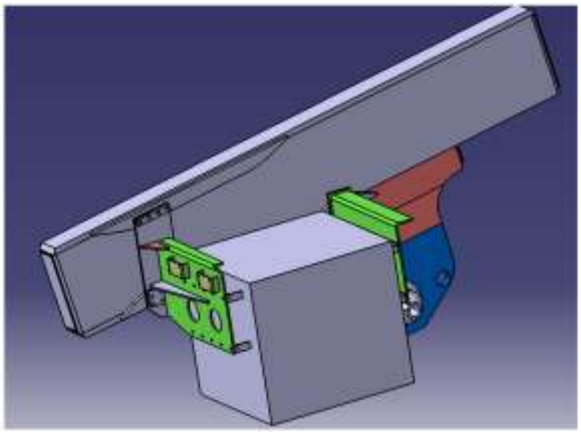

Fig 3 Assembly of Frame and Tank

4.2 Finite Element Analysis

To mesh, apply the loads and constraints to contemplate true condition using ANSYS**.** From here Finite Element Analysis of the parts starts. The assembly done in CATIA is then imported to ANSYS. They are then meshed, boundary conditions are applied and loads are applied. To find maximum stress and Factor of Safety (FOS) of the mounting brackets. Various stresses like Max. Principal stress, equivalent stress, shear stress, deflection, etc are hence found out in ANSYS. The FOS is also determined which in turn determines the durability of brackets.

Principal plane through a point within a material under stress is that place, the stress across which is wholly normal and no shear stress exists along this plane. The normal stress along the principal plane is known as principal stress. At any point in a strained material, there are three mutually perpendicular planes carrying direct stresses only. The maximum stress out of these three planes is called Max. Principal stress which is applied parallel or tangential to a face of a material as opposed to a normal stress which is applied perpendicularly. The term used to describe the degree to which a structural element is displaced under a load. The deflection of a member under a load is directly related to the slope of the deflected shape of the member under that load and can be calculated by integrating the function that mathematically describes the slope of the member under that load.

A strain that acts parallel to the surface of a material that it is acting on and normal strain in contrast, acts perpendicular to the surface.

Static structural analysis option is selected which means that static structural analysis is to be done. The holes which are bolted to the rail are fixed using 'Fixed Support' option in the Support icon of the environment toolbar. Loads are applied on all the holes of the brackets as the loads act only in those sections where the tank would be bolted. The forces act upwards (reaction forces) as the center of gravity of tank acts downwards at the center.

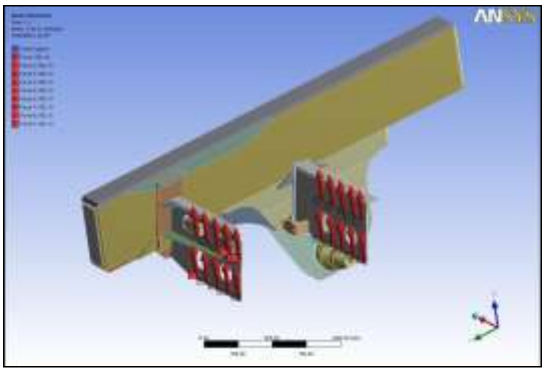

Fig 4 Loading conditions

The material properties such as young's modulus, poisson's ratio, density, ultimate strength are entered in the Engineering data window. (Further details of the material and its properties are given in manual calculations part) Now, the model is meshed. Here it is a combined element type mesh.

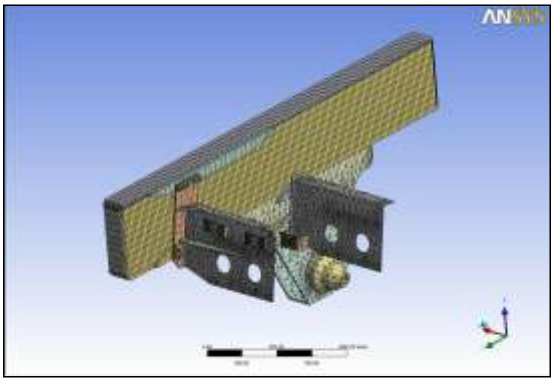

Fig 5 Mesh

The required parameters such as maximum principal stress, equivalent stress, Deformation, Shear stress, Shear strain, middle stress, etc are added to the solution list from the solution toolbar. Solve command is given so that the software computes the solution for the entire meshed model. Once, the solution is calculated by the software, they can be graphically seen and the values of it are also displayed.

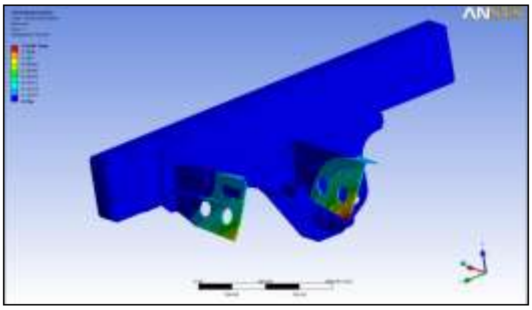

Fig 6 Total Deformation

The maximum deformation of the plate is = 1.41 Mpa which is in a very small region. Hence, the design is said to be safe.

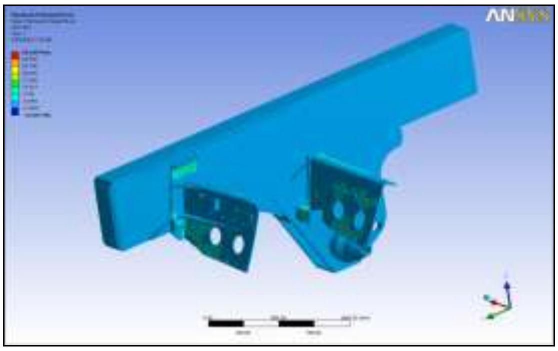

Fig 7. Maximum principal stress

Max. principal stress 60.188 Mpa. It is seen that maximum principal stress of most of the model lies in the range of 5.36 to 27.40 Mpa. So, safe design.

International Journal of Advances in Engineering and Management (IJAEM)
Volume 4, Issue 1 Jan 2022, pp: 802-808 www.ijaem.net ISSN: 2395-5252

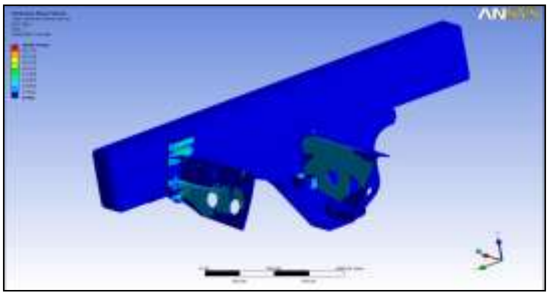

Fig 8. Maximum shear stress

Maximum shear stress 28.81 Mpa. Most of the regions lie between the ranges of 6.40 to 12.80 Mpa. Hence, the design is safe. The maximum value of middle principal stress 18.18 Mpa. Most of the part lies in the range of -2.89 to 9.74 Mpa. Hence, the design is safe.

The maximum value of minimum principal stress = 8.68 Mpa. As most of the regions lie in the range of -17.37 to 2.17 Mpa, the design is treated to be safe and maximum shear stress 22.28 Mpa. Most of the regions lie between the ranges of -3.00 to 7.80 Mpa. Hence, the design is treated to be safe.

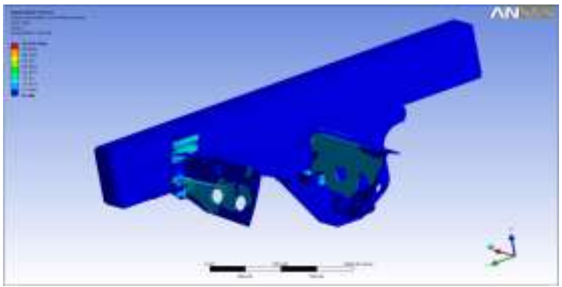

Fig 9. Equivalent Stress

The equivalent stress of the material is 51.63 Mpa which is the working stress of the model. This is used in the calculation of the Factor of Safety. The material like high strength structural steel.

Yield strength	410 Mpa
Tensile strength	540 Mpa
Modulus of Elasticity	$2 * 10^5$ Mpa

V. CALCULATION

Capacity of hydraulic tank 510 liters, Oil used SAE 10W and Density of oil 865 kg/m^3 and Mass = Density * Volume m_1= 865 X 510 X 0.001
Mass of hydraulic oil, m_1= 441.15 Kg
Weight of hydraulic oil,
w_1 = 441.15 X 9.81 = 4327.60 N
Mass of hydraulic tank,
m_2 = 765 Kg
Weight of hydraulic tank,
w_2 = 765 X 9.81 = 7504.65 N
Total weight,
$w = w_1 + w_2$,
4327.60 + 7504.65 = 11832.25 N for 20 bolts weight for one bolt = 11832.25 / 20
Hence, W = 592 N. factor of safety approximately above two.

VI. CONCLUSION

To conclude that the tank mounting bracket structure analysis of the hydraulic tank is to be performed. The hydraulic tank is modeled from the structural steel material and supported by the mounting bracket made of steel material. FEA was performed using ansys. The displacement plot, von misses stress plot are used to compare results. Displacements and stresses are high in original design hence three different modified structures were proposed to find the best one which will meet one million cycles of life. The FE analysis was conducted for the initial design of the hydraulic tank and based on the results it was found that the life of the tank was under the expected life. The design iteration was done for each design iteration, FE analysis was carried out by performing all the tasks. All the time the design changes were considered based on FE results and 3D models were created in design software. Design is finalized based on the FE result and considering the ease of manufacturing. The structural design is found to be more than expected one million cycles. The values are within the standard and recommended values.

REFERENCES

[1]. Umesh s ghorpade, Mahendra gaikwad “Finite element analysisand 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.

[6]. Ustaoglu, H.B., et al., "Static and Dynamic Analysis of Plastic Fuel Tanks Used in Buses," 3rd International Conference on Material and Component Performance under Variable Amplitude Loading, Elsevier, 509-51

[7]. Gajendra G “Design and optimization of HTV fuel tank assembly by finite element analysis.

[8]. Michael Davis, Zhingxing fu, Qunhui Han, “optimizing fuel tank design for high-speed vehicles” June 9-12 2009.

[9]. Abhishek P1, Hardeep S M 2, Sarvocch G 3, Anil L, “Finite Element Analysis of Fuel Tank Mounting Bracket”.

[10]. Wang, Y., Liew, J.Y.R, Lee, S. Ch., "Structural performance of water tank under static and dynamic pressure loading," International Journal of Impact Engineering,

[11]. Kichang Kim and Inho Choi, Design Optimization Analysis, SAE TECHNICAL, 2003-01-1604

[12]. Doo-Ho Lee, Jeon, Woo Chang, Chan-Mook Kim, Optimal Shape Design of an AirConditioner Compressor Mounting Bracket in a Passenger Car, SAE TECHNICAL PAPER SERIES, 2003-01-1667

[13]. Hong Suk Chang, A Study on the Analysis Method for Mounting Brackets, SAE TECHNICAL,PAPER SERIES, 2006-01-1480.

[14]. Michael Champrenault and Clayton A. Maas, Jack Cunningham, Magnesium Powertrain Mount Brackets: New Application of Material Being usedin this Sub-System for VehicleMass Reduction, SAE TECHNICALPAPER SERIES, 2007-01-1031.