

Design and Development of Hydraulic Tank through Structural and Fatigue analysis

Prakash M A¹, Manjunath T V²

¹Prakash M A, PG scholar, Cambridge Institute of Technology, Bangalore, Karnataka, India

²Manjunath T V, Assistant prof, Cambridge Institute of Technology, Bangalore, Karnataka, India

Abstract - The present study of the design and development of the hydraulic tank used in heavy-duty and earthmoving equipment's through structural and fatigue analysis is to improve the life of the tank to have a minimum of 2 million cycles. The static structural analysis was carried out for base design and the high stress and displacement were noted and based on the results, modification was carried out to reduce the stress and displacement on the tanks. To improve the life of the tank series of iteration were carried out by considering the results of the base model. The model is analyzed for a total of 6 loads cases for which the tank will undergo during the working of the vehicles. Since most of the time vehicle will work with heavy load against the gravitation force the worst case of 5G force was analyzed for forward, backward and side-side direction. The total of three design iterations were carried out, each design iteration was analyzed for all six load cases and von Mises stress, principal stress was used to calculate the fatigue life. Based on the results design changes were carried out to reduce the stress in each direction hence to increase the fatigue life.

Key Words: Hydraulic Tank, Fatigue Life, Principal stress, Design Optimisation

1. INTRODUCTION

The FEA methodology has been used to develop the common hydraulic tank which can be used in various heavy trucks and also earthmoving equipment like excavators, pavers, backhoe, loaders, rollers and many other vehicles where the hydraulic tank plays a major role in the smoother functioning of the equipment's. Since the physical testing of a prototype of tanks would cost huge hence it will be best to develop the tank for the required life using the most robust and reliable source of finite element analysis methodology.

We have considered several methodologies from a literature survey to develop this tank. The structural static analysis and fatigue methodology to achieve the required life and structural strength of the hydraulic tank. The working loads and boundary conditions are considered as per the fixation of the tank to the frame. The load conditions are considered by the worst load under which the equipment's work considered also considering the 5 times the gravitational force against which the vehicles work during most of its operations.

The hydraulic tank undergoes frequent fatigue because of the sloshing of the oil in the tank. The loose floor movement

of the liquid in a partially crammed field because of outside force is known as sloshing. Oil sloshing has an extensive variety of programs in engineering fields which include hydraulic tanks, bins transporting liquid on highways, large ships and big vehicles, seismically excited storage tanks, dams, and nuclear reactors. Sloshing causes fantastically localized strain over partitions which can flip purposed damage and add enough to have an effect on the stableness of the car which contains the container. The movement of the free floor motion of liquid relies upon sort of movement, amplitude, frequency of external force, tank shape, tiers and liquid properties. Thus, sloshing phenomena and related to structural behavior have been analyzed whilst designing containers for liquid garage and hydraulic structures.

The fatigue methodology uses the Goodman's theory considering the mean stress effect into consideration since most of the earth moving vehicles normally works in area where the land will be always being unequal also the force will be fluctuating due to hydraulic oil sloshing in the tank. The stress from the structural analysis were considered for the fatigue calculation the principal stress being the major causes for the direction fatigue the max and minimum stress are considered for the amplitude and mean stress. The stress from the FEA is also verified using simple hand calculation which would replicate tank supported to the mounting bracket. The hand calculation for the fatigue life was also performed to make sure the expected life of 2 million cycles.

2. LITERATURE SERVEY

[1] Has carried out a study to improve the primary regular recurrence of fuel tank sections for hardcore vehicles utilizing FEA. The static 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. In this venture, structure alterations on essential plan to improve normal recurrence and maintain a strategic distance from high-pressure areas and investigations the most extreme worry of the fuel tank and section utilizing ANSYS 15 software.

[2] Have done the plan of fuel tank mounting section for business vehicles. The task incorporates the geometry and limited component demonstrating of fuel tank mounting section plan. Geometrical demonstrating was finished utilizing Pro-E; limited component displaying utilizing Hyper mesh programming and investigation was finished utilizing Opti struct solver. This venture manages the static investigation of section gathering. This venture gives a strategy to an investigation of a get together comprising of segments made of metal parts.

3. OBJECTIVE

The scope of the project is to evaluate the structural integrity of the hydraulic tank by performing structural analysis for the load cases shown above and make fatigue life calculations for each individual component and welds to achieve the minimum life of 2 million cycles for all the components.

3.1 Model Information

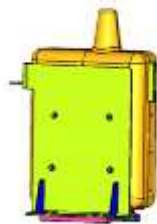


Fig-1 Model Information

Fig-1 shows 3D models of hydraulic tank developed to achieve the 2 million life cycles for the rated load cases the 3D models of tanks are created in solid work modeling software as per drawing with scale 1:1.

3.2 FE modeling Details

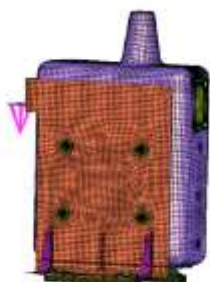


Fig- 2 Meshing of the component

As shown in Fig-2 Hydraulic Tank and L Bracket is modeled with shell element except surrounding inserts where it is modeled with hex elements. All the bolts, washers and inserts are modelled with hex elements. Mounting blocks are modeled with hex elements. Contacts are defined at a position where two plates contact each other. The Filter is represented as a mass element at its CG location. Hydrostatic

pressure load is applied in inertia load analysis to represent fluid effect.

3.3. Material Property

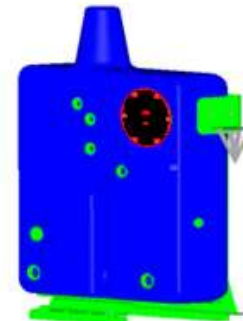


Fig- 3 Material property

As shown in Fig-3 Material properties are assigned as per the color code. Composite material is assigned to Tank. The L bracket and the inserts are made of the steel material and are assigned with steel shown in green color. The diffuse is modeled as mass element and density is tuned to achieve the mass of the component. Young’s modulus and passions ratio for each component is assigned as per the below table. The fatigue material properties of the materials are evaluated from the FE safe

Table -1: Material Property

| Material | Young’s Modulus | Poisson Ratio |
|---------------|-----------------|---------------|
| Composite Mat | 4500 (MPa) | 0.45 |
| Steel | 210000 (MPa) | 0.3 |

3.4. Load Condition

The above load case represents the condition when the vehicle moves in the forward direction as oil gets flushed towards the front face of the hydraulic tank, to represent this phenomenon the pressure is applied as shown above. Hydrostatic pressure(P)=Fluid density*acceleration(3G)*H1. Along with pressure inertia load of 5G (5 times the gravitation force) applied in the vehicle moving direction.

So there are a total of six load conditions for which the tank is analyzed.

Load case 1- Hydraulic oil Pressure + 5G inertial Load positive X direction

Load case 2- Hydraulic oil Pressure + 5G inertial Load negative Load X direction

Load case 3- Hydraulic oil Pressure + 5G inertial Load Positive Y direction

Load case 4 - Hydraulic oil Pressure + 5G inertial Load negative Y direction

Load case 5 - Hydraulic oil Pressure + 5G inertial Load Positive Z direction

Load case 6 - Hydraulic oil Pressure +5G inertial Load negative Z direction.

3.5. Boundary Condition

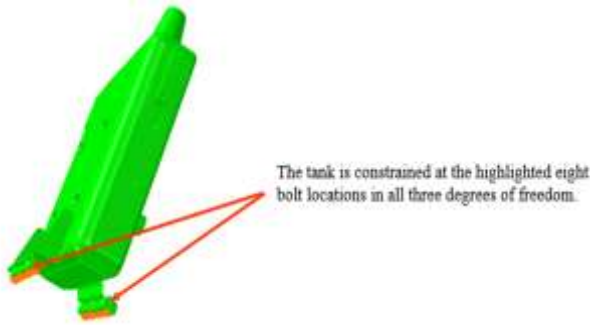


Fig- 4 Boundary Condition

As shown in Fig-4 the Hydraulic tank is bolted to the frame by the bolts at the above-highlighted location of the mounting brackets. Hence the model is constrained in all directions to represent the real behavior of the assembly when the load conditions are applied to simulate the assembly.

4. FINITE ELEMENT ANALYSIS OF EXISTING DESIGN

The below results are given to show the displacement on the tank and L bracket for all the above stated 6 directions. Loadings. The variation of the displacement is shown on the legends on the left side. The red and blue color represents the max and min displacement respectively.

Load Case1

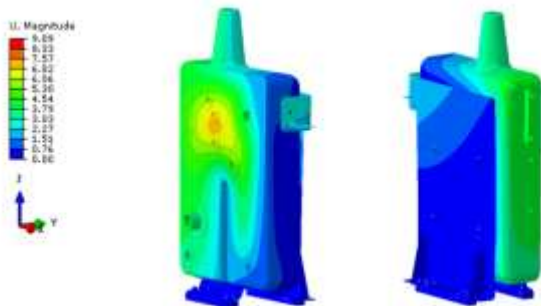


Fig-5 Load case 1 displacement plots

Load Case2

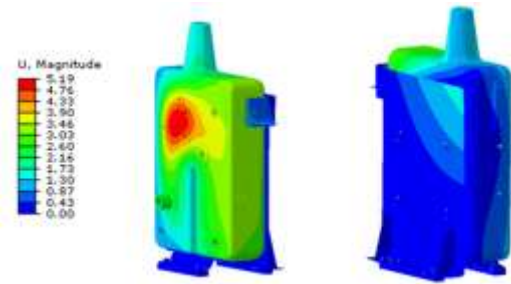


Fig-6 Load case 2 displacement plots

Load Case3

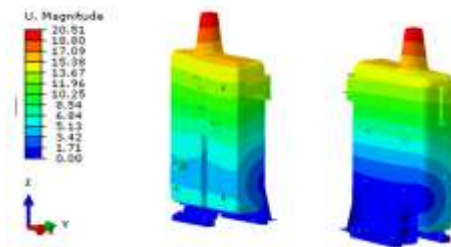


Fig-7 Load case 3 displacement plots

Load Case4

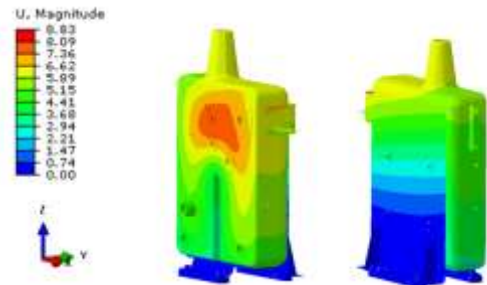


Fig-8 Load case 4 displacement plots

Load Case5

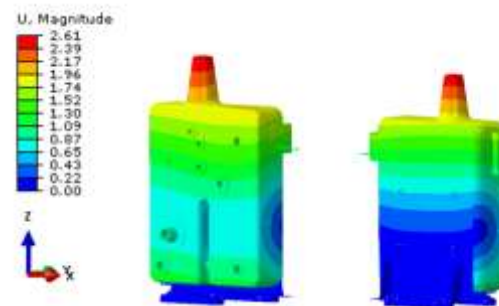


Fig-9 Load case 5 displacement plots

Load Case6

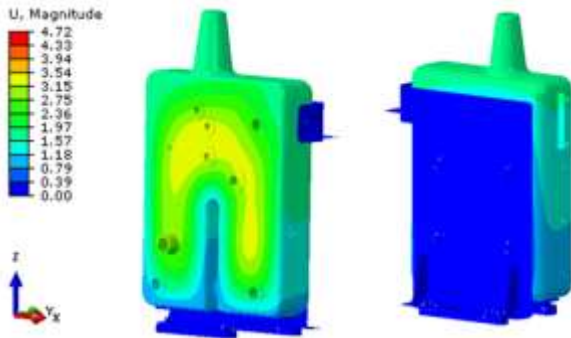


Fig-10 Load case 5 displacement plots

4.1 FEA Results for inertia load analysis for fatigue evaluation

The Stress amplitude was calculated from the stress of positive and negative load direction for X, Y, and Z loading directions. The stress amplitude is calculated on all the nodes using max principal stress. The highest stress amplitudes are probed and shown on the tank and on to the L brackets. The stress amplitude is then directly compared with the endurance limit of the material.

Load case 1 and 2: 5G X direction loading condition for stress amplitude

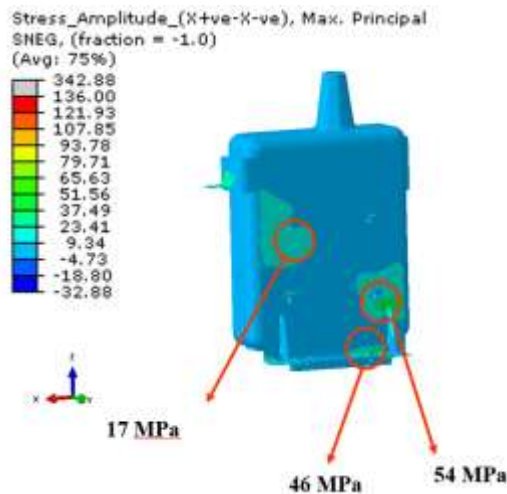


Fig-11 Load case 1-2 Stress Amplitude

Load case 3 and 4: 5G Y direction loading condition for stress amplitude

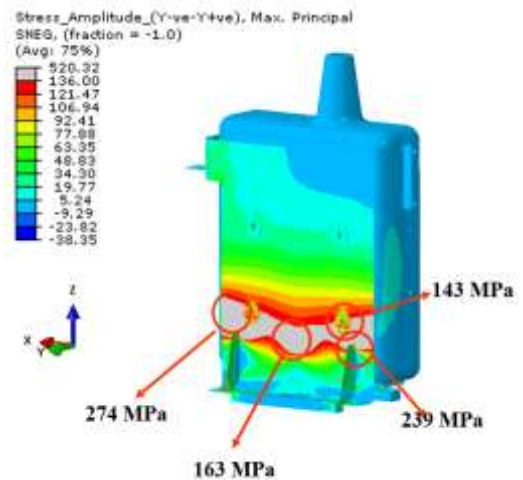


Fig-12 Load case 3-4 Stress Amplitude

Load case 5 and 6: 5G Z direction loading condition for stress amplitude

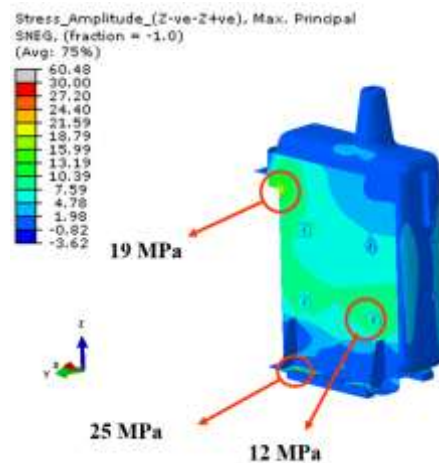


Fig-13 Load case 5-6 Stress Amplitude

4.2 FEA Results and Conclusion

The stress in the Y direction load case is more than the fatigue strength of the material. Since stress is above the fatigue limit of the material only 1000 cycle of life is expected. Also at the welds the stress was above the fatigue limit of welds material. Design iterations are carried out to make design safe for all the load conditions mainly for fatigue life and also to achieve the minimum life of 2 million cycles.

5. DESIGN MODIFICATION OF THE TANK

The above results show that the design does not meet the minimum requirement of 2 million cycles. Design iterations are performed to make the design safe for fatigue in all load directions also to achieve the minimum required life.

5.1 Design Iteration 1

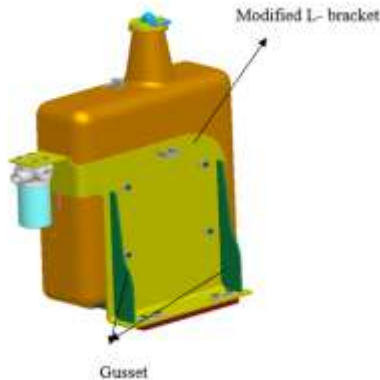


Fig-14 Design Iteration 1

The strength of the L bracket is increased by adding the gusset of larger length compared to the initial design. The design of the L bracket is changed as shown in Fig-14.

5.2 Design Iteration 2

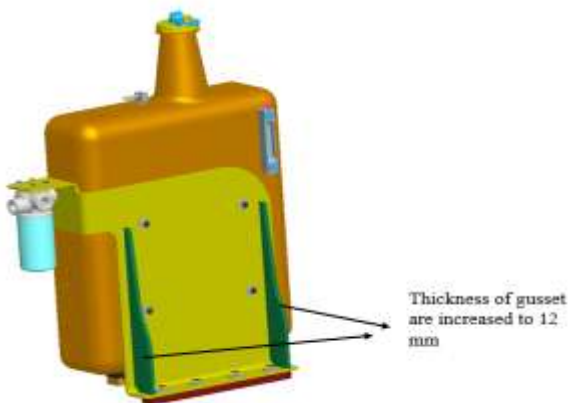


Fig-15 Design Iteration 2

The strength of the L bracket is increased by adding the gusset also the thickness of the gusset is increased to 12 mm.

5.3 Design Iteration 3

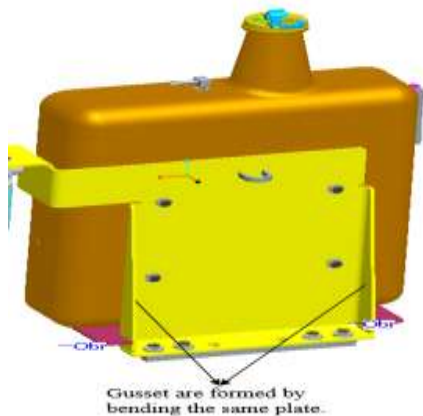


Fig-16 Design Iteration 3

The effect of the front gusset was achieved by making the gusset in the same plate by bending the plate as shown in Fig-16.

6. FINITE ELEMENT ANALYSIS OF FINAL DESIGN

The analysis was performed for all the above 3 design modifications shown above and design Iteration 3 was finalized based on the results which yielded the minimum life of 2 million cycles in all the load directions explained above. Also, design 3 was finalized based on ease of manufacturing where the effect of the gussets was achieved through bending the L bracket it's self which will reduce the welding of the gussets to the L bracket. The finite element analysis results for displacement and stress amplitude were summarized below.

Load Case 1

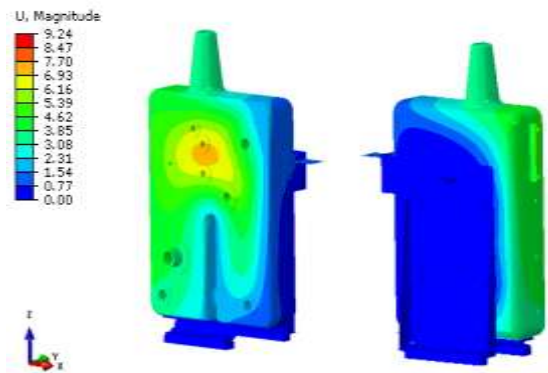


Fig-17 Load case 1 displacement plots

Load Case 2

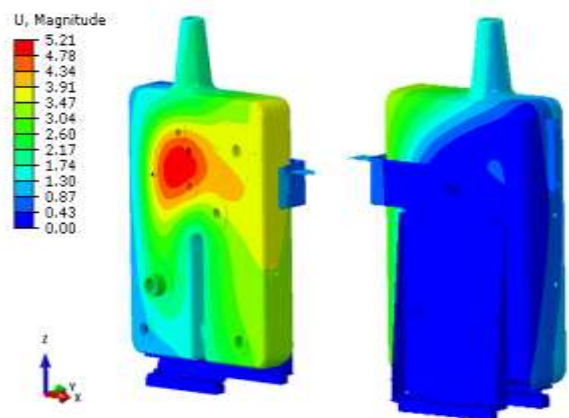


Fig-18 Load case 2 displacement plots

Load Case3

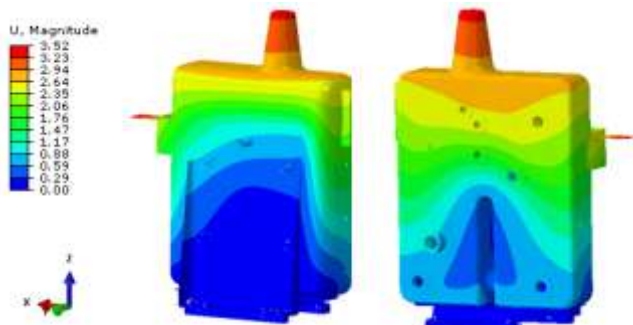


Fig-19 Load case 3 displacement plots

Load Case4

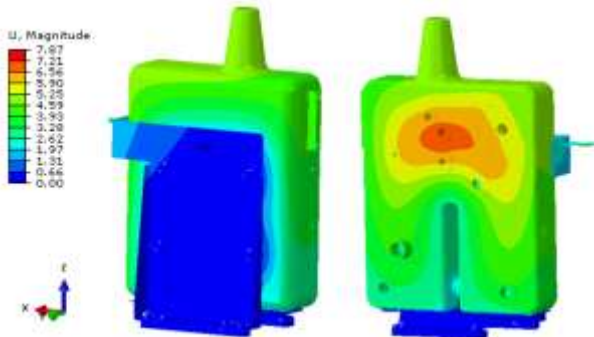


Fig-20 Load case 4 displacement plots

Load Case5

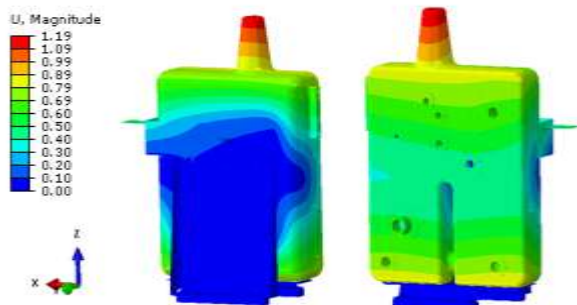


Fig-20 Load case 4 displacement plots

Load Case6

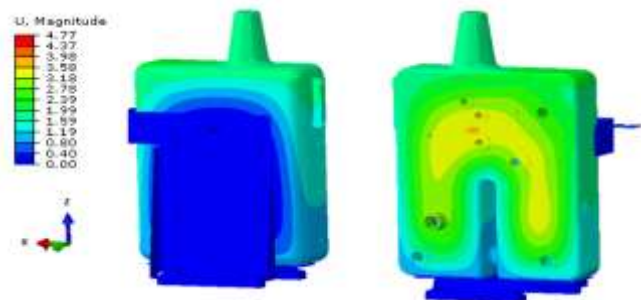


Fig-21 Load case 4 displacement plots

The variation on the displacement is shown on the legends on the left side. The red and blue color represents the max and min displacement respectively.

6.1 FEA Results for inertia load analysis for fatigue evaluation

The Stress amplitude was calculated from the stress of positive and negative load direction for X, Y, and Z loading directions. The stress amplitude is calculated on all the nodes using max principal stress. The highest stress amplitudes are probed and shown on the tank and on to the L brackets. The stress amplitude is then directly compared with the endurance limit of the material.

Load case 1 and 2: 5G X direction loading condition for stress amplitude

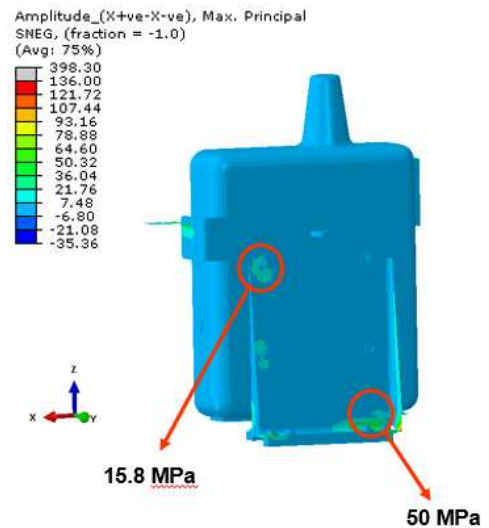


Fig-22 Load case 1-2 Stress Amplitude

Load case 3 and 4: 5G Y direction loading condition for stress amplitude

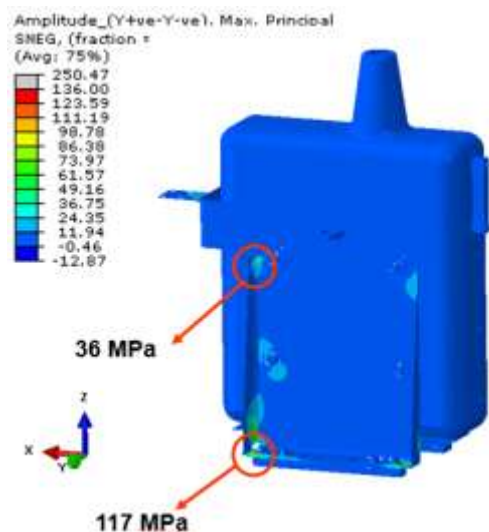


Fig-23 Load case 3-4 Stress Amplitude

Load case 5 and 6: 5G Z direction loading condition for stress amplitude

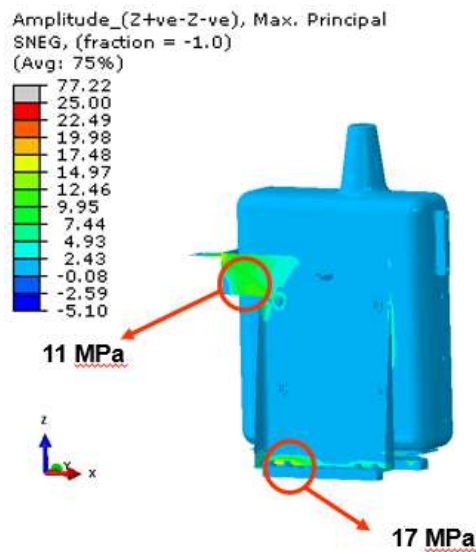


Fig-23 Load case 3-4 Stress Amplitude

6.2 Result Comparison between Original and Modified Design

The Table shows the FEA results for the original and modified design. In the original design the displacement and Stress in the Y direction are very high 20.51 mm and 274 MPa respectively.

Design iteration 3 the displacement and stress were brought down to 7.24 mm and 72 MPa respectively. The fatigue life calculation was also performed for Y direction which is the worst case in all 3 loadings.

Table -2: FEA Results Comparison

| Load Direction | FEA Comparison Result | | | |
|------------------|-----------------------|------------------------|-------------------|------------------------|
| | Original Design | | Modified Design 3 | |
| | Displacement (mm) | Stress Amplitude (MPa) | Displacement (mm) | Stress Amplitude (MPa) |
| 3G +ve and -ve X | 9.09 | 72 | 9.24 | 52 |
| 3G +ve and -ve Y | 20.51 | 274 | 7.24 | 72 |
| 3G +ve and -ve Z | 4.72 | 25 | 4.77 | 24 |

7. CONCLUSIONS

Structural FEA of the Hydraulic tank is performed. The Hydraulic tank is modeled from the Composite Material and supported by the L shape mounted bracket made of steel material. FEA was performed using Abaqus. Displacement plots and von Misses stress plots 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 2 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 of 2 million cycles. A total of 3 design iteration was done and for each design iteration, FE analysis was carried out by performing all the tasks mentioned above. Each time design changes were considered based on FE results and 3D models were created in design software. Design iteration 3 was finalized based on the FE result and considering the ease of manufacturing. The life for the design 3 is found to be more than expected 2 million cycles.

REFERENCES

- [1] 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
- [2] Gajendra G "Design and optimization of HTV fuel tank assembly by finite element analysis.
- [3] Michael Davis, Zhingxing fu, Qunhui Han, "optimizing fuel tank design for high-speed vehicles" June 9-12 2009.
- [4] Abhishek P1, Hardeep S M 2, Sarvocch G 3, Anil L, "Finite Element Analysis of Fuel Tank Mounting Bracket".
- [5] 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,

AUTHORS

Prakash M A, scholar, Cambridge Institute of Technology, Bangalore

Manjunath T V, Assistant prof, Cambridge Institute of Technology, Bangalore, Karnataka