Fatigue Life Prediction of Heavy Duty Vehicle Radiator under Pulsating Pressure Loading

Pravin Ganesh Menariya[1], Girish Narayan Kotwal[2]
Department of Mechanical Engineering
[1]PG Student,Vishwakarma Institute of Technology, Pune.
[2]Asst. Professor, Vishwakarma Institute of Technology, Pune.

Abstract*-Objectives: This study is held to find maximum stresses in the commercial vehicle radiator tank and header plate assembly subjected to pulsating pressure loading and to predict its life in cycles against the pressure loading. Methods/Analysis: This study uses Finite Element Method to find the maximum stresses and to predict the life cycle. Header tank and header plate are modeled in CAD software and meshed with good mesh pattern. Critical areas are arrested with fine meshing with better quality checking. Outcomes: Deformation and stresses at the component are studied well to evaluate Finite Element Analysis results. The radiator deformation is found at the tank inlet where area was maximum. Then von-mises stresses are measured at various locations, the maximum stress found at header plate was about 65 Mpa. Since the radiator component material was ductile von-mises stress criterion is employed. To calculate the pressure cycle life of the component, the material properties, loading conditions and geometry are required. Using the above stuff the pressure cycle life of the component is calculated in FEA software. Calculated pressure cycle life at max. stress point is found to be 28370 cycles. The virtual life meets the results with the tested one. Application/Improvements: Prototypes for the pressure cycle test can be reduced. Approximate pressure cycle life in header plate is estimated.*

Keywords-Finite Element Modeling, Header plate, PA66GF30, Radiator, Pressure cycle life, Radiator, Von-mises stress.

I. INTRODUCTION

Modern truck engine generates lot of Heat. During the combustion process 33% of the heat is converted into power to drive the vehicle and its accessories. Another 33% of the heat is pushed as smoke into the surrounding environment over the exhaust system. The remaining 34% of the heat is rejected from the engine by the cooling system[1]. Engine cooling system aids in dissipating the engine heat to the surrounding and the engine temperature is kept under controlled levels. Modern engine cooling system consists of a Radiator, Charge Air Cooler and Fan Shroud. Radiator is the main heat exchanger, where the engine coolant rejects heat to the passing air and again passed to the water jacket to absorb some more heat from the engine. Design of the radiator is becoming interesting due to higher operating pressure and temperatures. In the Radiator lifetime, it is subjected pressure thermal cycle loads, road vibration loads, creep, internal erosion and external corrosion[2]. Pressure cycle failure is one of the main contributor for the radiator failures rates. In this paper Finite element Analysis technique is used to understand the behavior of the heat exchanger due to pressure cycle loading.

II. MATERIALS AND METHOD

A. Construction of a Commercial Vehicle Radiator

Heavy Duty truck radiator consists of a tube, header, fin, gasket, tank and side piece. The construction of the Radiator assembly is briefly reviewed below[2]. Radiator core is the very heart of the heat exchanger. It consists of tubes, fins and side piece. Tubes are generally welded or extruded tubes. Welded tubes are used for the heavy duty application. Fins are also called as ambient fins, because it is positioned in the upstream air direction and in front of the grille. Fins may be louvered or non-louvered type based on the vehicle application. Heavy duty trucks louvered Fin configuration is employed. Side piece at the either end of the core, provides additional support and stiffness when the core is expanding in the side-side direction due to the internal pressure load. Radiator Tank Assembly consists of an inlet and outlet tank. The Top Inlet tank holds the heated coolant pumped from the engine before it passes over the radiator core. It has an inlet port to receive engine coolant.

A restocking port is to allow the user to add engine coolant. Some time it also houses the pressure cap. The bottom outlet tank that holds the cooled coolant before it is returned to the engine. It has an outlet port for the coolant and a drain cock. Header is the connection link between the tank and core assembly. It has number of slots equivalent to the total number of tubes. The tubes are inserted onto the slots in the header before the brazing process. The Header has a well portion to accommodate the rubber gasket. Gasket is positioned in the well, then tank is placed onto the gasket, the tank is compressed and the header tabs are crimped onto the tank.

B. Finite Element Analysis

Finite Element Analysis (FEA) is defined as Discretization of a Domain (Solid or Surface Geometry) by means of points called "Nodes" having flexibilities called degrees of freedom (DOF) and connected to each other by geometrical entities called "Elements" for the transfer of information. FEA is widely used numerical method to solve both simple and complex problems[9]. The whole domain is divided into smaller geometrical entities; the stiffness is calculated for each element and assembled to solve the stiffness equation[13]. Displacement is calculated and stress/strains are derived from the resulting deformations. FEA is widely used in automobile world to study basic structural problems, strength/stiffness studies, crash simulation etc. Three basic steps[6]are involved in FEA analysis, pre-processing, solution and post-processing. Pre-processing includes the Discretization or meshing of the structure, material assignment, loads and boundary condition. Solution involves the assembly and solving of stiffness matrix. Post processing includes analysis of the solution results.

III. FINITE ELEMENT MODELING

Geometric Cleanup

Computer Aided Design (CAD) geometry of the radiator model is built in CATIA CAD package and imported into a Finite Element (FE) modeling software. Native CAD or Universal CAD formats like STEP, IGES etc. can be used to import the geometry into the FE Modeling Software. In this study, the CAD model is exported as iges format7 from the CATIA software. The imported CAD geometry is thoroughly checked for any irregularities and imperfections7 using Geometric cleanup tools. Free edges in the geometry indicate gaps and improper connectivity of the model which needs to be corrected. When two or more surfaces sharing the same edges, they are called Non-Manifold surfaces, which also represent incorrect connectivity. Free edges, Non-manifold surface, Missing surfaces, unnecessary fillet lines and duplicate surfaces are repaired and removed. Symbols in the tank are removed using defeature options. Once the Radiator geometry is clean we can advance to the subsequent phase of Finite element modeling. Clean geometry is a vital prerequisite to have a better mesh pattern and accuracy of simulation results. It also saves lot of computational time & effort8.

Discretization of Domain

Commercial Vehicle Truck Radiator system consists of a tube, header, fin, gasket, tank and side piece2. To lessen the computing time, here it is decided to analyze only the upper tank and header plate assembly in the analysis. As the fluctuating loads will occur mostly in the upper tank of the radiator, it would become more complex to analyze whole radiator assembly. The finite element model contains the stiffness of the component taken under study.

Pressure cycle failure is one of the main reasons of failure. The model consists of radiator plastic tank and header plate and it is shown with cross section of tank and plate. The plate is clinched to the tank. It is shown in fig. 1

As no relative sliding is permitted and both are the different materials, so clinched is being better option for joining the two components. The two connections for joining the two components are modeled in Hypermesh V1210. The rubber gasket is not considered in the model, as it decreases to 60% of its original thickness and it is there only for preventing leakages. It also isolates the two components form vibration.

Element Definition

The vehicle radiator tank geometry is meshed using both, the shell and the solid elements. The elements used for Abaqus 6.13. The component is modeled in Hypemesh V12.

Here, it is desired to apply Tie contacts, for applying tie contacts the solid facets are required. The whole model is firstly meshed with Tria elements and then it is converted to 10 nodes tetra element as 10 nodes tetra having less stiffness than 4 nodes tetra element.

Finite Element Model –Sizing & Model Quality

The model is consisting of nearly 85000 nodes for lessen the solving time and it reduces the computing time for the solver. The overall element size is kept as 3 mm and it is varied along the model in the range of 0.3 mm to 10 mm. Near critical areas fine mesh is applied. Both the components are meshed separately; it has started with tria elements of first order. As the shell meshing is done, it is taken for different quality checks like min. size, max. size, jacobian, warpage, aspect ratio and tet-collapse in case of 3D elements11. If the mesh does not meet the criteria it is re-meshed again to get the desired criteria. Then the shell mesh is converted to solid mesh after changing order for the shell mesh to 2nd order.

Then it becomes 10 nodes tetra elements

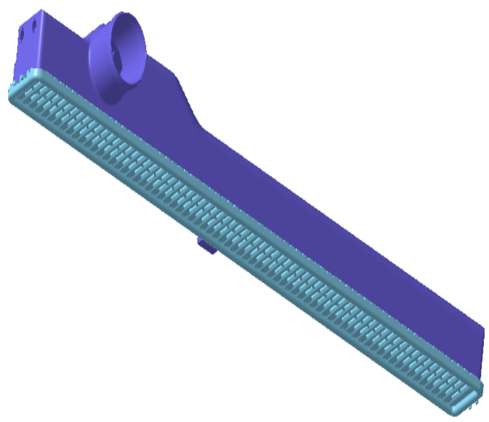

Figure 1 a. Radiator upper tank and header plate assembly.

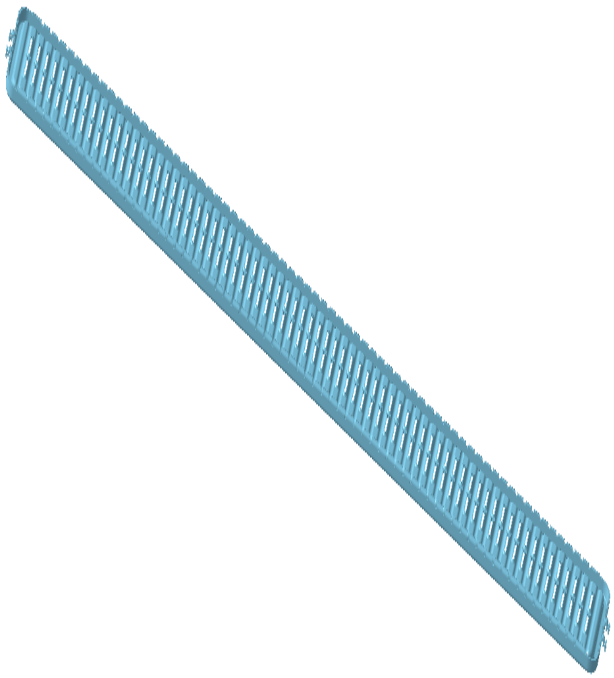

Figure 1 b. Header plate with holes

Material Properties

The header plate section from the model is made up of Al 3003. It is from the class of wrought Aluminum. The radiator plastic tank is made up of PA66GF30. It is polyamide 66 with glass fibers proportion up-to 30%. Young's modulus for Plastic and the Aluminum are defined. Poisson's ratio for both the components is defined. Once the material properties are given, then applying boundary conditions is next stage.

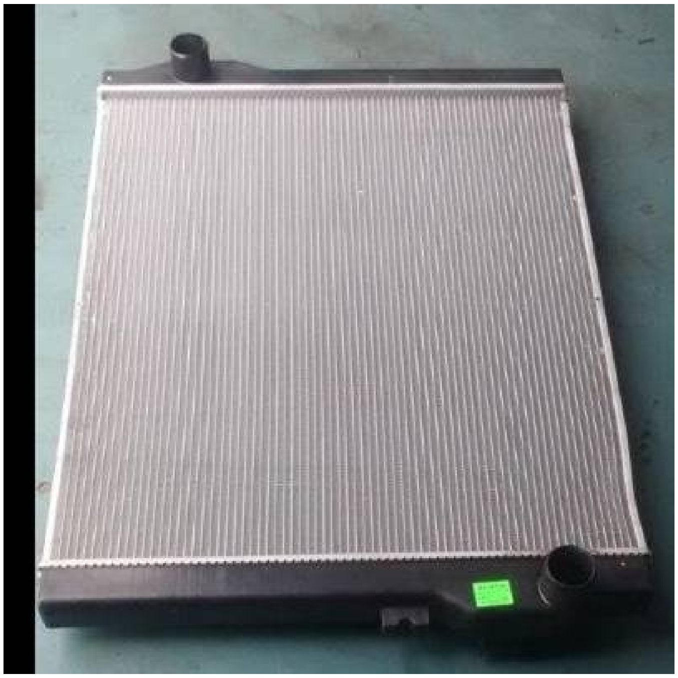

Figure 2 a. Radiator with core and tanks assembly

Figure 2 b. Radiator Upper tank top view

Loading, contacts and Boundary conditions

As no relative sliding is permitted throughout the work, the components are attached with tie joints using solid facets of the solid meshing.

The whole inner section of the tank is subjected to load of 2.5 bar. Here the load is cyclic; it is needed to apply to load cases. One for loading and another for unloading. In loading elemental DLOAD of 0.25 Mpa is applied at inner portion of the both components. For unloading the on-load OP option is marked for the same elements selected before for the first load-case.

The plastic tank is bolted to the assembly. It is having two holes on both sides for constraining purpose. It is shown in fig.3b. Also the lower end of the header plate is having several openings; these openings are for attachment with the core tubes. Attaching the tubes will give more stability to the component. To get more realistic results, the holes in the header plate are also constrained. It is shown in fig. 3c.

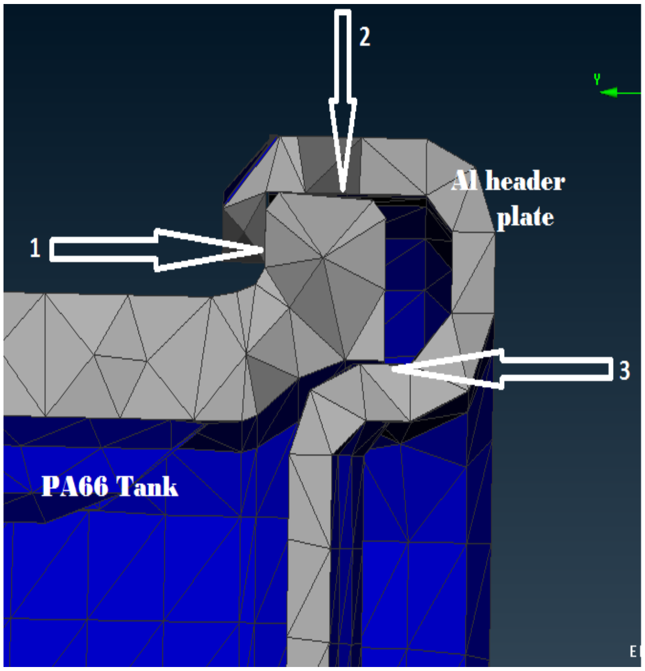

Figure 3 a. Contact locations between plate and tank

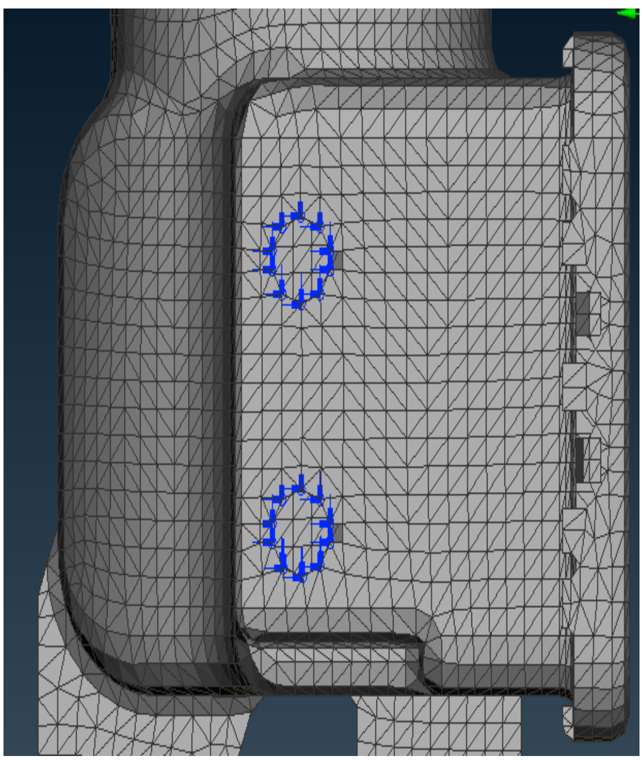

Figure 3 b. Constraining bolt locations at the tank

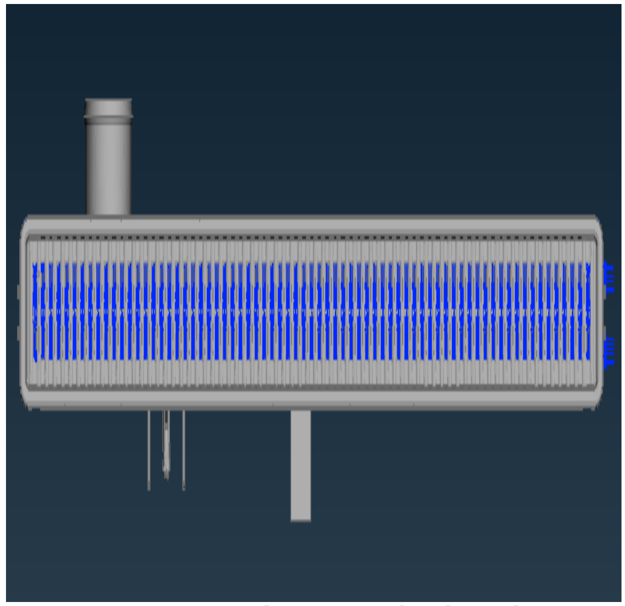

Figure 3 c. Constraining locations at the plate-tube cross section

Step manager for Abaqus 6.13

In this case, the problem is of material non-linearity, as it consists of Plastic. The loading is cyclic, the load varies from 0 bar to 2.5 bar and again it comes down to 2.5 bar to 0 bar.

Defining the step manager in load-step, some step parameters are to be defined. The parameters are Increment; which defines the increment in cutbacks while solving.

Abaqus 6.13 uses the Newton-Raphson method to solve the problem iteratively. Every-time the solution tries to get the deformed shape of the model with minimum residual force.

Non-linear geometry is marked on for geometrical non-linearity. And the model is un-symmetric, it is also checked.

Two steps are needed to be defined here, Loading and Unloading to define a cycle of loading.

In loading step, loadcols are defined and for Output blocks the required output is defined. The output required for node was Reaction force and Displacement. The output require for element was Stress.

In Unloading step, the above procedure is repeated and Load-op option is marked to unload the loadings. Then the model is taken for solution in Abaqus 6.13.

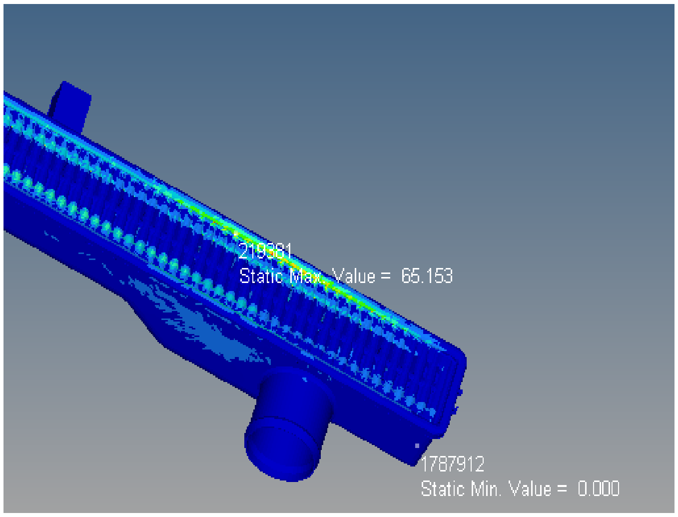

Figure 4 a. Maximum stress location

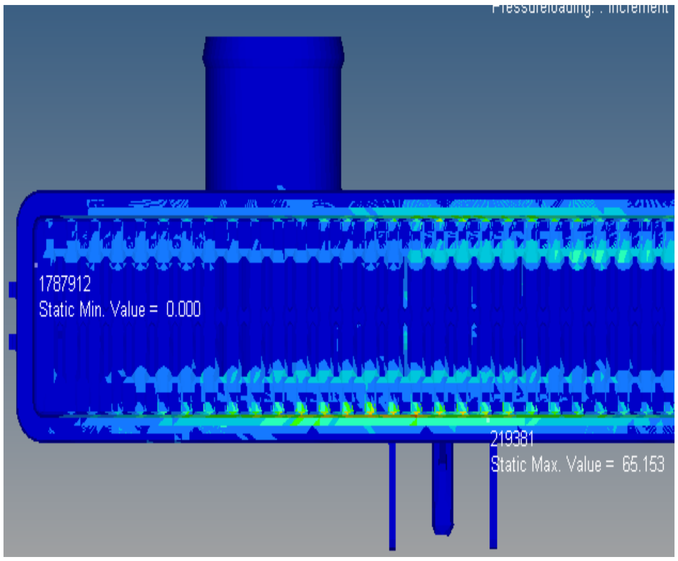

Figure 4 b. Maximum stress location- bottom view

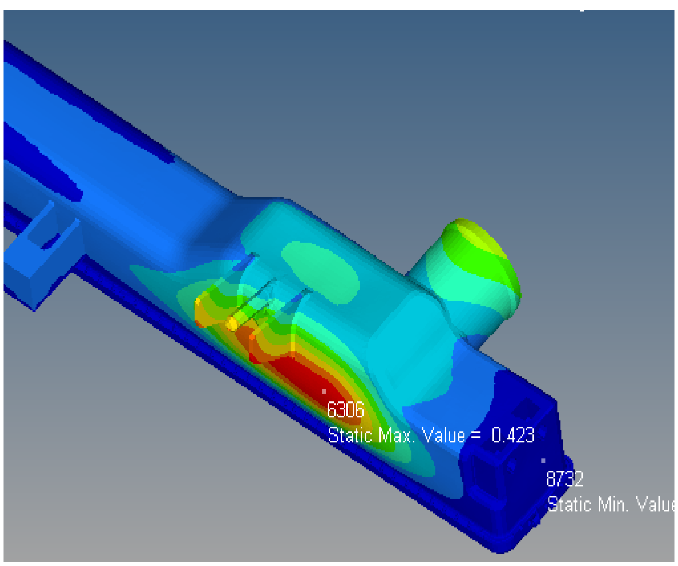

Figure 5 a. Maximum displacement location

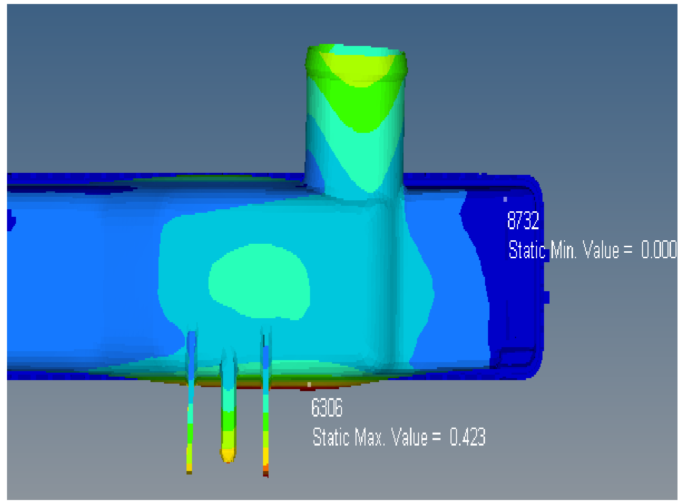

Figure 5 b. Top view – maximum displacement

Fatigue solution in FEMFAT

The model from the Abaqus solver is taken for the Fatigue life solution. Due to insufficient fatigue properties of both the materials the upper lower method is used for the fatigue life calculation. This method needs the maximum and minimum loadings on the model, geometry of the model and material behavior. The nodes of each component are selected separately to give separate behavior according to the material.

Material class is to be chosen to define the material behavior.

For Al3003 the material class chosen was wrought aluminum and for PA66GF30 the class chosen was reinforced plastics.

By choosing the material class, we allow the software to take some of the insufficient data from its database. Here, only the ultimate tensile strength is defined and software automatically calculates the Young's modulus and other properties with the use of material class.

IV. EXPERIMENTAL TESTING OF RADIATOR

The experimental testing was done at TATA TOYO Radiator, Pune. The experimental testing uses the SAE standard for Surface vehicle recommended practice J159712

The company promises the warranty of the radiator up-to 10000 cycles or 2.5 lac kilometers. After testing up-to near about 10000 cycles and if no relative damage is seen in the component; the test rig is switched off and the component is declared passed for the fatigue life. Here, to minimize the time for the testing the load applied is about 1.5 to 2 times of the actual loading conditions.

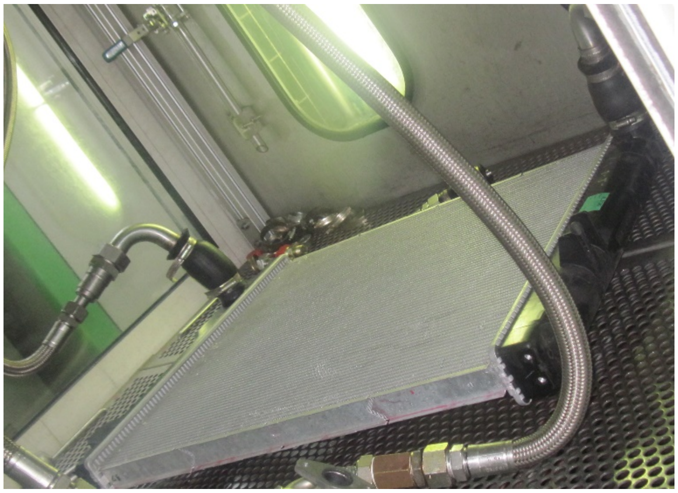

Figure 6. Experimental testing

TEST CONDITIONS & OBSERVATIONS

Pressure Cycle Test				
Test Parameter	**Unit of Measurement**	**Test Specification**	Sample 1	Sample 2
Ambient Medium		Air	Air	
Ambient Temperature	Deg c	Ambient	Amb	
Test Medium	-	Ethy-gly	Ethy-gly	
Test Medium Temperature	Deg c	80	80	
Static Pressure of the Test Medium	Bar			
Lower Limit of Pressurization	Bar	0.2	0.2	
Higher Limit of Pressurization	Bar	1.4	1.4	
Pressure hold time at peaks	s	0	0	
Pressurization rate	Bar/sec	-	-	
Frequency of Pulsation	cycles/min	6	6	
No of Pressure Cycles	Cycles	10000	9200	
Flow Rate of Test Medium	l/h	3000	3000	
Pressure Wave Type	Sine	Sine	Sine	PASS

Figure 7.Experimental testing report

V. RESULTS AND DISCUSSIONS

Here, the first solution is of solving for stresses and displacement. The maximum stress is at the header plate location and is found about 65 Mpa. The maximum deformation is found near the inlets of the plastic component and found about 0.4 mm. During the pressuring the tank expands sideways. The maximum Von- Mises stress of the commercial vehicle radiator is 65 Mpa at the header plate location.

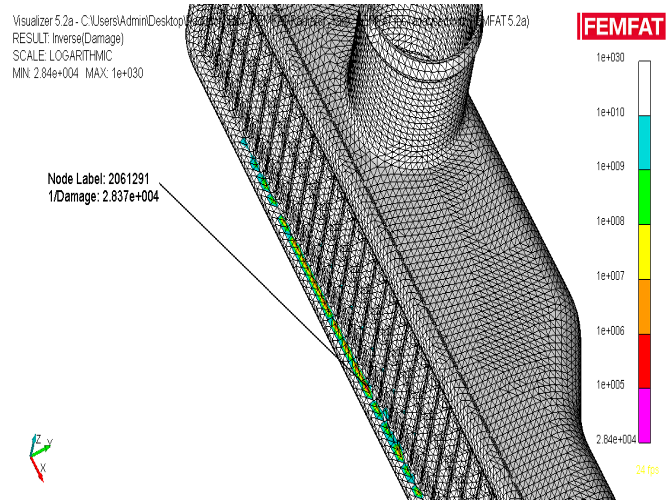

Figure 7. FEMFAT minimum life location location

After, solving for the fatigue life in FEMFAT, the life of the component is found to be near about 28000 cycles which also says that the component will live more than 10000 cycles; which is the requirement of experimental testing.

Here, while using FEMFAT due to insufficient properties, the material behavior is selected to the relative material behavior. Total model is not taken under study and the exact stiffness provided by the tubes to the header plate and directly constraining the header tube differs. Due to these factors the fatigue life does not reach the exact value which experimental results will give.

The experimental testing is going up-to 10000 cycles and declares component to be safer and the software is going up-to 28000 cycles and showing the damage location at the header plate also declares the component to be on safer side.

VI. CONCLUSION

Finite Element Analysis helps in approximating the virtual validation of the components, which are going to be tested experimentally. The upper tank and header plate of the component are modeled in Hypermesh and solved in Abaqus for Non-linear static problem. The step manager for Abaqus is set up in Hypermesh. The results are analyzed for the radiator component. The maximum value of deformation is found about 0.4 mm at the plastic tank inlet location. Deformation plot shows the radiator tank expands sideways. The maximum Von-Mises stress is found at header plate of 65 Mpa.

The solution for fatigue life of the model shows the life of 28000 cycles before the damage at the header plate location.

The experimental testing shows that the model passes 10000 cycles life and it is said to be passed component.

Both the experimental and Virtual results for the life of the component says that component is passing the criteria of target life of 10000 cycles. It shows that, based on the finite element analysis stress results we can calculate the pressure cycle life of the commercial vehicle radiator. The FEA and fatigue life analysis virtually meets the experimental criteria of the Radiator. We can say that, the FEM results will help in reducing the number of prototypes to be tested experimentally and helps in cost reduction.

REFERENCES

[1] Eton Yat-Tuen Ng, "Vehicle Engine Cooling Systems: Assessment and Improvement of Wind-Tunnel Based Evaluation Methods",

[2] Jochen Eitel, Gerald T. Woerner, "The Aluminum Radiator for Heavy Duty Trucks",SAE,1999-01-3721.

[3] K. Priyadharshini," Finite Element Analysis of Radiator Fins to increase the Convection

[4] Efficiency of Radiator by using Al Alloy, Cu and Brass Material", ISSN: 2393-8447

[5] A.Muniappan, C.Thiagarajan, Taukir Azam, X.Joseph Raj," Design Modification & Analysis of Header Tube Joint In Heat Exchanger", ISSN: 2319-8753

[6] N.A.FLACK, R.A.SMITH,"Fatigue life prediction of structural steel under service loading",Int. j. Fatigue. 6 no.4. (1984),pp 203-210.

[7] Sinan Eroglu,Ipek Duman,"Durability analysis of Heavy Duty Engine Exhaust Manifold Using CFD-FE coupling",SAE,2016-01-0375.

[8] Abolhassan khosrovaneh,Ravi pattu,"Discussion of Fatigue Analysis Techniques in Automotive Applications",SAE 2004-01-0626.

[9] Nitin G.,Deshpande S,Bedekar S, Thithe A."Practical Finite Element Analysis",2008,1st edition.

[10] Altair Hypermesh user Guide 12.0.

[11] Abaqus 6.13 user Guide

[12] SAE standard for Surface vehicle recommended practice J1597.

[13] Silva EPD .Application of Finite Element Method for structural analysis in a coffee harvester. Engineering. 2014; 123:138-47.