Indian Journal of Science and Technology, *Vol 9(33), DOI: 10.17485/ijst/2016/v9i33/98483, September 2016*
ISSN (Print) : 0974-6846
ISSN (Online) : 0974-5645

Fatigue Life Prediction of a Commercial Vehicle Radiator under Internal Pressure Cycling Loading

P. Robin Roy[1]*, V. Hariram[2] and M. Subramanian[3]

[1]Department of Automobile Engineering, Hindustan Institute of Technology and Science, Hindustan University, Chennai – 603103, Tamil Nadu, India; robin_cae@hotmail.com
[2]Department of Mechanical Engineering, Hindustan Institute of Technology and Science, Hindustan University, Chennai – 603103, Tamil Nadu, India; hariram@hindustanuniv.ac.in
[3]Department of Automobile Engineering, BS Abdur Rahman University, Chennai - 600048, Tamil Nadu, India; ramani@bsauniv.ac.in

Abstract

Objectives: The study is conducted to find the hot spot stresses in the commercial vehicle heavy duty radiator subjected to internal pressure loading and predict the pressure cycle life. **Methods/Statistical Analysis:** The finite element analysis technique is employed to predict the hot spot stress and pressure cycle life of the radiator. Finite element model of the radiator is built by the including the mass and stiffness of the radiator component. Header tube joint is the critical area which affects the pressure cycle life and needs to be analyzed. The critical joint area is captured with a good mesh pattern and better element quality to avoid stress singularities and spurious stresses. **Findings:** Deformation and stresses are the studied in detail to evaluate the finite element analysis result. The radiator deformation shows core side to side expansion which correlates well with the pressure cycle test. Then von-mises stresses are measured at various locations, the header tube stress is found to at 75MPa. Since the radiator core material is ductile material, von-mises stress criterion is employed. To calculate the pressure cycle life, three components such as geometry, loading and material are required. The geometry component is represented by the stress concentration factor (kt) at the header tube joint. The Kt value can be measured based on test failure history. The loading is represented by the internal pressure loading. The S-N curve and cyclic stress-strain properties for the aluminium material is required. Once the above three parameters are known, we can calculate the pressure cycle life at the header tube joint using strain life approach. Calculated pressure cycle life at the critical joint is found to be 150,000 cycles to failure. The virtual life meets the typical pressure cycle life at the header tube joint. **Application/Improvements:** Prototypes for the pressure cycle test can be reduced. The approach identifies the weak areas early in the design phase. Approximate pressure cycle life at header tube joint is estimated.

Keywords: Critical Joint, Core Deformation, Finite Element Modeling, Pressure Cycle Life, Radiator, Von-mises Stress

1. Introduction

A 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 through 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 challenging due to higher operating pressure and temperatures. In the Radiator lifetime, it is subjected to pressure thermal cycle loads, road vibration loads, creep, internal erosion and external corrosion[2]. Pressure cycle failure is

---

*Author for correspondence

one of the major 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

2. Materials and Method

2.1 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[3]. 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 through the radiator core. It has an inlet port to receive engine coolant.

A replenishing port is to allow the user to add engine coolant. Some time it also accommodates 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.

2.2 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[4]. The whole domain is divided into smaller geometrical entities; the stiffness is calculated for each element and assembled to solve the stiffness equation[5]. 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.

3. Finite Element Modeling

3.1 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 step format[7] from the CATIA software. The imported CAD geometry is thoroughly checked for any irregularities and imperfections[7]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 proceed to the next step of Finite element modeling. Clean geometry is an important prerequisite to have a better mesh pattern and accuracy of simulation results. It also saves lot of computational time & effort[8].

3.2 Discretization of Domain

Commercial Vehicle Truck Radiator system consists of a tube, header, fin, gasket, tank and side piece[3].To minimize the computational time and modeling time, a half symmetry model of the Radiator is built. The finite element model includes the stiffness of the complete radiator assembly to simulate the exact physical behavior of the radiator. Also the model takes into account of the header tube joint as shown in Figure 1, which is critical location studied during the pressure analysis[9].Commercial vehicle radiator is manufactured by brazing process[10]. The Brazing joints are considered by merging the nodes at the joint location. Tube, fins, header and core side are assembled and then brazed through a Controlled Atmosphere Brazing furnace. The header tabs in the header, crimps with the plastic radiator tank onto the core system. The crimping tab is modeled to include the header-tank joint stiffness. HyperMeshV.13[11] is used for finite element modeling. The radiator tank is bolted to the radiator frame channels by means of rubber isolator mounting. The rubber isolators are modeled and connect to the tank assembly. The rubber isolator helps to isolate the vibration to the core and also helps in the core expansion.

3.3 Element Definition

Commercial Vehicle Radiator geometry is discretized using both the Solid and Shell Elements. The Finite elements used in the Optistruct[12] are validated and verified by the NAFEMS Benchmark[13].Tank is meshed using second order tetrahedron elements (10 node element) for better stress accuracy. Tube, header, rubber gasket, core side and fin are modeled using combined hexagonal (eight node element) and penta element (six noded elements) respectively.

3.4 Finite Element Model –Sizing& Model Quality

The complete model size is restricted, close to one million nodes for quick solving and which reduces the computational time. The global element size is kept less than 5mm near the critical areas; coarse mesh is used in other regions. Tube-header joint, headers, tube are considered as areas of interest. Core element size away from the header region has coarse mesh. Tank is meshed with an average element size, to study the stresses in the rib region. Once the complete geometry is meshed with elements, the model is checked for edges, duplicates and T-connections. Once the mesh connectivity is established, then element are checked for various quality measures like min/max interior angles, Jacobian, aspect ratio, warpage, tetra collapse etc for both 2D/3D elements[11]. If the mesh does not meet the desired criteria, the mesh pattern is re-meshed to meet the quality criteria. Once the Finite element model as shown in Figure 2 meets the desired criteria and mesh pattern, then problem is setup for the solution.

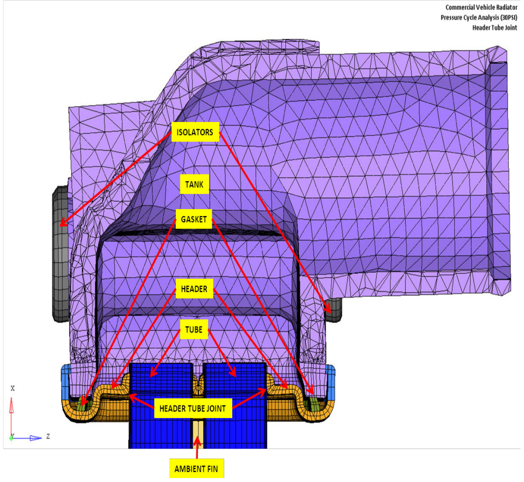


Figure 1. Header Tube Joint in a Section View.

3.5 Material Properties

The Core system including the tube, fins and core side are made from Al3003[3]. The radiator plastic tank is an injection molded from fiberglass-reinforced Nylon PA66 and the Gasket is made from EPDM material[3]. Young's Modulus and Poisson ratio are defined for the Al 3003 and Plastic tank. Fin geometry is modeled as block to reduce model size and computational time. Equivalent orthotropic property of the fin geometry is calculated and assigned for the fin blocks. Young's Modulus in three principal and shear planes is considered for the analysis. After material assignment to the radiator components, we need to apply the loads and boundary condition.

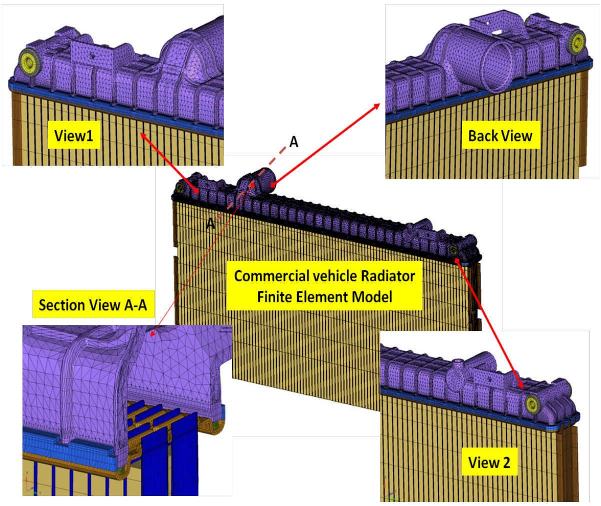
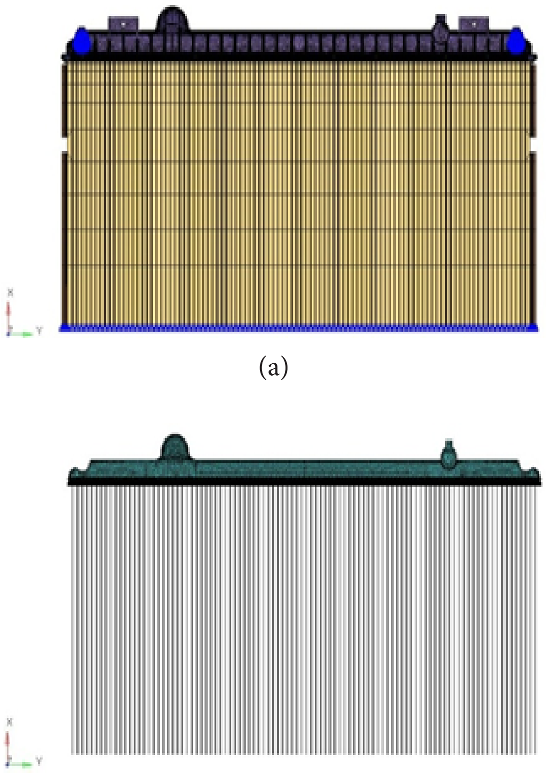


Figure 2. Finite Element Model of the Commercial Vehicle Radiator.

3.6 Boundary Conditions

Symmetrical boundary condition is assumed in the pressure analysis as shown in Figure 3A.The radiator model is symmetrical about YZ plane. The normal axis X is constrained in the YZ symmetrical plane. Flat faces of the isolators are constrained in all the DOF. During the truck life, the radiator is subjected to cyclic pressure load or pulsating load. This cyclic load induces stresses in the radiator structure. The max pressure amplitude is taken as the applied pressure load. The internal wetted surface of the tank, header and tubes are assigned with the applied pressure load. Typical pressure of 30PSI is applied to the internal pressure surface as shown in Figure 3B.Generally for heavy duty commercial vehicle radiator the applied pressure will be 1.5times the operating pressure. Linear pressure analysis is performed using sparse matrix solver.

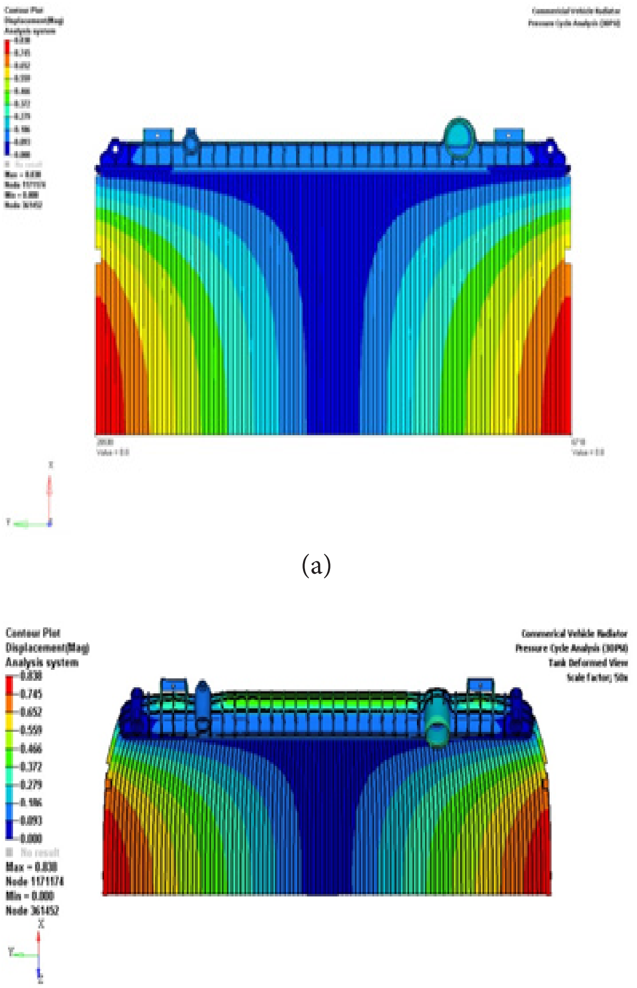

(a)

(b)

Figure 3. Boundary condition (A) and Internal Pressure surface (30 psi) (B).

4. Results and Discussion

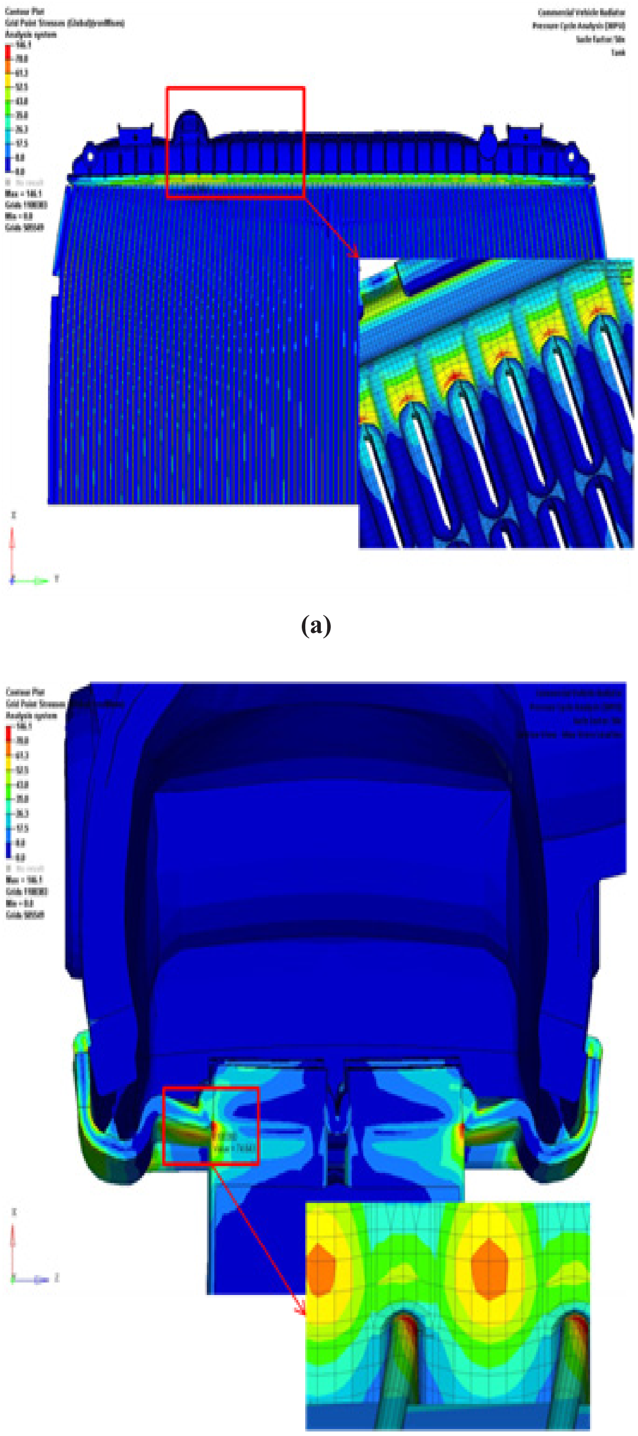
Once the solution is run successfully care should be taken in understanding the deformation plot of the radiator assembly. During the pressure pulsation loading, the radiators expand in the side to side direction. Deformation plot of the radiator assembly is shown in the shown in Figure 4A. The maximum deformation of 0.8mm occurs at the core sides. Deformation plot shows the symmetrical distribution of contour on both sides of the radiator. It shows that pressure load is correctly applied in the simulation model. Plastic tanks deforms in the vertical x-direction due the fixation at the isolator location. Tank deforms about 0.5mm as shown in Figure 4B.



(a)

Figure 4. Deformation plot for Radiator at 0.8 mm (A) and Tank for 0.5 mm (B).

The Max Von-Mises stress of the commercial vehicle radiator is 81MPa at the header location (Figure 5A). Maximum stress at the header tube joint corner is identified as 75MPa (Figure 5B). Header tube joint is typical area of interest for the pressure analysis[6]. The radiator tank stresses are close to 16MPa at the rib location. The pressure cycle life of header tube joint is calculated using strain life approach[14], the pressure cycle life at the header tube joint is found be greater than 150,000 cycles[15].


(a)

(b)

Figure 5. Maximum Von-Mises stress plot (A) and Header tube joint stress (B) in MPa.

5. Conclusion

Finite Element Analysis[16]aids in the visualization of the commercial vehicle radiator behavior subjected to pressure cycle loads. A half symmetry Finite Element model is modeled using Hypermesh and linear pressure analysis is performed using Optistruct. Deformation and Stress[17] results are studied for the radiator assembly. The max magnitude of the deformation is at 0.8mm at the core sides. Deformation plot shows that the radiator is expanding in the side-side direction. Max displacement is seen at the core ends. Core expansion of the header tube joint causes a high stress gradient. Tank deforms in the vertical direction due to constraints in the isolator location. Rib structure in the tank provides stiffness to resist the deformation due to the internal pressure load. Tank deformation is 0.5mm. Max Von-Mises stress of the commercial vehicle radiator is 81MPa (Figure 5a). Header tube joint corner stress is identified as 75MPa (Figure 5b). The radiator tank stresses are close to 16MPa at the rib location. Pressure cycle life at the header tube joint is found be greater than 150,000 cycles which meets the target life requirement of the commercial vehicle radiator. Based on the Finite Element Analysis stress results we can compute the pressure cycle life of a commercial vehicle Radiator.

6. References

1. Bennett S. Medium and Heavy Duty Truck Engines, Fuel & Computerized Management Systems. Cengage Learning: 2008; 960 pp.
2. Beatenbough PK. Radiator Design and Material recommendation Based on Failure Mode Analysis. Society of Automotive Engineers.1992; 1–8.
3. Jochen Eitel J, Gerald T, Woerner W, Horoho S. The Aluminum Radiators for Heavy Duty Trucks. Society of Automotive Engineers. 1999; 1–10.
4. Robert D, Cook C, David S, Malkus M, Michael E, Plesha P, Robert J, Witt W. Concepts and application of Finite element analysis, 3rd ed. John Wiley & Sons. 1988.
5. Silva EPD, Silva FMD, Magalhaes RR. Application of Finite Elements Method for Structural Analysis in a Coffee Harvester. Engineering. 2014; 123:138–47.
6. Dheeraj S, Sabarish R. Analysis of Truck Chassis Frame Using FEM. Middle East Journal of Scientific Research. 2014; 20(5):656–61
7. Li C, Fan S, Shi M. Preparation of CAD Model for Finite Element Analysis. International Conference on Computer,

Mechatronics, Control and Electronic Engineering (CMCE), Changchun. 2010. p. 491–4.

8. Nitin G, Deshpande S, Badekar S, Thithe A. Practical Finite Element Analysis, 1st ed. Pune: Finite to Infinite. 2008.
9. Millard G. Analysis of Durability Characteristics of Heavy Duty Radiators. Society of Automotive Engineers. 1977; 1–8.
10. Miller WS, Zhuang L, Bottema J, Wittebrood AJ, Smet PD, Haszler A, Vieregge A. Recent development in aluminium alloys for the automotive industry. Materials Science and Engineering. 2000; 280(1):37–49.
11. Altair® HyperMesh, User Guide, Version 13, Altair Engineering Inc. 2015.
12. Altair® Optistruct, User Guide, Version 13, Altair Engineering Inc. 2015.
13. Macneal RH, Harder RL. A Proposed Standard Set of Problems to Test Finite Element Accuracy, Finite Elements in Analysis and Design, 1985, 1(1), pp.3–20.
14. Abolhassan K, Pattu R, Schnaidt W. Discussion of Fatigue Analysis Techniques in Automotive Applications. Society of Automotive Engineers. 2004; 1–12.
15. Reemsnyder HS. Constant Amplitude Fatigue Assessment. Society of Automotive Engineers. 1979; 24(5):309–27.
16. Kim KS. Analysis of the Powder Forging Processes for Vehicle Timing Belt Pulley by Finite Element Analysis. Indian Journal of Science and Technology. 2015 Jan; 8(S1):1–6.
17. Kumar S, Hasan N, Kumar H, Kumar A. Finite Element Analysis of a force Transducer. Indian Journal of Science and Technology. 2011 Oct; 4(10):1–2.