Failure Analysis of Radiator Corner Tubes Subjected to Internal Pressure Cycle Loading

Rajdeep Pawar, Mihir Bangale, Vedant Parulekar, Shravan Shetty, N.P Sherje

Mechanical Department, Savitribai Phule Pune University

Abstract

Commercial radiators are subjected to internal pressure cycle loading during their service period. Due to these operating conditions, radiator assemblies are prone to failure at the header tube joint. Thus, to analyse the aforementioned failure, the radiator was modelled and subsequent FEA was performed. Deformation and stresses are studied in detail to evaluate the finite element analysis result.

Introduction

A lot of heat is generated in a modern truck engine. During the combustion process, only 1/3rdof the total power is used to drive the vehicle and its accessories. Rest of the power is converted into heat energy. About half of this heat is eliminated as smoke through the exhaust system. The remaining heat is rejected from the engine by the cooling system. Engine Cooling System (ECS)promotes the proper dissipation of the engine heat to the surrounding, thereby keeping the engine temperature under controlled levels. Modern ECS consists of the following major components:

1) Radiator
2) Charge Air Cooler
3) Fan Shroud.

The primary heat exchanger in the system is the radiator, where the engine coolant rejects heat to the passing air, and again passed through the jacket of cool water to absorb some more energy in the form of 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 the following operating conditions:

1) Pressure thermal cycle loads
2) Road vibration loads
3) Creep
4) Internal erosion
5) External corrosion

The major contributor for the radiator failure rates is pressure cycle failure. In this paper, Finite element Analysis technique is used to understand the behaviour of the heat exchanger due to pressure cycle loading.

Construction of a Commercial Vehicle Radiator

The primary components of a commercial vehicle radiator are the top tank, header plate, bottom plate, core supports, tubes, and fins. The radiator core comprises of the tubes and fins, arranged alternately one after the other. The major heat transfer process takes place in the core. The core supports are placed on either side of the core. During its operation, the core tends to expand under thermal loading conditions. The core supports provide mechanical support to the core during this condition. Tubes are generally welded or extruded tubes. Welded tubes are used for the heavy-duty application. Fins are positioned in the upstream air direction and in front of the grille. This enables maximum heat transfer rates at the fins, leading to efficient functioning of the radiator.Radiator tank assembly consists of an inlet and outlet tank. The top tank is the inlet tank. The heated coolant pumped from the engine enters this tank, and through this it is passed to the radiator core. An inlet port is provided to the top tank to receive the incoming coolant fluid.The bottom outlet tank holds the cooled coolant before it is returned to the engine.To bring down the temperature of the coolant before returning to the engine, fins are provided. Fins are arranged in an alternate fashion between two tubes. The header plate is used to hold together the entire assembly. It consists of a number of slots equal to the

number of tubes. The tubes are fit into the slots and then joined to the header using brazing process. The header is joined to the core support and the tank using crimping procedure.

Modelling

In order to perform FEA, the components need to be modelled on a CAD software and converted to a format that is readable by the FEA Software. Thus, individual components of the radiator were modelled and assembled in CATIA V5 and then converted into STP format to be meshed in Altair HYPERMESH.

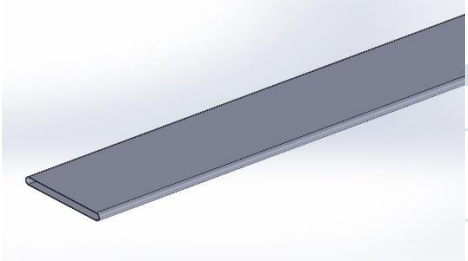

Fig. 1 Tube

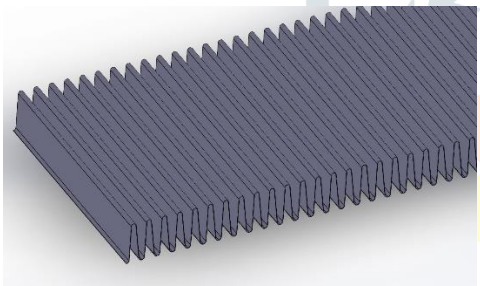

Fig. 2 Fins

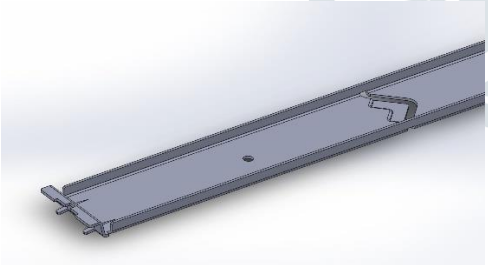

Fig. 3 Core Support

Fig. 4 Header Plate

Fig. 5 Radiator Assembly

Finite Element Analysis

Finite Element Analysis (FEA) is a computerised method for predicting how a component/product reacts to real world forces, vibration, heat, fluid flow and other physical effects. The primary objective of FEA is to reduce the amount of time and money spent on prototyping and testing, and optimize the product in the design phase itself. FEA consists of three basic steps:

1. Pre-processing
2. Solving
3. Post-processing

The first step, i.e. pre-processing involves discretization of the required domain into a number of divisions called “elements”. The common point between two elements is called a “node”. The results of the analysis such as stress, displacement, etc. are obtained at the node points. The entire process of dividing the structure/domain into a number of small elements is called “meshing”. Pre-processing also includes providing input parameters like force, pressure, etc. In the second step, the partial differential equations formed according to the specified problem statement are solved to ultimately obtain a solution to the problem. The third step, i.e. post-processing involves displaying the results in a form that is understandable to the user. The post-processor displays the data in the form of contours, graphs, tables, etc.

1. Geometric Clean-up

Computer Aided Design (CAD) geometry of the model is built in CATIA V5 and imported into a Finite Element modelling software, HyperMeshV13. The imported CAD model is checkedthoroughly for irregularities and

imperfectionsusing geometric clean-up tools.Clean geometry is an important prerequisite for having a better mesh pattern and accuracy of simulation results. It also saves lot of computational time & effort.

2. Discretization of Domain

Commercial vehicle radiator system consists of a tube, header plate, finand core support. The model considers the header tube joint, which is a critical location studied during the pressure analysis. The brazing joints are taken into account by merging the nodes at the joint location. Fins, tubes, header plate and core support are assembled and then brazed through a controlled atmosphere brazing furnace. The tabs in the header plate attaches with the plastic radiator tank using crimps onto the core assembly. The tab used for crimping is modelled to include the stiffnessof header-tank joint.

3. Model Quality and Sizing

The element size is kept less than 1mm near the critical areas.Coarse mesh is used in other regions.The complete model size is restrictedclose to one million nodes for quick solving andreducing the computational time. Tube-header joint, header plate, tubes are considered as the areas of interest. The region away from the header plate has coarse mesh. The quality measuresfor both 2D/3D elements are as follows:

Quality parameter	*Value*
Maximum interior angles	120
Minimum interior angles	20
Jacobian	0.7
Aspect ratio	5
Warpage	15
Tetra collapse	0.1
Skew	70

Ifthe desired criteriaare not met, the mesh pattern is meshed again to meet the quality requirement.

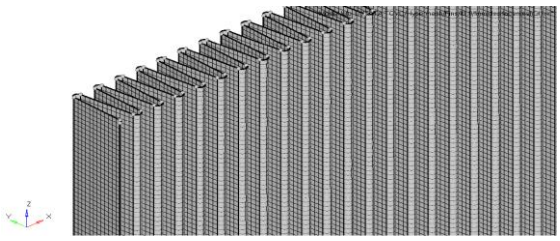

Fig. 6Fins

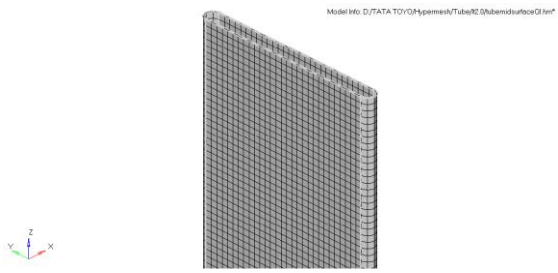

Fig. 7Tubes

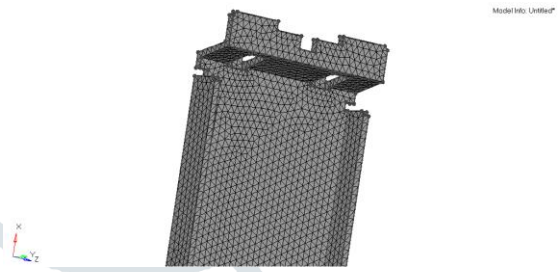

Fig. 8 Core Support

Fig. 9Header Plate

4. Material Properties

The core system including the tube, fins and core support are made of Aluminium 3003. The radiator plastic tank is made from injection moulded fiberglass-reinforced Nylon PA66 and the Gasket is made from EPDM material. Poisson ratio and Young's Modulus are defined for the Al 3003 and Plastic tank. Fin geometry is modelled as a block to reduce thecomputational timeand model size.

5. Boundary Conditions

After assigning the material to the radiator components, we need to apply the loads and boundary condition.Symmetrical boundary condition. The radiator model is symmetrical about a plane. The normal axis is constrained in the same symmetrical plane. The flat faces are constrained for all the DOF. During the life of a radiator, it is subjected to cyclic pressure load or pulsating load. This cyclic load induces stresses

in the structure. The maximum pressure amplitude is taken as the applied pressure load. Appropriate pressure is applied to the internal pressure surface.

Generally, for a heavy-duty commercial vehicle radiator, the applied pressure is 1.5times the operating pressure.

Conclusion

Finite Element Analysis helps in the visualization of the commercial vehicle radiator behaviour subjected to pressure cycle loads. Deformation and stressresults were studied for the radiator assembly. The maximumdeformation was observedat the core supports. Deformation plot shows that the model is expanding in the side direction. Core expansion of the header tube joint leads to a high stress gradient. Pressure cycle life at the header tube joint was found. This value meets the target life requirement of the commercial vehicle radiator.Based on the stress results, we can compute the pressure cycle life of a commercial vehicle radiator.

References

1.Robert D, Cook C, David S, Malkus M, Michael E, Plesha P, Robert J, Witt W- Concepts and application of Finite element analysis, John Wiley & Sons 1988.

2. P. Robin Roy, V. Hariram and M. Subramanian- Fatigue Life Prediction of a Commercial Vehicle Radiator under Internal Pressure Cycling Loading, 2016.

3. Dheeraj S, Sabarish R.- Analysis of Truck Chassis Frame Using FEM. Middle East Journal of Scientific Research2014.

4. Millard G.- Analysis of Durability Characteristics of Heavy-Duty Radiators, Society of Automotive Engineers.

5. 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.