Roy et al. 2017. *Int. J. Vehicle Structures & Systems*, 9(1), 7-10
ISSN: 0975-3060 (Print), 0975-3540 (Online)
doi: 10.4273/ijvss.9.1.02

International Journal of Vehicle Structures & Systems
Available online at www.maftree.org/eja

Accelerating the Product Development of a Commercial Vehicle Radiator using Finite Element Analysis

P. Robin Roy[a], V. Hariram[b] and M. Subramanian[c]

[a]*Dept. of Automobile Engg., Hindustan Institute of Tech. & Sci., Padur, Chennai, India*
Corresponding Author, Email: robin_cae@hotmail.com

[b]*Dept. of Mech. Engg., Hindustan Institute of Tech. & Sci., Padur, Chennai, India*
Email: hariram@hindustanuniv.ac.in

[c]*Dept. of Automobile Engg., BS Abdur Rahman University, Vandalur, Chennai, India*
Email: ramani@bsauniv.ac.in

ABSTRACT:

Emissions such as Nox and CO resulting from the combustion of the diesel engines in the commercial vehicles leads to environmental degradation and ozone layer depletion. Alarming environment trend forces the government institutions to develop and enforce strict emission laws for the next generation transportation vehicles. Stricter emission laws mean higher operating pressure, temperature, reduced weight, tight packaging space, engine downsizing etc. Engine cooling systems are the critical components in the managing the engine cooling requirement of the commercial vehicle. Generally engine cooling system includes radiator, charge air cooler, engine oil cooler etc. Product development of thermal management system using the traditional design process takes more time, resource and money. To solve the complex design problem, numerical technique such as finite element analysis is performed upfront in the product development of the radiator to evaluate the structure behaviour under mechanical loading. In this paper, internal static pressure analysis of a radiator is presented to showcase the benefits of using the finite element technique earlier in the product design phase. Pressure cycle life at a critical joint of the radiator is calculated using strain-life approach. Finite element analysis aids in visualization of the hot spots in the design, comparing different design options with less turnaround time. Experimental testing and prototypes can be reduced. Risk of a product being failed is greatly minimized by performing the numerical simulation.

KEYWORDS:

Engine cooling system; Radiator; Finite element analysis; Linear static analysis; Life estimation

CITATION:

P.R. Roy, V. Hariram and M. Subramanian. 2017. Accelerating the Product Development of a Commercial Vehicle Radiator using Finite Element Analysis, *Int. J. Vehicle Structures & Systems*, 9(1), 7-10. doi:10.4273/ijvss.9.1.02.

1. Introduction

A commercial truck engine generates a lot of heat during the combustion process. 33% of the heat energy is converted into power to drive the vehicle and its accessories; another 33% of the heat is forced out as exhaust gases into the surrounding environment through the exhaust system. The remaining 34% of the heat is rejected by the engine cooling system [1] to the surroundings and the engine temperature is kept under controlled level. Commercial vehicle engine cooling system consists of a radiator and charge air cooler. The radiator is the main heat exchanger, where the engine coolant heat is rejected to the passing air and re-circulated to the water jacket to absorb some more heat from the engine. Radiator design is complex due to higher operating pressure and temperatures. Radiator is subjected to various mechanical loads, thermal loads, road vibration loads, and environmental loads [2]. In this paper, linear static analysis of a radiator subjected to pressure load is performed to showcase the benefits of the finite element technique usage earlier in the product design phase.

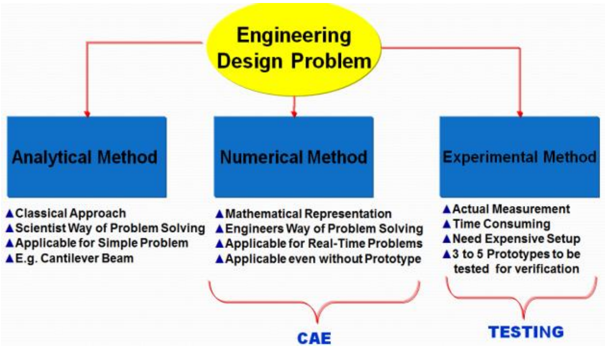

Fig. 1: Engineering problem solving methods

Any simple or complex engineering problem can be solved using three scientific solving methods [3] as shown in Fig. 1. The first method is called analytical method, also called as classical approach, very effective for solving simple problems. Numerical method is the most commonly used practical method to solve design problems ranging from simple to complex in nature. The solution of the numerical method has better accuracy and correlate with the physics of the real field application. Finite element method, boundary element method, finite volume method and finite difference method are three different types of numerical techniques. Third method of problem solving is the experimental method, which is the build, test and break approach. Very expensive, destructive method and time consuming due to recent developments in the computational resources and high end workstation, numerical techniques are gaining prominence during the product design of the automotive components.

2. Finite element analysis

The Finite element analysis (FEA) is defined as discretization of a domain 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. The whole domain or geometry is divided into smaller basic geometric entities such as trias, quad, penta, hexa etc. The stiffness of each of the entities is calculated and assembled to solve the assembly stiffness equation [5]. Displacement is calculated first and then stress/strains are derived from the resulting displacement. FEA is widely used in the automotive industry to analyze basic structural problems, strength/stiffness studies, crash simulation etc. Pre-processing, solution and post-processing are three basic steps involved in the finite element analysis [6]. Pre-processing includes the discretization or meshing of the geometry, material definition, loads and boundary condition. The solution involves the assembly and solving of the stiffness matrix. Post processing includes analysis and visualization of the solved results. Internal pressure analysis of the radiator is shown in Fig. 2.

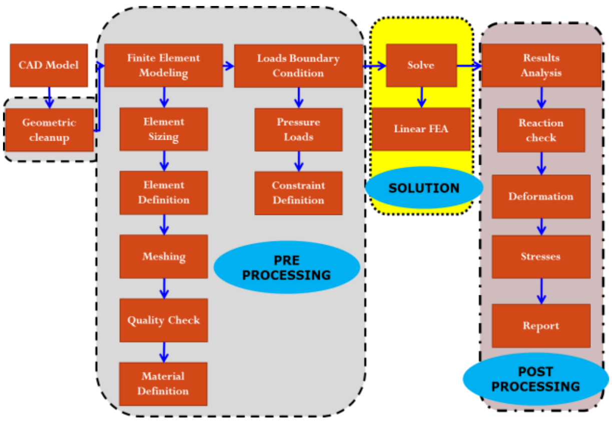

Fig. 2: Internal pressure analysis process flow

3. Linear static analysis

3.1. Geometric clean up

Computer Aided Design (CAD) geometry of the radiator model is modelled using CATIA software 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 radiator geometry into the finite element modeling software. In this study, the CAD model is exported as step format [7] from the CATIA software imported CAD model is thoroughly checked for any irregularities and imperfections [7] using geometric cleanup tools. Once the radiator geometry is free from defect, we can proceed to the next step of finite element meshing. Defect free geometry is an important prerequisite to have a better mesh pattern, less computational effort and greater accuracy of simulation results [8].

3.2. Discretization of domain

Commercial vehicle truck radiator consists of a tube, header, fin, gasket, tank and side piece [8]. To minimize the computational and modeling time, an 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 behaviour of the radiator. Also the model takes into account of the header tube joint as shown in Fig. 3, which is critical location studied during the internal pressure analysis [9]. Commercial vehicle radiator is manufactured through brazing process [10]. The brazing joints in the finite element analysis are considered by merging the nodes at the brazing location. The header tabs in the header, crimps with the plastic radiator tank onto the core system. The crimping tab is modelled to include the header tank connection stiffness. Hypermesh version 13 software [11] is used for finite element modeling. The radiator tank is bolted to the steel frame channels through rubber isolators. The rubber isolator is modelled and connects to the tank assembly. The rubber isolator dampens the vibration transmitted to the core and also aids in the radiator core expansion. 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 study [13]. Tank is meshed using second order tetrahedron elements (10 noded element) for better stress accuracy. Tube, header, rubber gasket, core side and fin are modelled using combined hexagonal (eight noded element) and penta element (six noded elements) respectively.

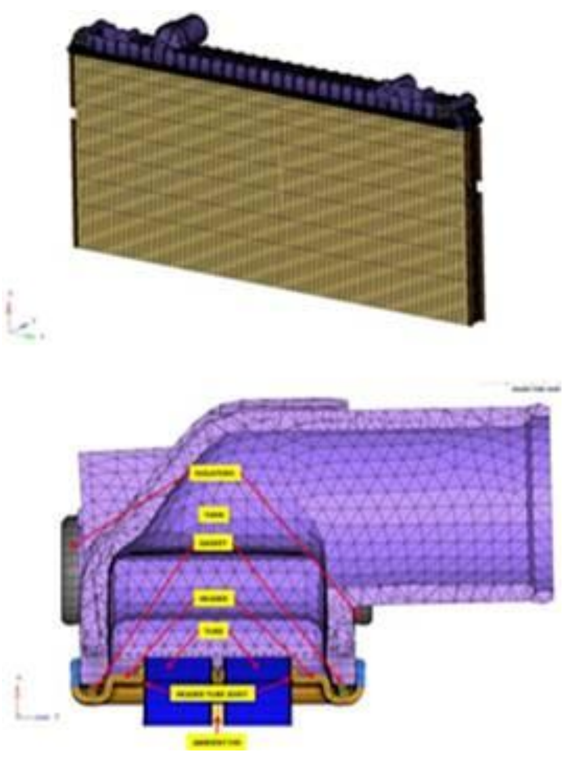

Fig. 3: Radiator FE model (top) & Header tube joint (bottom)

3.3. Material properties

The core system, including the tube, fins and core side are made from the AL 3003 material [8]. The radiator plastic tank is an injection moulded component from the fiberglass-reinforced Nylon PA66. Gasket material is EPDM rubber. [3]. Young's modulus and Poisson ratio are defined for the AL 3003 and plastic tank. Linear material model is assumed. Fin geometry is modelled as solid block to represent the equivalent orthotropic property to reduce the fin model size and save computational time. Young's modulus in three principal and shear planes is defined for the fin geometry. After the material property assignment, loads and boundary conditions are applied to the finite element model.

3.4. Loads and boundary conditions

The symmetrical boundary condition is applied in the radiator due to the symmetrical geometry. The radiator model is symmetrical about YZ plane; the normal X-axis is constrained (Fig. 4). Flat faces of the rubber isolators are also constrained in all the DOF. The radiator is subjected to the internal pressure load. Cyclic pressure load induces stresses in the radiator structure. The internal wetted surface of the tank, header and tubes are assigned with the applied pressure load (Fig. 4). The typical pressure magnitude of 30 psi is considered.

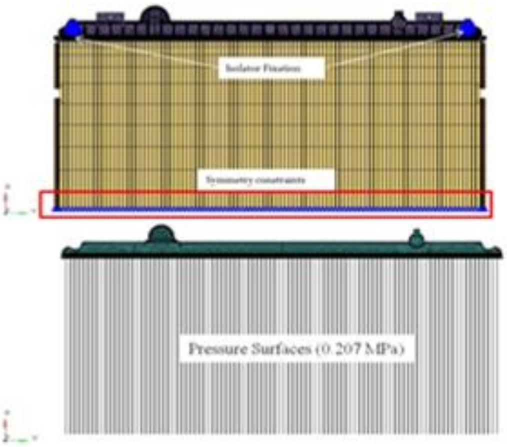

Fig. 4: Boundary conditions (top) and internal pressure application surface (bottom)

3.5. Linear static solution

Linear pressure analysis is performed using sparse matrix solver. Once the solution is completed successfully, the deformation plot of the radiator assembly is studied. During the pressure pulsation loading, the radiators expand in the core side to side direction. The deformation plot of the radiator assembly is shown in the shown in Fig. 5. The maximum deformation of 0.8mm occurs at the core ends. 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. Maximum von-mises stress on the commercial vehicle radiator is identified as 81 MPa at the header location. Maximum stress on the header tube joint corner is found as 75 MPa (Fig. 6). The header tube joint is typical area of interest for the internal pressure analysis [6]. The radiator tank stresses is measured as 16 MPa at the rib location.

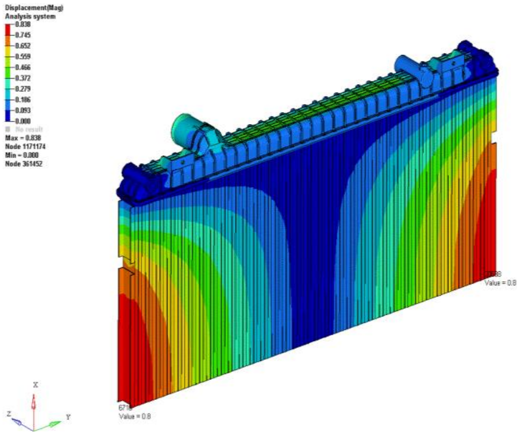

Fig. 5: Deformation plot (0.8 mm)

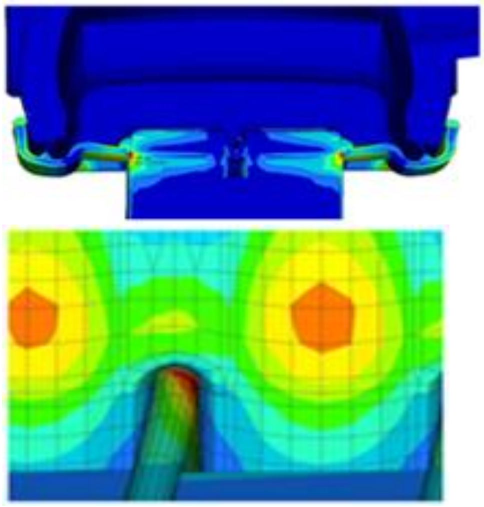

Fig. 6: Maximum von-mises stress plot (top) & Header tube joint stress (bottom) in MPa

3.6. Fatigue life estimation

The pressure load acting in the radiator is cyclic in nature. The max stress at the header tube joint is used to estimate the pressure cycle life at the joint. The fatigue life can be calculated using the material data, load and component geometry. In the study, the material properties of AL3003 are known [10]. The geometry can be defined by K_t, stress concentration factor which take in account of surface finish, surface infirmities etc. K_t of 1.7 is used for the analysis. The pressure cycle life of the header tube joint is calculated using strain life approach [14]. With the above three inputs the fatigue result is calculated using commercial software as shown in Figs. 7 and 8. The pressure cycle life of the header tube joint is found at 151,900 cycles to failure [15].

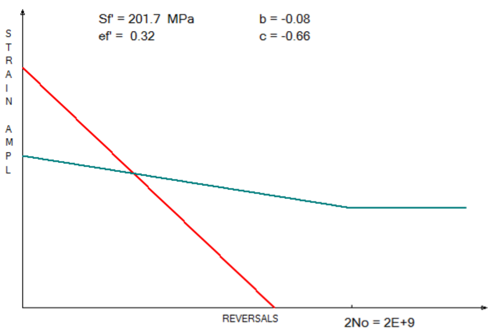

Fig. 7: Life estimation at header tube joint – Strain vs. Cycles

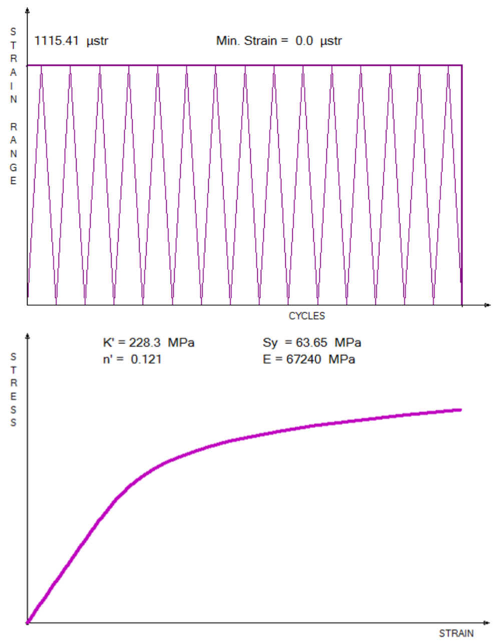

Fig. 8: Life estimation at header tube joint – Spectrum (top) and Stress vs. Strain (bottom)

4. Conclusions

Numerical Method such as finite element analysis leads to shorter product development cycle time. Product failure risk can be greatly reduced by performing up-front analysis. Reduced experimental testing, less prototypes are key benefits of the virtual simulation. Multiple design option cane studied and compared in a short span of time. Internal pressure analysis and fatigue life estimation is showcased to explain the influence of virtual simulation in accelerating the product design of the radiator. Linear static analysis identifies the hot spots in the design and evaluate the stresses in the critical locations such header-tube joint, crimping tabs, tube, header etc. The stiffness of the tank design due to the internal pressure loading can also be studied. The stresses from the linear pressure analysis can be used to estimate the fatigue life. Fatigue life estimation at the header-tube joint is calculated using Strain-life Approach. Geometry, loading and material data are the three components required to calculate the pressure cycle life. In the case study, the header-tube joint is measured as 75 MPa. Then pressure cycle life is evaluated and found to be 151,900 cycles to failure.

REFERENCES:

[1] S. Bennett. 2008. *Medium & Heavy Duty Truck Engines, Fuel & Computerized Management Systems*, 4th Edition Cengage Learning.

[2] P.K. Beatenbough. 1992. Radiator design and material recommendation based on failure mode analysis, *SAE Tech. Paper*, 920178. https://doi.org/10.4271/920178.

[3] R.D. Cook, D.S. Malkus, M.E. Plesha and R.J. Witt. 1992. *Concepts and Application of Finite Element Analysis*, 3rd Edition, John Wiley & Sons.

[4] E.P. da Silva, F.M. da Silva and R.R. Magalhaes. 2014 Application of finite elements method for structural analysis in a coffee harvester, *Engineering*, 6(3), 138-147. https://doi.org/10.4236/eng.2014.63017.

[5] S. Dheeraj and R. Sabarish. 2014. Analysis of truck chassis frame using FEM, *Middle East J Scientific Research*, 20(5), 656-661.

[6] C. Li, S. Fan and M. Shi. 2010. Preparation of CAD model for finite element analysis, *Proc. Int. Conf. on Computer, Mechatronics, Control and Electronic Engineering*, Changchun, Jilin, China.

[7] J. Eitel, G.T. Woerner and S. Horoho. 1999. Aluminium radiators for heavy duty trucks, *SAE Tech. Paper*, 1999-01-3721.

[8] G.S.T. Millard. 1977. Analysis of durability characteristics of heavy duty radiators, *SAE Tech. Paper*, 770024.

[9] W.S. Miller, L. Zhuang, J. Bottema, A.J. Wittebrood, P. De Smet, A. Haszler and A. Vieregge. 2000. Recent development in aluminium alloys for the automotive industry, *Materials Science and Engineering: A*, 280(1). https://doi.org/10.1016/S0921-5093(99)00653-X.

[10] *Altair HyperMesh*, User Guide, Version 13, Altair Engineering Inc.

[11] *Altair Optistruct*, User Guide, Version 13, Altair Engineering Inc.

[12] R.H. MacNeal and R.L. Harder. 1985. A Proposed standard set of problems to test finite element accuracy, *Finite Elements in Analysis and Design*, 1, 3-20. https://doi.org/10.1016/0168-874X(85)90003-4.

[13] A. Khosrovaneh, R. Pattu and W. Schnaidt. 2004. Discussion of fatigue analysis techniques in automotive applications, *SAE Technical Paper*, 2004-01-0626.

[14] H.S. Reemsnyder. 1979. Constant amplitude fatigue assessment, *SAE Technical Paper*, 820688.