WWW.IJRPR.COM

International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

Enhancement of Radiator Efficiency using CFD Analysis

Arjun Sastiya [a], Devendra Singh Sikarwar [a]

[a] *Department of Mechanical Engineering, Patel College of Science and Technology, Indore (MP), India*

A B S T R A C T

Experimental testing for fin-and-tube automotive radiator thermal properties is frequent but expensive and time-consuming. CFD analysis can estimate these radiators' thermal properties without experimental testing. CFD models cannot be utilized as design tools because to the intricate geometric structures of the fins, which demand a lot of computer power and time. Heavy-duty radiators complicate this issue. A basic model can predict radiator thermal behavior. This research analyses a simplified radiator tube using ANSYS fluent thermal CFD. Shell tria elements modelled tubes and tetrahedral elements modelled fluid media. Simulations used changing tube cross section. The tube outlet fluid cooling effect was studied and compared. The changing cross section cooled more than the simple tube in simulations.

Keywords: Radiators, Computational Fluid Dynamics, Ansys Fluent, Fin and Tube, Cooling Effect.

1. INTRODUCTION

Off-the-cuff engines use one-third of fuel energy to generate mechanical energy, one-third to exhaust heat, and one-third to cooling system heat load. Leading automakers are making more powerful and efficient engines. More stronger engines generate more energy and heat. Radiator cooling capacity improves with heat load. Engine designers estimate cooling capacity. Thus, cooling ability is a known input. Automotive companies also set cooling system size limits. The input parameters must be used to build a cooling system that meets engine cooling capability.

The engine cooling system removes waste heat. It controls engine surface temperature for efficiency. Radiators, pressure caps, cooling fans, thermostats, and water pumps are typical engine cooling systems. System's core is radiator. Radiators eliminate engine coolant heat. Outside air heats. Radiator assembly has core, output tank, and input tank. Core contains two passages, tubes, and fins. Coolant travels through air and fin tubes. Hot coolant heats fins through tubes. Heat is removed by outside air going via fins. Car radiators cool engines. It prevents banging, cylinder distortion, piston deformation, and other issues. Cooling system and engine performance improve if radiator works well. Air flow management is crucial to convective heat transfer via designing panels.

Real-world radiator design for engine cooling is now intriguing. Instead of mechanical design restrictions, thermal influences dominate design. Radiator thermal analysis is difficult. Calorimeter and air-to-boil tests can assess radiator heat capacity. These exams are costly and time-consuming. Radiator performance improvement may need several tests, making experimentation more time-consuming and expensive. Simulation methods like CFD analysis may be utilized for design, unlike experimental methods. Due to the radiator's complicated and repetitive geometrical properties, solving it requires a very large number of components (more specifically the complex geometry of the fins). Even modern computers cannot manage such a vast mesh. This work develops a new fin-tube radiator design approach using CFD. Radiator modelling underpins this strategy. Modeling simpler radiator tubes with fluid medium has greatly reduced computational mesh elements.

2. LITERATURE REVIEW

This literature study includes radiator efficiency improvement material. Ch. Indira Priyadarsini et al. developed a fluid cooling/heating heat exchanger (Radiator) design. It also addresses an upgraded fan-assisted air-cooled heat exchanger used in automobile, combustion engine (IC) engines, refrigeration systems, and power production. Mahmud, M. S. et al. evaluated the thermal performance of a simple automotive radiator under different coolant types and input velocities. Coolants include Al_2O_3, CuO, and TiO_2 nanofluids. The base fluids are water and 50% water/ethylene glycol (EG). Nanoparticle volume proportions were 1%, 2%, or 3%. Water-EG nanofluids have the lowest output temperature and quickest heat transfer.

Bengt Sunden tested small heat exchangers using car radiators. He employed thermal balances and correlations and finite control volume CFD. He discussed engineering methods including LMTD, NTU, correlations for heat transfer coefficients, frictional losses, area change, fluid acceleration, inlet, outlet losses, and centrifugal forces on fluid particles. Ismail et al. researched small heat exchanger thermohydraulic design. For the wavy fin design, Colburn and Fanning friction factors were computed for different Reynolds numbers. On three compact plate-fin heat exchanger setups, header flow maldistribution was computed. Header flow distribution simulations examined how nozzle orientation affects thermal performance. Results show that uniform header flow distribution improves heat exchanger thermal performance, depending on header nozzle orientation.

Varol et al. performed a finite difference research. Two-dimensional solid thin fin connected to porous right triangle enclosure. Darcy's equation models porous domain. Left wall is insulated, right wall is cooled, and lower wall is heated. Changing the fin placement produced streamlines and local Nusselt number fluctuation along the hot wall. The results show that the fin controls heat transfer and fluid flow passively. Yang et al. calculated forced convection in three-dimensional porous pin finned channels. Fin geometries and sizes were analysed. They employed Forcheimer-Brinkman model for flow characterisation and two-equation energy model for porous domain heat transfer. Literature-derived analytical formulas computed inertial, viscous, and heat transfer coefficients. Pressure drop and heat flux are examined for porous and solid fin designs. Optimized porous pin fin reduces pressure loss and boosts heat transfer.

3. CFD AND MODEL PREPARATION

CFD predicts fluid flow, heat and mass transport, chemical reactions, and other phenomena by numerically solving the governing mathematical equations. It simulates fluid movement, heat transport, and other physical processes on a computer. It solves fluid flow equations across an area with defined boundary conditions. The powerful approach has many industrial and non-industrial applications. Aerodynamics of aero planes and vehicles, hydrodynamics of ships, turbo machinery, electrical and electronic engineering, chemical process engineering, marine engineering, biomedical engineering, etc.

3.1 Modelling Setup and Governing Equations:

Continuity equation:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \quad \textbf{(1)}$$

X-Momentum equation:

$$\rho\left(u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right) \quad \textbf{(2)}$$

Y-Momentum equation:

$$\rho\left(u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right) \quad \textbf{(3)}$$

Z-Momentum equation:

$$\rho\left(u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial p}{\partial z}+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right) \quad \textbf{(4)}$$

Energy:

$$u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}+w\frac{\partial T}{\partial z}=\frac{1}{\alpha}\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}+\frac{\partial^2 T}{\partial z^2}\right) \quad \textbf{(5)}$$

Computational methods are needed to solve these complicated problems. Solving these linked equations requires a computing algorithm.

Numerical methods solve fluid flow issues in CFD programs. All CFD codes have three components.

1. Pre-processor
2. Solve
3. Post-processor.

Preprocessor

Pre-processing involves entering a flow problem into a CFD program using an operator-friendly interface and transforming it into a form the solver can utilize. The user defines the geometry of the region of interest, selects physical and chemical phenomena to mimic, defines fluid characteristics, and specifies boundary conditions for cells that contact the domain border.

CFD code numerical solver:

Fluent uses limited volume. The solver follows these steps:

• Simple functions for unknown flow variables.

Discretization by substituting approximations into controlling flow equations and mathematical manipulations

Algebraic equations solved.

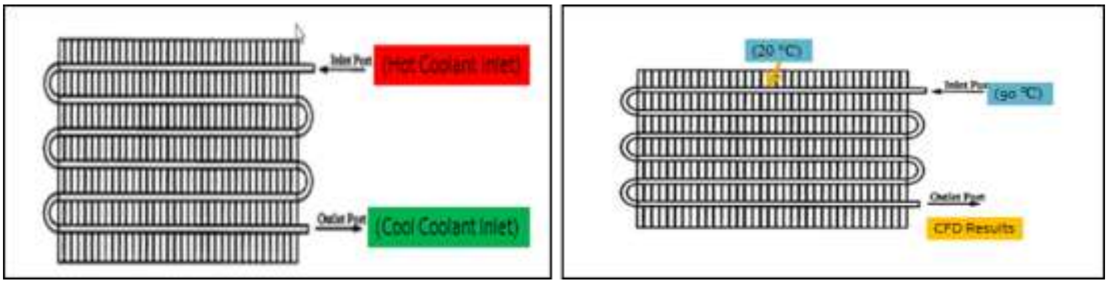
Post Processor:

Due to the prevalence of engineering workstations, many of which have excellent graphics capabilities, top CFD systems now feature extensive data visualization tools. Domain geometry and grid display Vector, line, shaded contour, and 2D and 3D surface plots

3.2 Model Preparation

This research proposes three-phase CFD modelling. Pre-processing entails. Radiator tube and fluid media modelling and parameterization. The solution step involves selecting a technique, tuning relaxation parameters, and solving. Post-processing processes the results. ANSYS 14.5 workstation and FLUENT® 14.5 are utilised for CFD analysis. Radiator water was modelled as a typical fluid domain.

Engine coolant is heated in the engine block, cooled in a radiator, then returned to the engine to cool it. Engine coolant is normally water-based but can be oil. Figure 1.(a) shows an axial fan and a water pump circulating engine coolant. 1.(b) displays radiator tube outer temperature.



(a) **(b)**

Fig. 1 - (a) Circulation of coolant inside the radiator; (b) Outer temperature of radiator tube.

3.2.1 Geometry

Geometry starts all difficulties. Geometry defines the issue to be solved. Its volumes, faces, edges, and curves advertise (points). Geometries can be constructed using the grid pre-processor or other tools (e.g., CAD, graphics). Figure 2 displays radiator tube geometry.

Fig. 2 - CAD model of radiator tubes

3.2.2 Meshing

It solves computational domain governing equations by discretizing components. Many cell/element and grid kinds exist. The present study's triangle and tetrahedron solvers determine the choice. Figure 3 displays the radiator tube and fluid domain finite element model.

Fig. 3 - Mesh of tubes and fluid.

Recommend conventional cooling improvements. Figure 4 shows a variable-cross section radiator tube model used to study its effect. Comparing both designs' output temperatures.

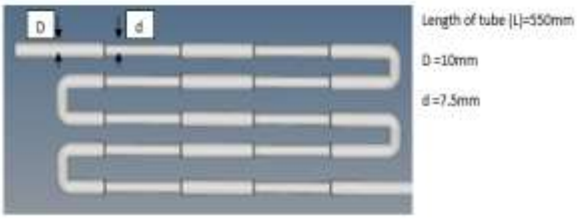
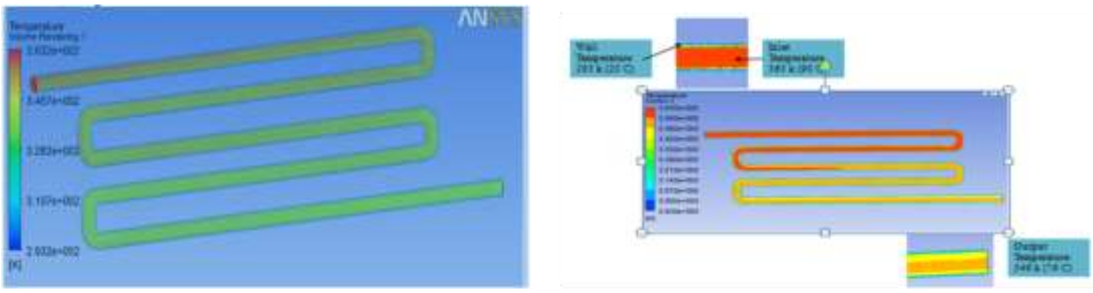


Fig. 4 - Modified tube

4. RESULTS

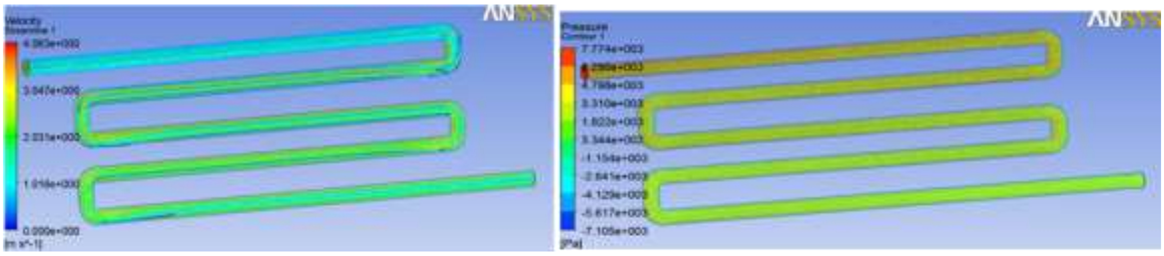
Figure 5(a) depicts the baseline temperature profile and radiator tube. As illustrated in figure, the fluid passes through the tube and cools till the output. Figure 5(b) illustrates the baseline case cross section and radiator tube temperature profile. As illustrated in figure, the fluid passes through the tube and cools till the output. Inlet calculus was 90 degrees. Due to cold water, the exterior wall exhibits 20 degrees calculus. Outlet temperature was 76 calculus.



(a) **(b)**

Fig. 5 – (a)Temperature variation of fluid inside the radiator tube; (b) Outlet and inlet temperature difference

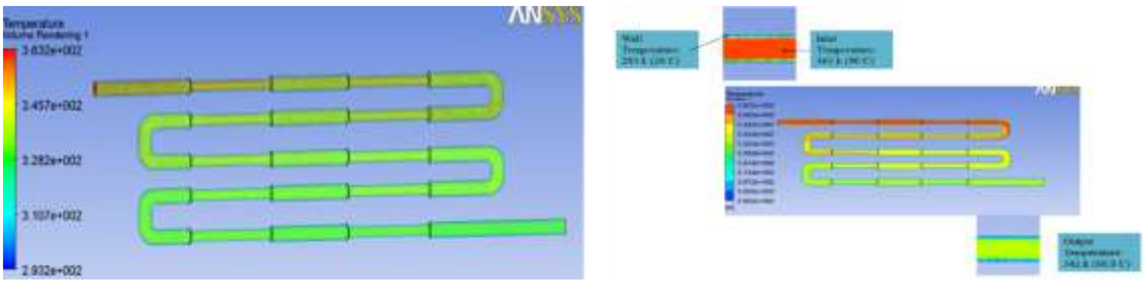
Figure 6 (a) depicts baseline velocity stream lines and the radiator tube. As illustrated in figure, the fluid enters the tube at 4m/s. The tube maintains fluid velocity. 6 (b) depicts the baseline pressure profile and radiator tube. Figure shows fluid travelling within the tube with 7.7 Pa beginning pressure. Fluid output pressure was 3.3 Pa



(a) **(b)**

Fig. 6 –(a) Fluid velocity streamlines; (b) Pressure streamlines

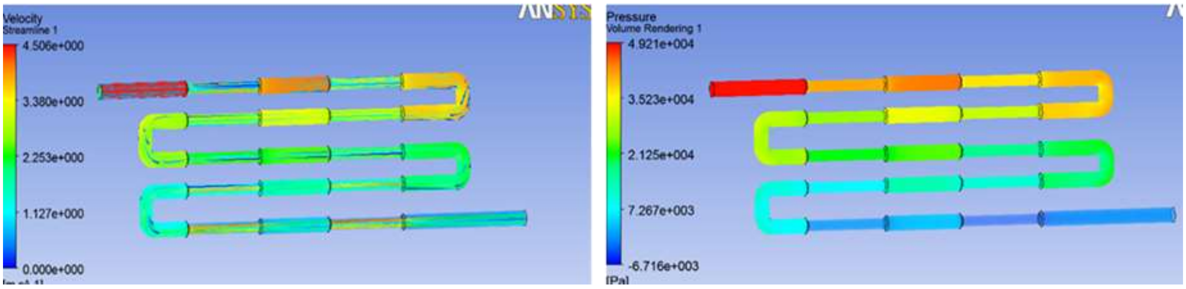
Figure 7 (a) illustrates the redesigned casing and radiator tube temperature profile. As illustrated in figure, the fluid passes through the tube and cools till the output. Figure 7 (b) illustrates the improved case cross section and radiator tube temperature profile. As illustrated in figure, the fluid passes through the tube and cools till the output. Inlet calculus was 90 degrees. Due to cold water, the exterior wall exhibits 20 degrees calculus. Outlet temperature was 68.8 calculus.



(a) **(b)**

Fig. 7 – (a) Temperature variation of fluid inside the radiator tube in modified design; (b) Outlet and inlet temperature difference in modified design

Figure 8 (a) depicts the updated case velocity stream lines and radiator tube. As illustrated in figure, the fluid enters the tube at 4m/s. The tube fluid velocity fluctuates. Figure 8 (b) illustrates the redesigned casing and radiator tube pressure profile. The fluid moves within the tube at 49 Pa, as indicated in figure. Fluid output pressure was 7.2 Pa.



(a) **(b)**

Fig. 8 –(a) Fluid velocity streamlines in modified design; (b) Pressure streamlines in modified design

5. CONCLUSIONS

Hyper mesh detailed CFD model meshing. ANSYS fluent analyzed CFD. With boundary conditions, the baseline case was simulated. Baseline temperature dropped 14 degrees Celsius. Modified design was applied to baseline example with varying cross section. Baseline and modified designs compared outcomes. The redesigned design dropped 21.2 degrees Celsius. Modified design boosts radiator efficiency by 8%.

Other parameters should be discussed here.

• The diameter decreases the hot fluid output temperature, cooling it. However, lowering the tube's diameter reduces its mass flow rate to very low, increasing cooling but decreasing cooling rate. However, lowering the diameter to compromise cooling rate is very difficult owing to manufacturing constraints. Since the circumferential area reduces with diameter, heat transmission diminishes and cooling decreases. Heat transfer depends on area and heat transfer coefficient, which varies on velocity, fluid regime, orientation, etc. As diameter lowers, velocity increases, increasing fluid turbulence. Reducing diameter increases cooling.

References

Priyadarsini, C. I., Reddy, T. R., & Devi, P. A. (2022). Design and Performance Analysis of Automotive Radiator using Computational Fluid Dynamics. International Journal of Mechanical Engineering, 7.

Mahmud, M. S., & Rijvi, F. R. (2022). Enhancing the Thermal Performance of Radiators using Nanofluids-A CFD Approach. Journal of Engineering Advancements, 3(02), 57-63.

Gautam, G., Gupta, A., & Kumar, R. (2019). CFD analysis of heat transfer performance of a car radiator using nanofluid. A Journal of Composition Theory, 12(7), 537-545.

Thomas, M. S., Karthikeyan, M. V., & Nallakumarasamy, G.(2018) Performance Calculation of Vehicle Radiator Group using CFD. International Journal for Research in Applied Science & Engineering Technology (IJRASET), 6, 908-921.

Oliet C., Castro J., Perez-Segarra C.D., “Parametric Studies on Automotive Radiators”, Applied Thermal Engineering, 2009.

Trivedi P.K., Vasava N.B., “Effect of variation in Pitch of tube on Heat Transfer Rate in Automobile Radiator by CFD analysis”, International Journal of Engineering and Advanced Technology, Volume 1 (6) 2012.

Salvio chacko, Dr. Biswadeep Shome, Vinod Kumar, A.K. Agarwal, D.R. Katkar, “Numerical Simulation for Improving Radiator Efficiency by

Air Flow Optimization”, Engineering Research Centre, Tata Motors Limited, Pune, 2013.

Yadav J.P., Singh B.R., “Study on Performance Evaluation of Automotive Radiator” S-JPSET, 2011.

Omprakash P., Rajesh J., Sanjay T., “VSRD International Journal of Mechanical, Civil, Automobile and Production Engineering”, Vol.III Issue X October 2013.

Thombare D.G., Khot A.R., “An Overview of Radiator Performance Evaluation and Testing”, IOSR Journal of Mechanical and Civil Engineering, 07-14, India, 2015.

Oduro S.D., “Assessing the Effect of Blockage of Dirt on Engine Radiator in the Engine Cooling System”, Design and Technology Department, Ghana, Vol. 2, number 3, July 2012.

Chavan D.K., Tasgaonkar G.S., “Thermal optimization of fan assisted heat exchanger (radiator) by design improvements”, International Journal of Modern Engineering Research, Vol. 3, June 2013.

Wen M.Y. ,Ho C.Y., “Heat transfer enhancement in fin and tube heat exchanger with improved fin design”, Applied Thermal Engineering, vol. 29, 2009.

Yan W.M., Sheen P.J., “Heat transfer and friction characteristics of fin and tube heat exchangers”, International Journal of Heat and mass transfer, vol. 43, 2000.

Ismail L.S., Ranganatakului Ramesh C.K., “Numerical Study of Flow Patterns of Compact Plate-fin Heat Exchangers and Generation Design Data for Offset and Wavy Fins”, International Journal of Heat and Mass Transfer, vol. 52 2009.

Varol Y., Öztop H., Varol A., “Effects of Thin Fin on Natural Convection in Porous Triangular Enclosures”, International Journal of Thermal Sciences, 2007.

J.Yang, M. Zeng, Q. Wang, “Forced Convection Heat Transfer by Porous Pin Fins in Rectangular Channels”, Quiwang Wang, vol. 132, May 2010.

Birch T., “Automotive Heating and Air Conditioning”, 4th Ed., Pearson Prentice Hall, 2006.

Do K.H, Min J.Y., Kim S.J., “Thermal Optimization of an Internally Finned Tube Using Analytical Solutions Based on a Porous Medium Approach”, ASME, Journal of Heat Transfer, vol. 129, 2007.

Kim S.J., Kim D., “Forced Convection in Microstructures for Electronic Equipment Cooling”, ASME, Journal of Heat Transfer, vol. 121, 1999.

Kim D., Kim S.J., “Compact Modelling of Fluid flow and Heat Transfer in Straight Fin Heat Sinks”, ASME, Journal of Electronic Packaging, vol. 126, 2004.

Jeng T. and Tzeng S., “A semi-empirical model for estimating permeability and inertial coefficient of pin-fin heat sinks”, International Journal of Heat and Mass transfer, vol. 48, 2005.

Kulasekharan N., Purushotham H.R., Junjanna G.C., “Performance improvement of a louver-finned automobile radiator using conjugate thermal CFD analysis”, International Journal of Engineering Research & Technology, vol. 1 issue 8, 2012.

DeJong N.C., Zhang L.W., Jacobi A.M., Balachandar S., Tafti D.K., “A complementary Experimental and Numerical Study of the Flow and Heat Transfer in Offset Strip-Fin Heat Exchangers”, ASME, Journal of Heat Transfer, vol. 120, 1998.

You H.,Chang C.H., “Determination of flow properties in non-darcian flow”, ASME, Journal of Heat Transfer, vol. 119, 1997.

Zukauskas A., Ulinskas A., “Efficiency parameters for heat transfer in tube banks”, Heat Transfer Engineering, vol. 6, 19-25, 1985.

Jain S., Deshpande Y., “CFD Modelling of a Radiator Axial Fan for Air Flow Distribution”, World Academy of Science, Engineering and Technology, vol. 71, 2012.

Zhang Z., Li Y., “CFD simulation on inlet configuration of plate-fin heat exchangers”, Cryogenics, vol. 43, 2003.

Wasewar K.L., Hargunami S., Atluri P., Kumar N. , “CFD Simulation of Flow Distribution in the Header of Plate-Fin Heat Exchangers”, Chem. Eng. Technol., vol.30, 2007.

Baliga B.R., Azrak R.R., “Laminar Fully Developed Flow and Heat Transfer in Triangular Plate-Fin Ducts”, ASME, Journal of Heat Transfer, vol. 108, 1986.

Zangh L., “Laminar flow and heat transfer in plate-fin triangular ducts in thermally developing entry region”, International Journal of Heat and Mass Transfer, vol. 50, 2007