MAT JOURNALS
Volume 4 Issue 1

Simulation Software Based Analysis of Automotive Radiators: A Review

Abhishek Gakare[1,*], Ashutosh Sharma[2], Mr. Gaurav Saxena[3]
[1, 2] M.Tech Students, [3]Assistant Professor
Department of Automobile Engineering, Rustamji Institute of Technology, BSF Academy Tekanpur, Gwalior, Madhya Pradesh, India
****Email:** Abhishekgakare@gmail.com*
***DOI:** http://doi.org/10.5281/zenodo.2563431*

Abstract

The thermal execution of a vehicle radiator assumes an essential role in the execution of a vehicles-cooling system and all other connected system. For various years, this element has experienced little consideration with almost no changing in its built-up cost, geometry and operation. Compared with the longstanding tubular heat exchangers at present frame the foundation of the present procedure industry with their innovative performance reading levels tubular heat exchangers they are commonly utilized. Present review focuses on the different research with respect to CFD Analysis to increase vehicle radiator efficiency. present paper is an examination about the impact of different parameters like size of radiator core, way flow of working fluid, frontal zone of radiator, space in fins, pitch area of tube, tube and fin shape, mass flow rate of coolant, fins material, velocity of fluid, pitch area of tube, inlet temperature of air and different parameters to develop vehicle radiator determination and efficiency of advanced geometry from above parametric investigation. The Computational fluid elements (CFD) simulations were comparing the pressure drop and heat transfer of heat exchanger with various parameters for ideal performance. CFD results have high correspondence level by authentic experimental outcomes. Several results of review recommend that CFD have been shown very effective in decreasing production cost and testing time.

***Keywords:** Radiators, CFD, Cooling performance, Heat transfer, ANSYS CFX*

INTRODUCTION

Vehicle Radiator are pleasant very power-packed with expanding capacity to weight or volume ratio. Expanded interest on power packed radiators, which can dissipate upper limit quantity of heat for some given space. The Panel radiators are the most generally utilized central-heating emitters to warm most homes and workplaces in Europe. There is popularity for panel radiators because of their reduced structure and less place necessities. 80% of the heat yield from radiators is natural convection, 20% of the heat yield from radiators is radiation. Although radiators are known as radiator, most of their output is by natural convection [1]. Using computational fluid dynamics (CFD) to model the flow of fluid and heat transfer performance characteristics for one such optimum design as a possible replacement for the conventional automobile radiators [2]. Fins are used to increase heat transfer area on the airside, since the air has the largest influence on the overall heat transfer rate, mass flow rate of air, pitch of tube and coolants are analyze successfully using numerical simulation built in commercial software ansys12.1. CFD analysis gives accurate and exact result. Role of CFD is very vital nowadays as a design tool. For CFD simulation various commercial software are available in market. Modeling is done by CAD then whole discretization model is resolved into small cells by meshing. Apply main equation to discrete element and calculated them through CFD solver. Numerical results are found concerning temperature distribution, pressure distribution air flow distribution and so on. Then result is elevated and that result is authenticated against base data. If this model is as per requirement, its prototype will be manufactured and tested then produce for real world applications [3].

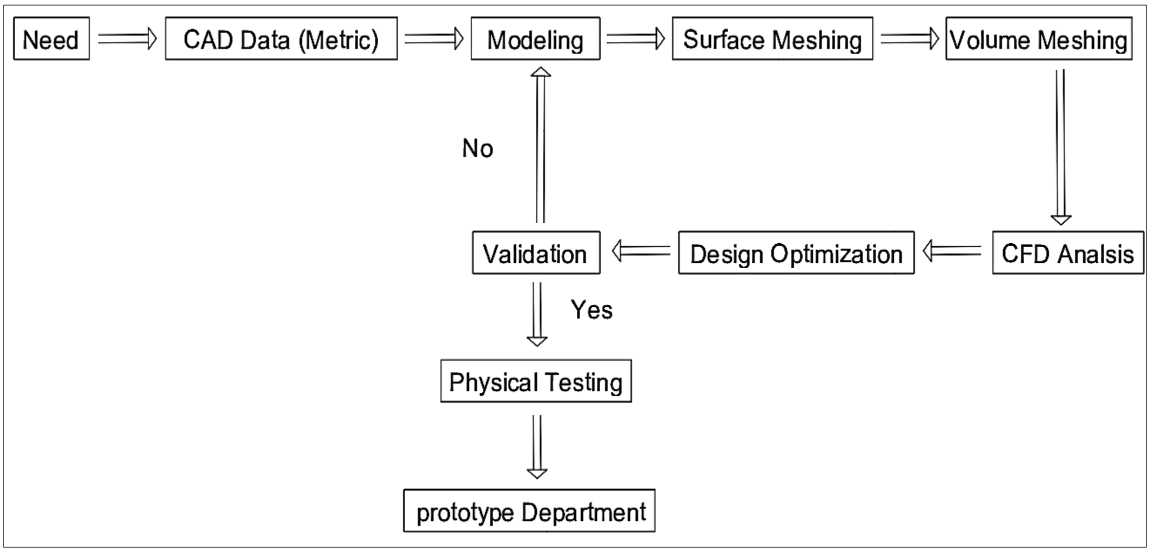

Figure 1: *Process Flow Diagram of CFD Analysis.*

REVIEW ON CFD ANALYSIS

Yeduru Jyothi et al., [4] developed a model of fin with louvered section. The two kinds of fins are taken with the end goal that underlying is a rectangular fine and the last is a trapezoidal shape. The tube-shaped of 5 sections are produced for the fins. The models are shaped by CATIA V5 R20 and investigated through computational examination in ANSYS 15.0 with easy interface. Analysis of radiator without louver fins and with louver fins is carried out. The original radiator has no louver fins. The computational analysis tool ANSYS is used to perform a CFD analysis on radiator. The initial parameters are inlet air velocity and air Inlet temperature. The analysis results as velocity, pressure and heat transfer rate is more for the radiator with louver fins that of the original model. Heat transfer analysis is done to analyze the heat transfer rate to determine the thermal flux. The material taken is Aluminum alloy 6061 for thermal analysis. The thermal analysis results, thermal flux is more for the radiator with louver fins that of the original model, so it was concluded that heat transfer rate is more.

R. K. Jaya Kumar et al., [5] experimentally investigated the performance of copper radiator used in passenger cars. Behavior of a copper radiator at different air velocities is determined both experimentally as well as by simulation. The obtainable copper radiator is shaped by SOLIDWORKS software and authentication of execution done by fluid flow software programming. The investigational and simulation result indicates just 8% of deviation. Therefore, it met engineering necessities of analysis and authentication with allowable deviation. So, this technique may be taken as an actual tactic for the forecast of heat transfer show in a radiator. This will frame as the base to examine the act of copper braze by changing the material properties and its parameter. The heat transfer coefficient and surface pressure drop increase with increase in air velocity is show in Figure 2. Deviation in results is mainly due to friction loss, quality of mesh and its accuracy.

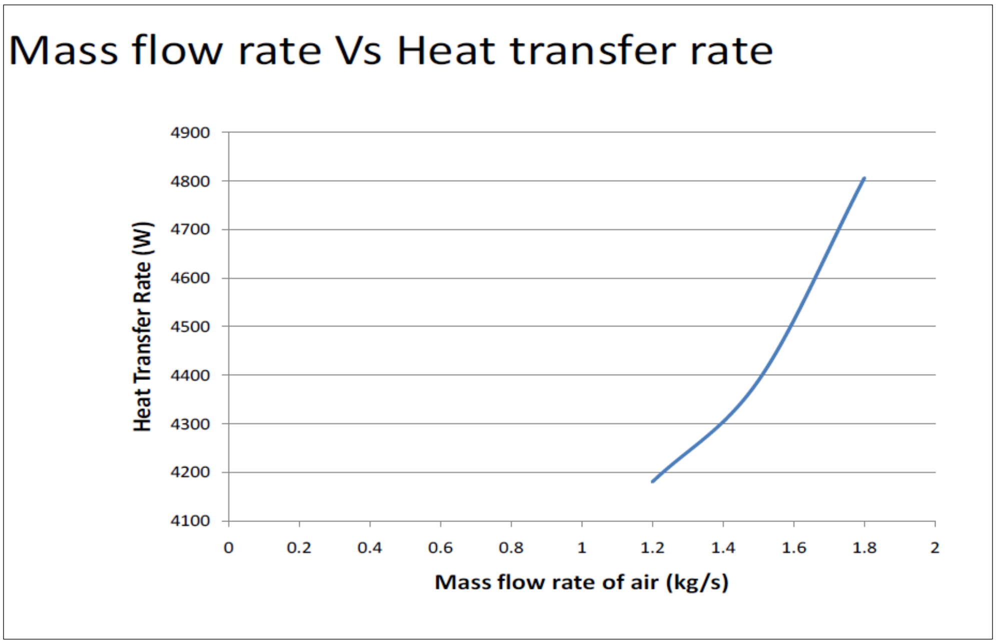

Figure 2: *Effect of Mass Flow Rate Vs Heat Transfer Rate [5].*

Hardik kumar B. et al., [6] performed an investigation by the solid modeling of the radiator in solid works and then this solid model is transferred to the ANSYS Workbench mesh module for meshing. Subsequent to finishing meshing, this meshed model is exchanged to ANSYS CFX for CFD Analysis. After CFD investigation is compiled with ANSYS CFX. The efficiency of the radiator can be upgraded by exchanging confident operating and geometrical parameter similar Composition rat, fluid Composition (Additives), Tube Diameter and so on. Figure 3 show the effect of tube side fluid on radiator performance, after completing all the above parametric study, best-configured radiator can be suggested for optimum performance.

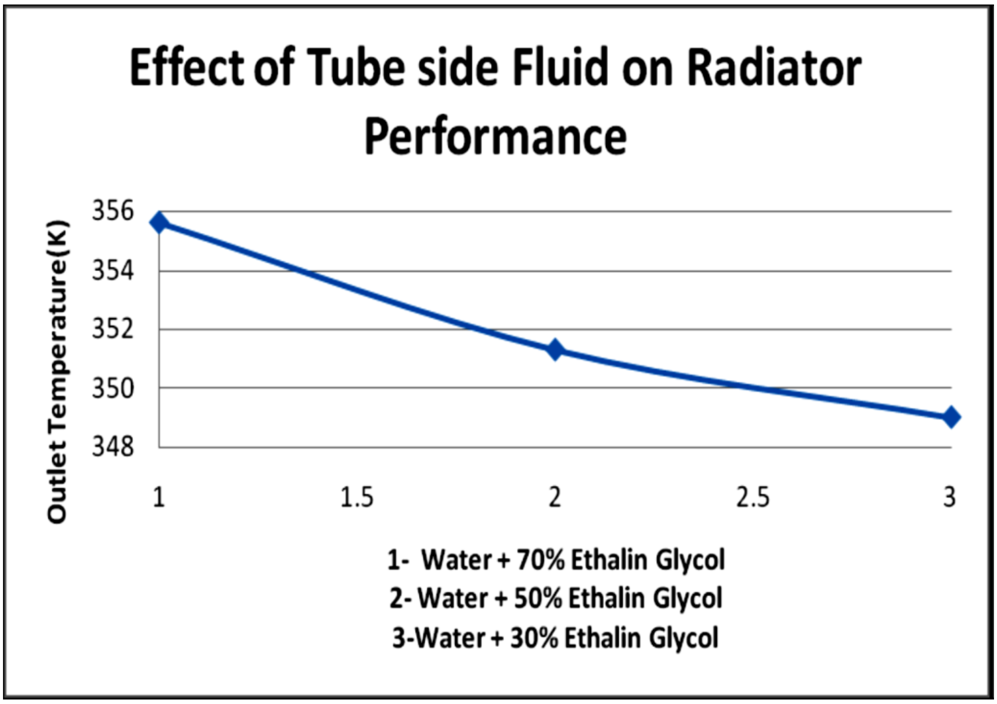

Figure 3: *Effect of Tube Side Fluid [6].*

P. K. Trivedi et al., [7] created the model of heat exchanger in solid works and then this solid model is transferred to ANSYS Workbench mesh module for meshing. In the wake of finishing meshing, this meshed model is exchanged to ANSYS CFX for CFD Analysis. Once CFD Analysis is done with ANSYS CFX, all flow parameters like as temperature contour, heat transfer rate, etcetera were recognized. Afterward getting all the flow parameter they inspect how the heat transfer rate of radiator can be improved. For this reason one geometrical parameter for example pitch of tube was changed. Because of this parametric analysis, the impact of pitch of tube for best designed radiator for ideal performance is recommended.

S. N. Sridhara et al., [8] experimentally studied a tube fin arrangement of an existing radiator for evaluating the fluid flow and heat transfer characteristics. The total pressure, mass flow rate and temperature supply of the coolant and air in and about the single tube-fin order through 32 fins were assessed. That the fluid flow simulation is operated directed utilizing commercial software FLUENT 6.1.

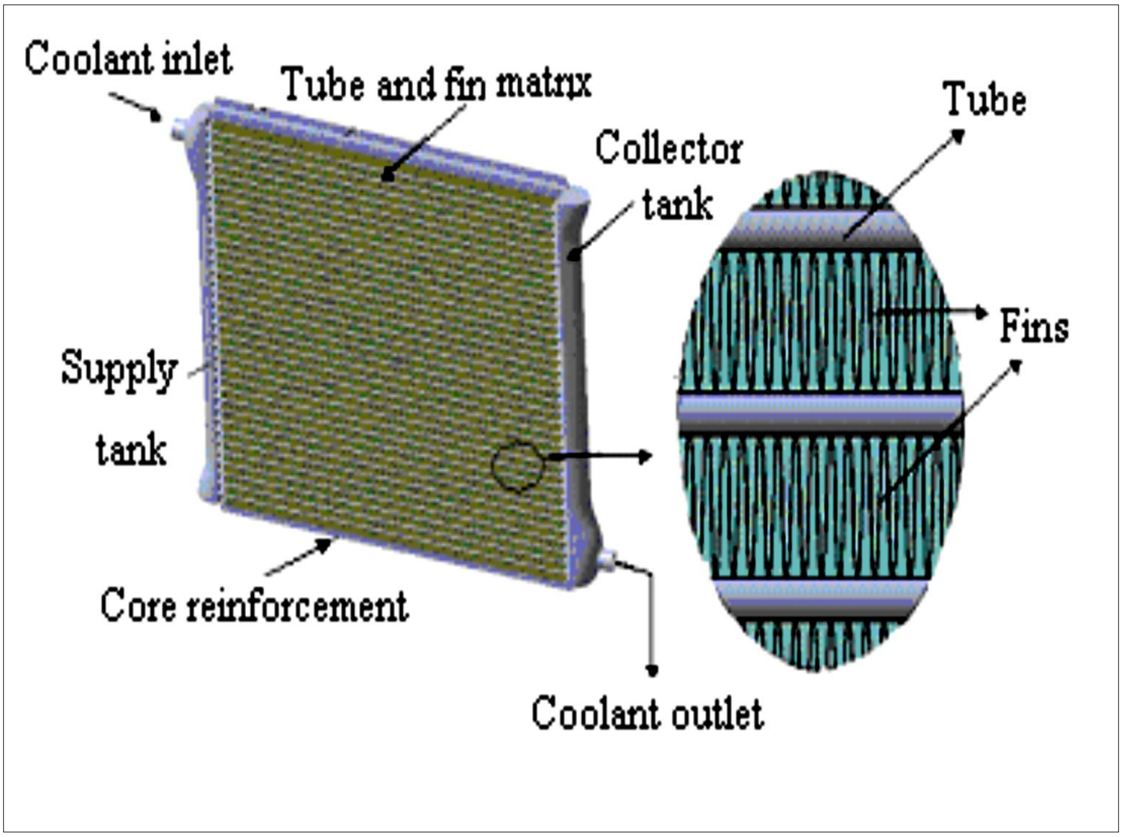

Figure 4: *Assembled CATIA Model of the Radiator [8].*

The pressure and temperature distribution along the tube length and tube width are presented and analyzed. The results obtained serve as good database for the further investigations for researchers. Figure 4 presents the experimentation work carried out, involving reverse engineering of a commercial automotive radiator for the required fluid domain, discretizing the fluid domain, simulation of the fluid flow, heat transfer at steady state and post processing to visualize the results and drawing bring suitable conclusions. The cumulative summary of simulation based research carried out on radiator is presented in Table 1.

Table 1: *Summary of CFD (Computational Fluid Dynamics) Analysis of Automotive Radiator.*

Author	SoftwareUsed	Study	Outcomes
Yeduru Jyothi et al. [4]	Ansys 15.0 Catia V5 R20	CFD analysis of an automobile radiator with and without louvered fins using nano fluids	The velocity, pressure and heat transfer rate is more for the radiator with louver fins as compared to conventional one.
R. K. Jaya Kumar et al. [5]	Solidworks	Experimental study and CFD analysis of copper radiator for passenger cars	Heat lost by the coolant is not equal to heat gained by the air.
Hardik Kumar B et al. [6]	Ansys CFX	Performance analysis of an automobile radiator using CFD	Increasing the water in the mixture causes decrease in tube side outlet temperature of the radiator. Decreasing the tube diameter, increases the tube side outlet temperature of the radiator.
P. K. Trivedi et al. [7]	Ansys CFX	Effect of variation in pitch of tube on heat transfer rate in automobile radiator by CED analysis	The pitch of tube is either decreased or increased, the heat transfer rate decreases, optimum efficiency is obtained at the pitch of 12 mm.
S. N. Sridhara et al. [8]	Fluent 6.1	CFD analysis of fluid flow and heat transfer in a single tube-fin arrangement of an automotive radiator	Air absorbs the heat due to forced convection gains with an increase in temperature by 9.5 K.

CONCLUSION

The present review on recent article shows that CFD analysis has reduced the cost, time of design and development of radiator as compared to conventional methods. It also reduces the need of prototype testing during design process while we do iterations to get optimized design. Prototype of the design for physical testing is only required. CFD analysis provides direct comparing of the heat transfer & pressure drop of heat exchanger with different parameters for optimum performance.

REFERENCES

1. Beck S. M. B., Grinsted S. C. & Blakey S. G., et al., A novel design for panel radiators. *Applied Thermal Engineering*, 2004; 24: pp. 1291-1300.
2. Patel J.R. & Mavani A.M. Review Paper on CFD Analysis of Automobile Radiator to Improve its Thermal Efficiency; 2: pp. 268-271.
3. Patel H.B., Dinesan M.D., Patel H.B. & Dinesan M.D. Optimization and performance analysis of an automobile radiator using CFD - a review. *Int J Innov Res Sci Technol*. 2015; 1(7): pp. 123-126.
4. Jyothi Y., Rajesh M. & Naidu M.C. CFD Analysis of an Automobile Radiator With and Without Louvered Fins Using Nano Fluids. *Development*. 2017; 4(8).
5. Kumar R.K. & Amirtha Gadeswaran K.S. Experimental study and CFD Analysis of Copper radiator for Passenger Cars. *International Journal of Engineering Research*. 2016; 5(9): pp. 778-782.
6. Patel H.B., Dinesan D., Patel H.B. & Dinesan D., Performance Analysis of an Automobile Radiator using CFG. *International Journal for Innovative Research in Science & Technology* 2016; 1: pp. 318-322.
7. 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. 2012; 1(6): pp. 180-3.
8. Sridhara S.N., Shankapal S.R. & Babu V.U. CFD Analysis of Fluid Flow and Heat Transfer in a Single Tube-Fin Arrangement of an Automotive Radiator. *In International Conference on Mechanical Engineering*. December 2005.

Cite this article as:
Abhishek Gakare, Ashutosh Sharma, & Mr. Gaurav Saxena. (2019). Simulation Software Based Analysis of Automotive Radiators: A Review. Journal of Automation and Automobile Engineering, 4(1), 13–18. http://doi.org/10.5281/zenodo.2563431