iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 9 Issue: VII Month of publication: July 2021

DOI: https://doi.org/10.22214/ijraset.2021.37186

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue VII July 2021- Available at www.ijraset.com

Review on Thermal Analysis of Automobile Radiator

Pavan Kushwah[1], Himanshu Sharma[2], Gaurav Saxena[3]
[1, 2, 3]Department of Automobile Engineering, Rustumji Institute of Technology, Tekanpur, Gwalior (M.P), INDIA

Abstract: Radiators are used to transfer thermal energy from one medium to another for the purpose of cooling. Low efficiency heat exchangers used in automotive as radiator may cause to serious dangers for the engine. Hence, thermal scientists and engineers always pursuit modern methods to enhance the heat removal of the engine. It seems nanofluids implementation in automotive cooling system promises to achieve high efficiency radiators. This paper reviews almost all performed studies in this area that are available in the literature. Author collects details about nanoparticles materials and size, base fluid, volume, concentration, flow regime and Reynolds number used in studies. Usually, maximum heat transfer enhancement and maximum need of pumping power that occurs at the highest volumetric concentration of nanoparticles, simultaneously. On the other hand, using nanofluids, due to the enhanced heat carrying capacity of the nanofluids; the pumping power required will also be reduced.
Keywords: Nano coolant, volumetric concentration, automotive radiator, circular tubes, heat transfer, connective heat transfer coefficient.

I. INTRODUCTION

Radiators is a heat exchangers device which transfer heat energy from one medium to the another medium unique for the goal of the cooling and the heating. The bulk of radiators are built to feature in motors, building like apartments, and electronics [1]. It is mostly a supply of warmth to its surroundings, despite the fact that this is probably for both the intention of heating the around, or for reduce the temperature of the fluid or coolant provided there to, as for engine cooling as shown in Fig. 1. Despite in the name, most of the radiators transfer through convection as opposed to thermal radiation [2]. The Roman hypocaust is the early example of the form of radiator for constructing space heating.

Franz San Galle, a Prussian-born Russian businessman dwelling in St. Petersburg, is suited with invention the heating radiator in around year 1855 had received the radiator patent rights in 1857 but the American Joseph Nason have developed a primitive [3]. American JosephNason advanced a primitive radiator in and received kind of U.S. Patents for respectable water and steam warmness [4]. A cooling system in vehicles is hired to put off excess warmth produced within the combustion engines. In most motors and the cooling gadget includes the subsequent of these additives, the radiator is the maximum distinguished part of the cooling system, because it transfers the heat in the complete gadget.

Their components are;

1) Radiator with fans
2) Pump, strain cap and thermostat.

The radiators applied in vehicles are mainly rectangular shaped, however air blown/sucked via the fan is circular in area which develops low speed zones in corners [5]. So, it is proposed to increase a round radiator which removes the corners [6]. The proposed circular radiator is compact in size in comparison with the traditional ones. Researchers also proposed higher and lower tank of radiator designed within the fan area [7]. The higher and lower tanks of radiator have fins designed with them which enhance the warm temperature switch nearly like air cooled engines [8]. Tubes and fins of radiator middle are designed in the draft place of the fan. Hence, warmness transfer will occur efficiently [9]. The proposed design of circular radiator is expected to be less complicated and compact one. Fig. 1 shows the different parts of a radiator.

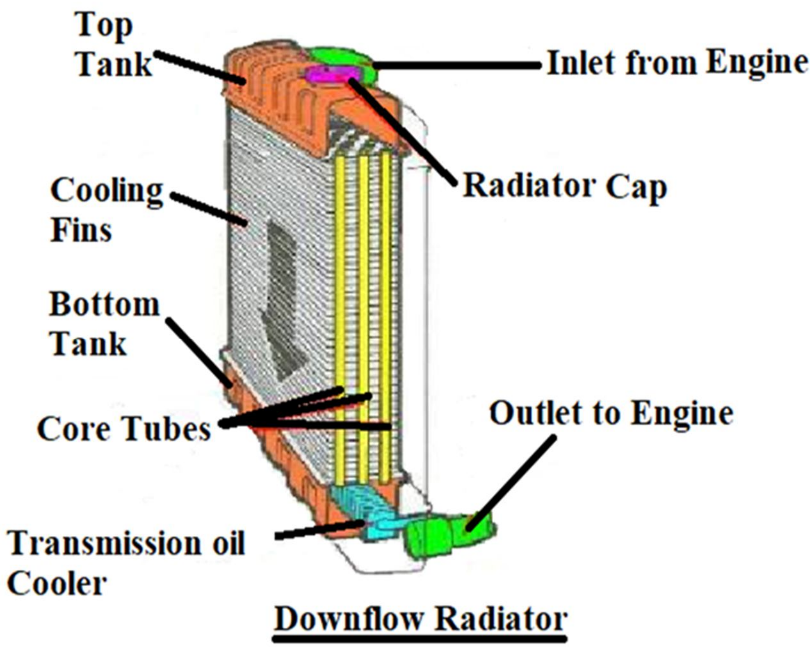


Fig. 1 Parts of Radiator

II. MODES OF HEAT TRANSFER

The radiator may also be defined as a device, having many numbers of fins arranged in a same manner, which have the contact with the tube carrying liquid pumped into the radiator tube. The coolant in the tube is in touch with in the fins attached and carries off heat from the metal tube surface. If air flow is obstructed by dust or damage, other obstruction to the fins that part of the radiator is ineffective at heat transfer no heat transfer take place at the part. In regular life, the word ‘‘radiator” means to the any of the devices by which the liquid is circulates through exposed or internal tube often with in fins or other means of projected metal surface around the tube.

Hence,heat transfer from radiator mainly occurs in the form of convection as well as conduction, these heat transfer from a radiator occurs by all the standard forms of mechanisms: thermal radiation and the convection into the flowing air or liquid, and conduction into the fluid. A radiator can transfer heat by means of phase transfer for example: drying the set of socks. Fig.2 shows the Line diagram of heat transfer through cooling system.

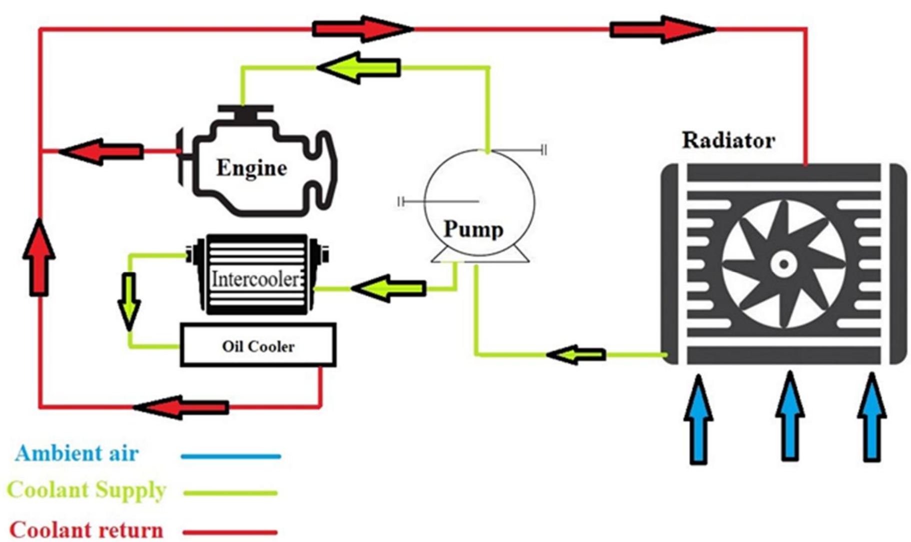


Fig. 2 Line diagram of heat transfer through cooling system

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue VII July 2021- Available at www.ijraset.com

III. METHODS OF ANALYSIS ON RADIATOR

In present article the methods of analysis are broadly classified as simulation based computational methods and experimental method. The computational methods used for simulation analysis is discussed briefly whereas as per the scope of article a broad review on experimental based analysis is presented.

A. CFD Analysis on radiator

Role of CFD is very vital nowadays as a design tool. For CFD simulation various commercial software are available in market. Modelling is done by CAD then whole discretization model is resolved into small cells by discretization. Apply governing equation to discrete element and solve them by CFD solver. Numerical solutions are obtained regarding pressure distribution, temperature distribution, air flow distribution etc. then result is optimized and that result is validated against base data. If this model is as per our requirement its prototype will be cast and test then produce in real world application. The steps of CFD analysis in the form of flow chart are shown in the Fig. 3.

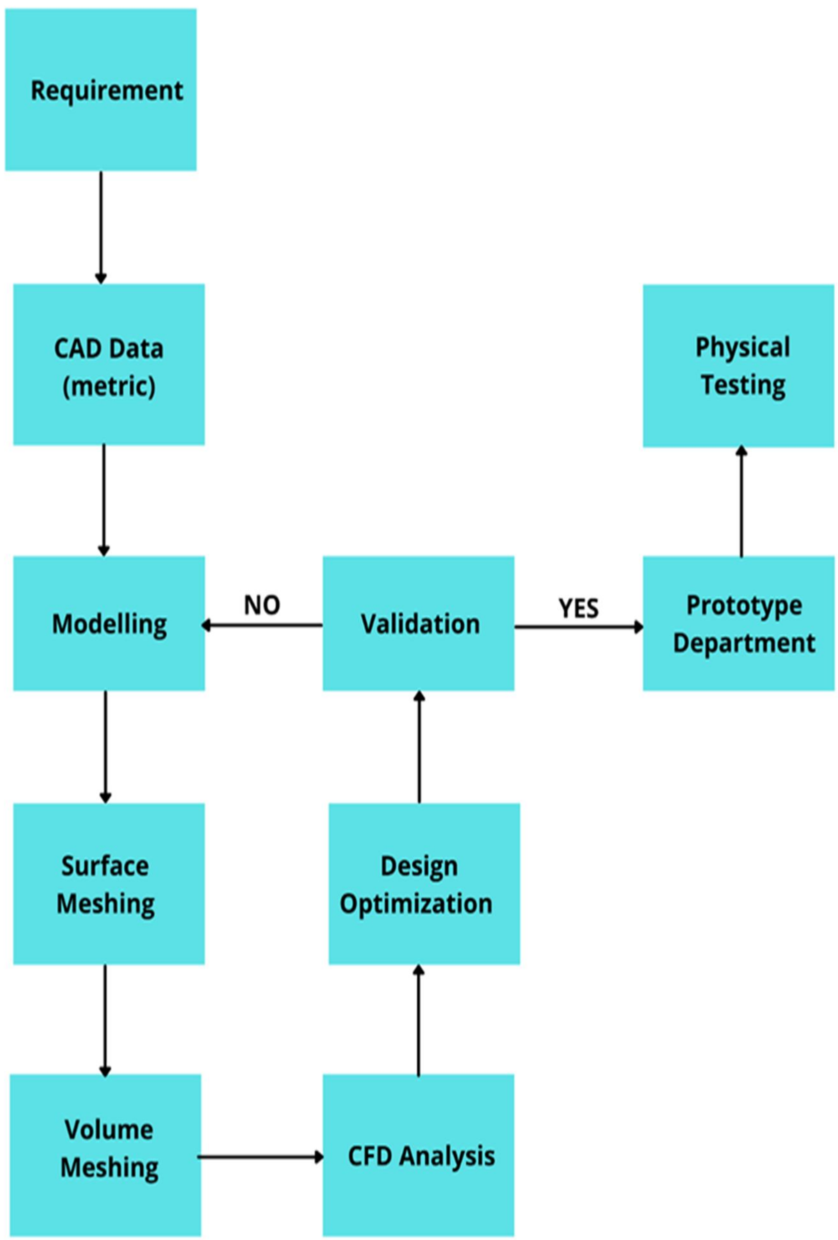


Fig. 3 Process Sequence for CFD Analysis

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue VII July 2021- Available at www.ijraset.com

Pungaiah & Kailasanathan [10] studied the working parameters of the nano-coated tubes using Computational Fluid Dynamics (CFD) and Taguchi methods. The CFD and Taguchi methods were used for the design of experiments to analyze the impact of nano-coated radiator the coolant fluid was transmitted at three different mass flow rates, at three different coating thicknesses, and coated on the top surface of the radiator tubes. Thermal analysis was performed for three temperatures as heat input conditioning for CFD. The most important parameter for nanocoated radiator tubes is the orthogonal array, followed by the Signal-to-Noise Ratio (SNRA) and the variance analysis (ANOVA). A proper orthogonal array was then selected and tests are carried out. The findings of ANOVA showed 95% confidence and were confirmed in the most significant parameters. The optimal values of the parameters are obtained with the results of taguchi analysis parameters and the parameters having a significant impact on the efficiency of the radiator. Fig. 5 shows flow chart process of Taguchi Analysis. The CFD and Taguchi methodology studies showed that all of the above-mentioned parameters contribute equally to the rate of heat transfer, effectiveness, and overall heat transfer coefficient of the nanocoated radiator tubes. Experimental findings were examined to assess the adequacy of the proposed method. In the study. Fig 4 shows results of Taguchi analysis.

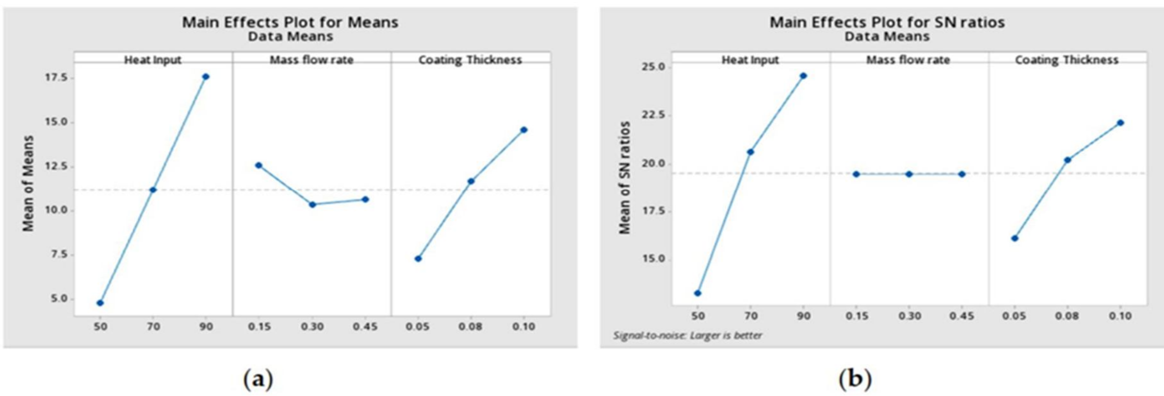


Fig. 4 Results of Taguchi Analysis [10]

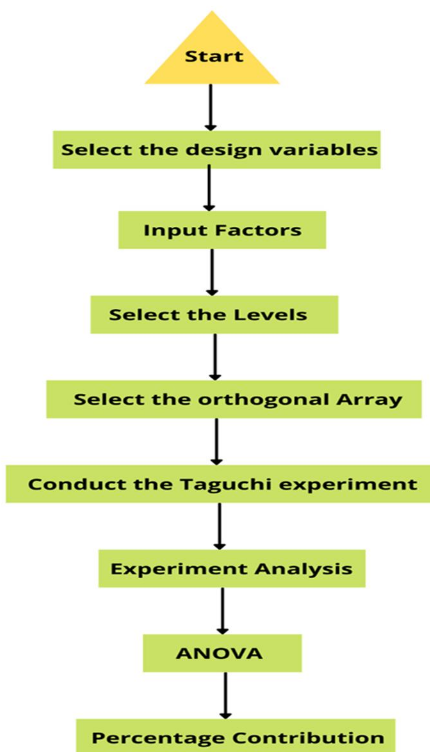


Fig. 5 Flow chart process of Taguchi Analysis

Aydına et al. [11] developed a radiator specifically for unmanned aerial vehicles and was analyzed using computational fluid dynamics (CFD). The radiator consisted of turbulator pins (core side) placed between two vertical columns (column side). CFD modeling of the radiator, which was developed, prototyped, and tested by Kale Oto radiator Co. was performed. Modeling of the turbulators and the fins outside the radiator channels is almost impossible due to the tremendous number of required cells. Therefore, the porous media approach was adopted for the core side. Radiator tests and CFD results showed very good agreement. The maximum differences between experimental and CFD analyses results were 5.6% for pressure drop, 3.1% for radiator outlet temperature, and 2.9% for heat capacity. It was concluded that a proper porous media model can be effectively used for both design and optimization considerations.

Chacko et al. [12] had designed a radiator cover to increase the radiator efficiency by air flow optimization. They started with CFD model of baseline model and it was validated against test data. CAD data were imported from AVE (Advanced Vehicle Engineering) and clean up using ANSA and surface mesh generation using ANSA then volume mesh generation using TGRID. CFD analysis was done by FLUENT and design optimization by ANSA level. Final optimized design CAD data were sent to AVE for validation. This process went up to four iterations. In fourth iteration the hot air recirculation was reduced to maximum extent which result into increase of average velocity through radiator core from 4.2m/s to 5.6m/s that is 34% against baseline case.

Pal et al. [13] designed radiator cover or under-hood by different methodology and different tools to reduce hot air recirculation in radiator cover by its better design options and IRFM sealing around radiator in new vehicle for air flow optimization to increase radiator efficiency. Forced convection was considered for cold simulation for baseline model to analyses flow and velocity distribution and results obtained were modified to increase flow characteristic and heat dissipations. KULI software for computing heat rejection, outlet temperature of coolant and charged air. Steady flow is assumed and RAN's one equation turbulence (Spalart-Allmaras) is used to model turbulence. Average velocity of air and outlet temperature of coolant both increased as compared to baseline model and above 80% correlation was obtained against test data by CFD analysis.

Chavan et al. [14] proposed to have circular radiator core where all conventional are in either square or rectangle shaped which have following disadvantages; fan deliver air flow in circular shape so its air distribution is not uniform over the entire core less at corners and almost zero at center along axial direction. CFD model had following new specifications: no material at center area which is equivalent to fan's hub area, design of tubes and the fins are so arranged that the outlet air had nearly constant velocity, fins of varying depth, maximum at the outer periphery which reduces along the inner periphery.

Bengt et al. [15] studied performance of compact heat exchanger in which example of automobile radiator was also discussed. Various turbulence models were discussed through CFD: zero equation, one equation, two equation, Reynold's stress, algebraic stress and large eddy simulation models. General concepts about FVM method, boundary conditions and about discretization were explained. The effect on fin efficiency was not large but increases as the convective heat transfer coefficient increases. By CFD analysis it was found that the temperature at inlet manifold is uniform. Below the inlet flow rate is high so temperature drop is small. Greater temperature change was found at right and left edges where flow rate is small. By optimized design with the help of CFD we can reduce this non uniformity in temperature distribution.

P. K. Trivedi, N. B. Vasava [16] illustrated the effect of Tube pitch for best configured radiator for optimum performance. Heat transfer increases as the surface area of the radiator assembly is increased. This leads to change the geometry by modifying the arrangement of tubes in automobile radiator to increase the surface area for better heat transfer. The modification in arrangement of tubes in radiator is carried out by studying the effect of pitch of tube by CFD analysis using CFX. Results Shows that as the pitch of tube is either decreased or increased than optimum pitch of tubes, the heat transfer rate decreases.

Wernik et al. [17] worked on the physical model in COMSOL Multiphysics package, a 3D model of a radiator has been developed. Using the finite element method, a number of numerical simulations were carried out. The results of the temperature distribution on the radiator surface were obtained. Numerical methods are subject to some errors, so it is important to validate the results. In order to verify the correctness of the numerical model, a test stand equipped with a radiator, a heating resistor, a fan and a FLIR SC7600 thermal imaging camera was constructed.

The heating resistor allowed the regulation of the heat flux by changing the DC voltage. Thermograms were obtained showing the temperature distribution on the radiator surface. It has been found out that it is possible to use thermo vision in laboratory tests to validate numerical models. The validation of the numerical results was carried out by comparing the results of the numerical simulations with temperature measurements in the real conditions. Therefore, both techniques can be methodically used when designing new equipment.

B. Experimental Analysis on Radiator

The key experimental analysis for determination of performance of radiator using various techniques such as CFD, ANOVA, SNRA, FLUENT, ANSA, etc. are discussed here:

JChen et al. [18] described that the radiator's working performance including heat dissipation rate, coolant pressure drop and air pressure drop mainly depend on its operating parameters such as the inlet coolant temperature, coolant flow volume rate and air velocity under given structural parameters and geometry. In order to investigate the effects of inlet coolant temperature, coolant volume flow rate and cooling air velocity on heat performances of the radiator, the experimental scheme was designed with the ternary quadratic form polynomial regression combinatorial design method. On the basis of the experimental results, it was concluded that the heat performances of the radiator clearly depended on its operating parameters. In the range of given experimental parameters the following conclusions could be obtained:

1) The heat dissipation rate of the radiator grew as the inlet coolant temperature, coolant volume flow rate and cooling air velocity increased.
2) The results also revealed that the air pressure drop and the coolant pressure drop were directly proportional to the air velocity and the coolant volume flow rate respectively.
3) Using the optimization technique of the experiment in the wind tunnel test of the radiator to investigate the effects of the operating parameters on the working performances of radiator was feasible.

Oliet et al. [19] studied different factors which influence radiator performance. It includes air and coolant flow, fin density and air inlet temperature. It is observed that heat transfer and performance of radiator strongly affected by air and coolant mass flow rate. As air and coolant flow increases cooling capacity also increases.

Shubham Sharma [20] illustrated the effect of Al/water nanofluids on the rheological performance of an automobile car radiator has been investigated. Nanofluids were fabricated by two-step methods, i.e., dispersing of aluminum metal bases nanoparticles of size 75–135 nm in double-distilled water. Experiments were conducted on single-pass cross-flow compact heat exchanger by varying the various parameters such as inlet temperature, flow rate through the heat exchanger, concentration of nanoparticles and velocity of air employed for cooling purpose. It was concluded that the hot side Nusselt numbers are improved by 3.37 and 5.0877% for 0.2 and 0.3% concentrations of nanofluids, respectively, at 318.15 K inlet fluids temperature as compared to base fluids. Colburn factor was increased by 12.94 and 23.45% for 0.2 and 0.3% nanoparticles volume concentration of nanofluids, respectively, at 318.15 K inlet temperature with respect to double-distilled water. Hot fluid side friction factor was increased by 14.04 and 20.916% for 0.2 and 0.3% nanoparticles volume concentration of nanofluids with respect to base fluids, but this average value of friction factor was decreased by 2.29 and 9.1412% when temperature was increased from 318.15 to 323.15 K and 328.15 K, respectively.

IV. PERFORMANCE PARAMETER

A. Coolant parameters

1) Nusselt Number

The Nusselt number was found for a rectangular cross section for fully developed laminar flow. The ratio of width over height of the tube is used.

$$Nu_{eg}=0.023\times Re_{eg}^{0.8}\times Pr_{eg}^{0.4} \qquad (1)$$

Re_{eg} = Reynolds number of ethylene glycol
Pr_{eg} = Prandtl number of ethylene glycol

2) Reynolds Number

The Reynolds number is the ratio of inertial forces to viscous forces. The Reynolds number is a dimensionless number used to categorize the fluids systems in which the effect of viscosity is important in controlling the velocities or the flow pattern of a fl.
Where,

$$R_{eg}=\frac{D_{hyd}\times\rho\times V}{\mu} \qquad (2)$$

D_{hyd}= Hydraulic diameter

3) Hydraulic diameter

The hydraulic diameter must be used because it is a non-circular cross section. The hydraulic diameter can then be used to estimate the Reynolds number. The equation for the hydraulic diameter calls for the wetted perimeter of the tubes. However, the difference in the outer and inner tube dimensions is so negligible that the outer perimeter is used for convenience.

Hydraulic diameter,

$$D_{hyd} = \frac{4A_{tube}}{P_{tube}} \quad (3)$$

Where,

A_{tube}= Area of the radiator tube

P_{tube} = Perimeter of the radiator tube

B. Air Parameters

1) Velocity of Air

$$V_{air} = \frac{Q_{air}}{A_{radiator} - (N_{tube} H_{tube} L_{radiator})} \quad (4)$$

Here,

Q_{air} = Total air volumetric flow rate

$A_{radiator}$ = Area of the radiator

N_{tube} = Number of tubes

H_{tube} = Height of the tube

$L_{radiator}$ = Length of the radiator

2) Reynolds Number

$$Re_{air} = \frac{V_{air} W_{fin}}{\nu_{air}} \quad (5)$$

Here,

V_{air} = Velocity of air

W_{fin} = Width of the fin

ν_{air} = Kinematic viscosity of air

3) Convective Heat Transfer Coefficient For Air Flow

$$h_{air} = \frac{Nu_{air} \times k_{air}}{W_{tube}} \quad (6)$$

Where,

Nu_{air} = Nusselt number of air

k_{air} = Thermal conductivity of air

W_{tube} = Width of the tube

V. ASSUMPTIONS MADE BY RESEARCHERS

Heat transfer analysis of a radiator in an automobile radiator in an automobile engine is done by most of the researchers considering following assumptions:

1) The radiator operates under steady-state conditions that is constant flow rate and coolant temperatures at the inlet and within the radiator are independent of time.
2) There are no thermal energy sources and sinks in radiator walls or coolant.
3) Either there are no phase changes in the coolant stream flowing through the exchanger.
4) The specific heats of ethylene glycol and air are constant throughout the radiator.
5) The fluid flow rate is uniformly distributed through the radiator on each fluid side in each pass. No flow stratification, flow bypassing or flow leakages occur in any stream.
6) Kinetic energy and potential energy changes are negligible.

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue VII July 2021- Available at www.ijraset.com

VI. CONCLUSION

The speed range of air striking on the fins of radiator for heat transfer analysis was kept in the range of 15 kmph to 75 kmph for analysis with ethylene glycol as coolant ,and the outcomes obtained by the researchers are as follows:

1) Nusselt number of the air is calculated for determination of convective heat transfer coefficient, and increases as the Reynolds number of the air increases,
2) The heat transfer coefficient values get increased , when the velocity of the air striking the radiator increases.
3) It is also observed that, at higher velocity of air striking the radiator, the Reynolds number is higher and as a result of it the efficiency of the fins is reduced slightly.
4) Overall heat transfer coefficient is the function of the heat transfer coefficient of the air as well as the coolant used (ethylene glycol).
5) Most researchers agreed that increasing the volume fraction more than (1 vol%) could decrease the heat transfer rate.
6) When engines run at high values of rpm to increase the speed of the vehicle, the heat generated in the parts of the engine also increases drastically. Hence, at higher speed the cooling process should also be effective in order to dissipate the heat to the atmosphere.

It is concluded by this analysis that, even at higher speed the given dimensioned radiator with given number of fins attached to it works properly with slight compromise in the decrease in efficiency of the fins used in the radiator.

VII. NOMENCLATURE

L	Length, m
H	Height, m
W	Width, m
D	Diameter, m
A	Cross-sectional area, m_2
P	Perimeter, m
V	Velocity, m/s
Q	Volumetric flow rate, m_3/s
N	Number
Re	Reynolds number
h	Convective heat transfer coefficient W/m2-K
Nu	Nusselt number
k	Thermal conductivity, W/m-K
Pr	Prandtl number
m	Coefficient for calculating efficiency
UA	Overall heat transfer coefficient

A. Greek Symbols

μ	Dynamic viscosity
ν	Kinematic viscosity
η	Efficiency

B. Suffixes

eg	Ethylene glycol
f	Fin
b	Base
hyd	Hydraulic
rad	Radiator

REFERENCES

[1] Y. Gao, H. Wang, A.P. Sasmito, A.S. Mujumdar, Measurement and modeling of thermal conductivity of graphene nanoplatelet water and ethylene glycol base nanofluids, Int. J. Heat Mass Transf. 123 (2018) 97–109.

[2] Z.J. Fan, Z.Q. Gu, S.C. Liu, Prediction of heat dissipation effect by simulating radiator group of mining dump-truck, Appl. Mech. Mater. 217–219 (2012) 2473–2479.

[3] R.K. Suresh, K. Ganesan, M. Mohan Prasad, An optimal balancing of multiple assembly line for a batch production unit, Int. J. Lean Think. 4 (2) (2013) 22–32.

[4] M. Mohan Prasad, K. Ganesan, K.P. Paranitharan, R. Rajesh, An analytical study of lean implementation measures in pump industries in India, Int. J. EnterpriseNetw. Manage. 10 (2) (2019) 133–151.

[5] R. Taylor, S. Coulombe, T. Otanicar, P. Phelan, A. Gunawan, W. Lv, G.Rosengarten, R. Prasher, H. Tyagi, Small particles, big impacts: a review of thediverse applications of nanofluids, J. Appl Phys. (2013).

[6] S.M. Sutharsan, M. Mohan Prasad, Integrated of lean principles with sustainablemanufacturing, Int. J. Lean Think. 3 (1) (2012) 102–106.

[7] A. Patra, S. Saha Ray, Analysis for fin efficiency with temperature dependentthermal conductivity of fractional order energy balance equation using HPST method, Alex. Eng. J. 55 (2016) 77–85.

[8] K. Ganesan, M. Mohan Prasad, R.K. Suresh, Lead time reduction through lean technique in an monoblock (swj1hp) pump industry, Int. J. Appl. Mech. Mater. 41 (13) (2014) 3075–3090.

[9] V. Delavari, S.H. Hashemabadi, CFD simulation of heat transfer enhancement of Al2O3/water and Al2O3/ethylene glycol nanofluids in a car radiator, Appl.Therm. Eng. 73 (2014) 380–390.

[10] Pungaiah, S.S. and Kailasanathan, C.K., 2020. Thermal Analysis and Optimization of Nano Coated Radiator Tubes Using Computational Fluid Dynamics and Taguchi Method. Coatings, 10(9), p.804.

[11] Omprakash Pal, Rajesh Joshi, and Sanjay T. Purkar, VSRD International Journal of Mechanical, Civil, Automobile and Production Engineering, Vol.III Issue X October 2013 e- ISSN:2249-8303, P-ISSN: 2319-2208

[12] Chavan D.K., and Tasgaonkar G.S., International Journal of Mechanical and Production Engineering Research and Development, ISSN2249-6890, Vol. 3, Issue 2June 2013, 137-146,

[13] BengtSunden,DepartmentofEnergySciences,Heat Transfer,LundUNniversity,Lund,Sweden,Internat ionalJournalofNumericalMethodsforHeatandFlui dFlow,Vol20No.5,2010,pp551-569

[14] J A Chen, D F Wang and L Z Zheng, 2001, “Experimental study of operating performance of a tube-and-fin radiator for vehicles”, Proceedings ofInstitution of Mechanical Engineers, Republic of China, 215: pp. 2-8.

[15] P. K. Trivedi, N. B. Vasava, “Effect of Variation in Pitch of Tube on Heat Transfer Rate in Automobile Radiator by CFD Analysis.” International Journal of Engineering and Advanced Technology (IJEAT) ISSN: 2249 – 8958, Volume-1, Issue-6, 2012

[16] C. Oliet, A. Oliva, J. Castro, C.D. Pe´rez-Segarra, “Parametric studies on automotive radiators”, Applied Thermal Engineering, 27, 2007

[17] Wernik, J., Grabowski, M. and Wołosz, K.J., 2018. Thermal analysis of radiator under natural and forced convection conditions using numerical simulation and thermography. Chemical Engineering Transactions, 70, pp.1501-1506.

[18] Sharma, S., 2018. Fabricating an experimental setup to investigate the performance of an automobile car radiator by using aluminum/water nanofluid. Journal of Thermal Analysis and Calorimetry, 133(3), pp.1387-1406.

[19] Omprakash Pal, Rajesh Joshi, and Sanjay T. Purkar, VSRD International Journal of Mechanical, Civil, Automobile and Production Engineering, Vol.III Issue X October 2013 e- ISSN:2249-8303, P-ISSN: 2319-2208

[20] Chavan D.K., and Tasgaonkar G.S., International Journal of Mechanical and Production Engineering Research and Development, ISSN2249-6890, Vol. 3, Issue 2June 2013, 137-146,

[21] BengtSunden,DepartmentofEnergySciences,Heat Transfer,LundUNniversity,Lund,Sweden,Internat ionalJournalofNumericalMethodsforHeatandFlui dFlow,Vol20No.5,2010,pp551-569

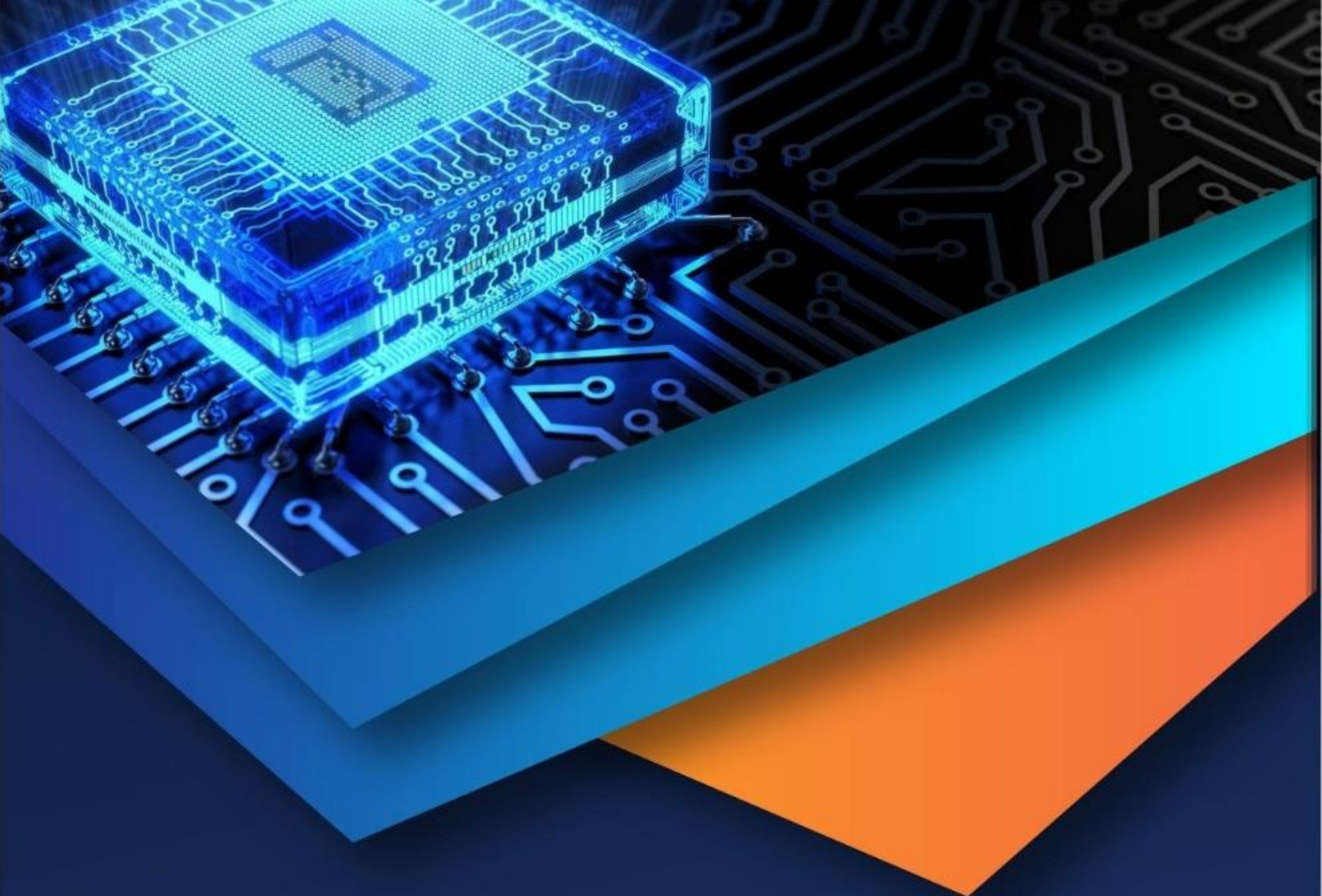
[22] J A Chen, D F Wang and L Z Zheng, 2001, “Experimental study of operating performance of a tube-and-fin radiator for vehicles”, Proceedings ofInstitution of Mechanical Engineers, Republic of China, 215: pp. 2-8.

[23] P. K. Trivedi, N. B. Vasava, “Effect of Variation in Pitch of Tube on Heat Transfer Rate in Automobile Radiator by CFD Analysis.” International Journal of Engineering and Advanced Technology (IJEAT) ISSN: 2249 – 8958, Volume-1, Issue-6, 2012

[24] C. Oliet, A. Oliva, J. Castro, C.D. Pe´rez-Segarra, “Parametric studies on automotive radiators”, Applied Thermal Engineering, 27, 2007

[25] Wernik, J., Grabowski, M. and Wołosz, K.J., 2018. Thermal analysis of radiator under natural and forced convection conditions using numerical simulation and thermography. Chemical Engineering Transactions, 70, pp.1501-1506.

[26] Sharma, S., 2018. Fabricating an experimental setup to investigate the performance of an automobile car radiator by using aluminum/water nanofluid. Journal of Thermal Analysis and Calorimetry, 133(3), pp.1387-1406.



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY