International Journal of Innovative Technology and Exploring Engineering (IJITEE)
ISSN: 2278-3075, Volume-8 Issue-8 June, 2019

Heat transfer Analysis of Automobile Radiator with Helical Tubes using CFD

S. Sudhakar babu, Shaik Yasin

Abstract*: To guarantee smooth running of a vehicle under the factor load conditions, cooling systems have to perform effectively. The coolant traces the engine block absorbs the heat, when streamed through the radiator loses heat to surrounding. This is a cyclic process and coolant is circulated in a closed loop in the engine block and radiator. Computational fluid dynamics (CFD) analysis is performed in ANSYS fluent to analyze heat transfer process. Geometric model of a 55hp engine radiator was considered and geometric model was designed in CREO. Comparative study between straight and helical tubes was performed considering varying mass flow rates of coolants. Further analyses were performed by varying coolants from conventional fluids air, water to Nano-fluids with different volume fractions. Temperature distributions, heat transfer coefficient, pressure distributions in the radiator are reported in the present work.

***Index Terms*: Engine radiator, Heat transfer analysis, Nano-materials, Temperature distribution, CFD analysis**

I. INTRODUCTION

Radiator is a heat transferring device used to exchange heat energy from one fluid stream to another either for heating or cooling. Radiators finds their use in vehicles, cooling of electronic chips, unconventional cooling of buildings. In automobile industry radiators are used to cool the engine block by convecting heat to surroundings. Even though radiation is un-avoidable its percentage contribution in heat transfer process is very small compared to convection mode of heat transfer. Spacecraft radiators necessarily must use radiation only to reject heat.

The Roman hypocaust is typical example ofradiator for space heating purpose. The very first radiator for heating application was invented by F. S. Galli, Russian businessman stationed at Petersburg, between in 1857

Fig. 1 Water air convective cooling radiator

The literature review in this thesis is taken from paper done by G. C. Junjanna [1]. Numerical simulation was performed in ANSYS Fluent 13.0 on compact heat exchanger. The effect of geometric parameters, flow rates, louver pitch was analyzed. Optimal values of flow rates, and louver pitch, and geometric parameters were stated for maximization of heat transfer. Insights on heat transfer process, corresponding contours were reported.Bharat Raj Singh and et al. [2]have performed numerical simulations to evaluate the performance of radiator with varying flow rates of the coolants. Complete information on the heat transfer mechanism was reported by performingparametric analysis by performing numerical methods and comparing it with analytical solutions. Finite difference method was used to perform numerical simulation and resistance concept was used to obtain analytical solution. It was reported that water was better coolant than propylene glycol. It was concluded that, water being a better coolant has a disadvantage of enhancing corrosion by dissolved salts in it. Transfer of heat from a radiator occurs by either diffusion and radiation mechanism [3]. In practice radiator is a term coined to any device in which fluid traces a specified path through pipes (may be of different kinds e.g. straight and spiral). Some times fins are also attached to enhance the heat transfer by increasing the effective heat transfer area.

A. Working principle of radiator

It is a simple cap, with a pressure relief valve, either manual or automatic, to maintain pressure in a closed system [4] [5] [6]. By maintaining roughly 7-20 psi, the boiling point of water can be raised by up to 270-290 degrees Fahrenheit, as well as lowering the freezing point.A radiator top is a coupled weight alleviation valve containing the primary valve itself (the rubber treated circle that fits the filler gap) with an adjusted spring behind it as shown inFig. 2. At the point when adequate weight expands upon inside the radiator to drive the valve to open against the contradicting spring weight, the radiator's weight is soothed past the valve – ordinarily into the flood repository. Contrary to this there is a small valve that does exactly opposite-when atmospheric pressure exerted on the overflow reservoir's contents is greater than the pressure inside the radiator (engine cold, prior overflow, resulting in a vacuum), coolant from the overflow reservoir is permitted to return to the radiator. This secondary valve operates in exactly the same manner as the larger primary valve, just in the opposite direction.

Revised Manuscript Received on June 05, 2019

S. Sudhakar Babu,Associate Professor, Department of Mechanical Engineering, Koneru Lakshmaiah Education Foundation (KLEF), Guntur. India

Shaik Yasin^{2*}, Postgraduate student, Department of Mechanical Engineering, Koneru Lakshmaiah Education Fondation (KLEF), Guntur. India

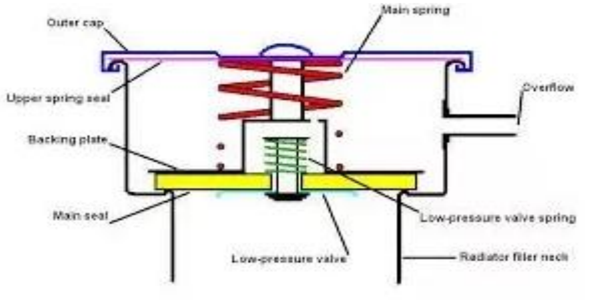


Fig. 2 Working of radiator

Almost all vehicles in the automobile industry today have a heat exchanger called as radiator. It is an essential component of cooling systems of the vehicle as shown in Fig. 3. It is one among of the many other sub-systems in the engine.

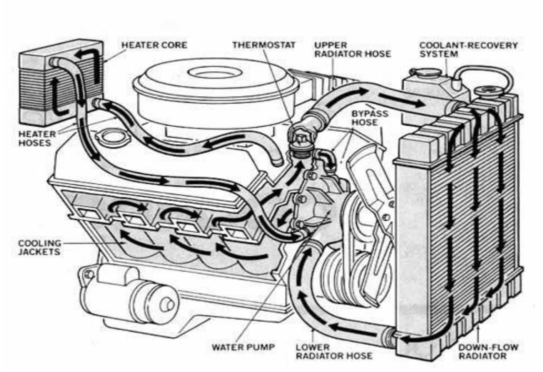


Fig. 3 Coolant path and Components of an Automobile Engine Cooling System

Radiators in the present automobiles are fabricated by aluminum. These radiators are made by brazing slim aluminum fins to smoothed aluminum tubes. The coolant streams from the inlet to the outlet through numerous cylinders mounted in a parallel arrangement. The fins lead the heat from the cylinders and exchange it to the air coursing through the radiator. Current radiator plans are incredibly constrained and have not encountered any significant progression as of late. As depicted over, the fundamental issue is that present radiators experience an extensive protection from heat exchange brought about via air streaming over the radiator. Current radiators additionally encounter head opposition, are extremely cumbersome, and force impediments on the plan of the vehicle.

II. PROBLEM DESCRIPTION

Radiators which are the heart of the cooling system of modern vehicles is considered for analysis in ANSYS fluent. Geometric details of the radiator are considered as per the guidelines stated in [7]. 2-D turbulent flow with heat transfer is performed, in a radiator with straight and helical tube. In the analysis, effect of variation of mass flow rates of the coolant was considered, with varying coolant fluids such as Air, water and Nano-fluids. Further in order to visualize the effect of volume fraction in the Nano-fluids, 2 analysis was performed at two different volume fractions. Variation of heat transfer coefficient, variation of pressure, Temperature contour are reported.

III. MATHEMATICAL DESCRIPTION

The fluid flow behavior through the pipe can be described by two governing equations namely continuity and momentum equations, and thermal characteristics can be described by energy equation.

The continuity equation for a finite-sized control volume in differential form is given as

$$\frac{\partial \lambda}{\partial t}+\frac{\partial(\lambda w_i)}{\partial e_i}=0; i=1,2 \qquad \text{Eq. (1)}$$

Where ‘t’ is the time, e_i is the coordinate axes and w_i is the velocity in the direction ofe_i and λ is the density of the fluid considered.Whenever fluid flows over a surface due to viscous effects, airfoil experiences normal and tangential forces. These forces and their influence can be evaluated by momentum equation also known in literature as Navier-Stokes (N-S) equation. The N-S equation is given as:

$$\begin{aligned}&\frac{\partial(\lambda w_i)}{\partial t}+u_j\frac{\partial(\lambda w_i)}{\partial e_j} \qquad \text{Eq. (2)}\\ &=-\frac{\partial P}{\partial e_i}+\frac{\partial}{\partial e_j}\left(\mu_v\frac{\partial w_i}{\partial e_j}\right)\\ &+ \qquad \rho H_i\; i,j=1,2\end{aligned}$$

where P is the pressure distribution, μ_v is the dynamic viscosity, H_i is the resolution of body force in i^{th}direction. The velocity components obtained from the continuity equation must satisfy the N-S equations. From the literature it is found that turbulent model $k-\varepsilon$ is widely accepted and used in industrial applications [1]. Even though large eddy simulations (LES) model which yields better results and is easier to model than $k-\varepsilon$ model, is however time consuming and hence $k-\varepsilon$ model is used in the present study Reynolds Averaged Navier Stokes (RANS) equation, which is obtained by applying time averaging to the Navier Stokes equation and is given as is solved in turbulent modelling and is given as

$$\begin{aligned}&\frac{\partial\widehat{w_i}}{\partial t}+\widehat{w_j}\frac{\partial(\widehat{w_i})}{\partial e_j}=-\frac{1}{\lambda}\frac{\partial\hat{P}}{\partial e_i} \qquad \text{Eq. (3)}\\ &\qquad+\frac{\partial}{\partial e_j}\left(\gamma\frac{\partial\widehat{w_i}}{\partial\widehat{w_j}}-\hat{\tau}_{ij}\right)\\ &i,j=1,2\end{aligned}$$

In the Eq. (3), the caps on the variable represents the time averaged quantities and τ_{ij} is the shear-stress tensor. The behavior of eddies in the flow is governed by Reynolds-Stress tensor and difficulty lies in the non-existent macroscopic terms to model it. In order to get a solution to the Eq. (3), Reynolds-Stress term is modeled utilizing the $k-\varepsilon$ model. In the $k-\varepsilon$ two equation model, the turbulent viscosity is related to the turbulent kinetic energy (k) and the rate of dissipation of the eddy’s energy (ε) and is given as

$$\mu_t=\lambda C_\mu\frac{k^2}{\varepsilon} \qquad \text{Eq. (4)}$$

From [1], the compete mathematical description of $k-\varepsilon$ is given as

$$\frac{\partial(\lambda k)}{\partial t} + u_i \frac{\partial(\lambda k)}{\partial e_i} = \frac{\partial}{\partial e_i}\left(\frac{\mu_t}{\sigma_t}\frac{\partial k}{\partial e_i}\right) + Y - \lambda\varepsilon \qquad \text{Eq. (5)}$$

$$\frac{\partial(\lambda \varepsilon)}{\partial t} + u_i \frac{\partial(\lambda \varepsilon)}{\partial e_i} = \frac{\partial}{\partial e_i}\left(\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial e_i}\right) + C_{1\varepsilon}\frac{\varepsilon}{k}G - C_{2\varepsilon}^{*}\lambda\frac{\varepsilon^2}{k} \qquad i = 1{,}2 \qquad \text{Eq. (6)}$$

The other terms in the above equation are found using the equations

$$Y = 2\mu_t S_{ij} S_{ij} \qquad \text{Eq. (7)}$$

$$C_{2\varepsilon}^{*} = C_{2\varepsilon} + C_{2\varepsilon}' \qquad \text{Eq. (8)}$$

$$C_{2\varepsilon}' = \frac{C_\mu \lambda \eta^3 \left(1 - \frac{\eta}{\eta_0}\right)}{1 + \beta\eta^3} \qquad \text{Eq. (9)}$$

$$\eta = S\frac{k}{\varepsilon} \qquad \text{Eq. (10)}$$

$$S = \sqrt{2S_{ij}S_{ij}} \qquad \text{Eq. (11)}$$

WhereS_{ij} is the shearing tensor, and g_i is the body force in the x_i direction. The coefficients needed in the model are to be fed in the ANSYS fluent. In the present study, default values suggested by ANSYS fluent are taken into consideration.

Further, since Nano-fluids is used, the effective thermo-physical properties are to be used. The correlations used for the determination of the properties are as follows

$$\rho_{nf} = \emptyset \times \rho_s + [(1 - \emptyset) \times \rho_w] \qquad \text{Eq. (12)}$$

Where,

ρ_{nf}	Is density of the Nano fluid
ρ_s	Is density of the Nano particles
$\emptyset$	Is the volume fraction
ρ_w	Is density of the fluid

Specific heat and viscosity of the Nano-fluid as stated in [8], is given as

$$\mathrm{C}_{p\,nf} = \frac{\emptyset \times \rho_s \times \mathrm{C}\rho_s + (1 - \emptyset)(\rho_w \times \mathrm{C}_{Pw})}{\emptyset \times \rho_s + (1 - \emptyset) \times \rho_w} \qquad \text{Eq. (13)}$$

$$\mu_{nf} = \mu_w(1 + 2.5\varphi) \qquad \text{Eq. (14)}$$

Where,

$\boldsymbol{\mu_{nf}}$	Is the viscosity of Nano-fluid
$\boldsymbol{\mu_w}$	Is the viscosity of fluid (water)

IV. RESULTS AND DISCUSSION

CFD analysis is performed to visualize the flow behavior and heat transfer characteristics. Turbulent flow simulation was performed with the effective properties of the Nano-materials shown in Table I

Table I Effective thermo-physical properties of Nano-fluids

Fluid	**Volume Fraction(φ)**	**Density(Kg/m^3)**	**Specific Heat (J/Kg-K)**	**Viscosity (Kg/M-S)**	**Thermal Conductivity (W/M-K)**
MAGNESIUM OXIDE	0.2	1510.56	2660.95	0.00150045	1.0325
	0.4	2022.92	1910.40	0.0020 06	1.8457
SILICON CARBIDE	0.2	1784.56	2264.21	0.0015 04	1.2481
	0.4	2570.92	5357.01	0.0020 06	2.625

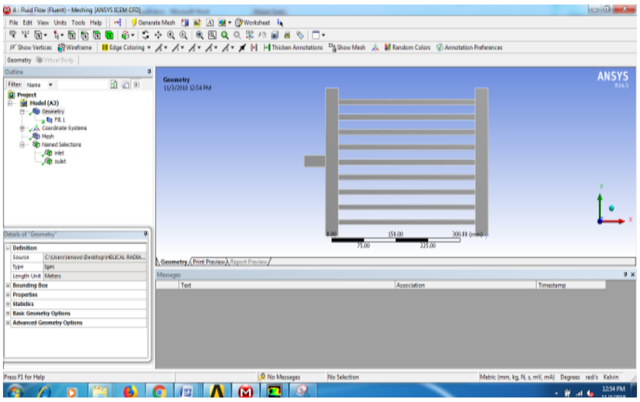
Cases considered for the simulation are

Table II Cases considered

Case/ Parameter	Mass flow rate of the coolant	Tube used	Fluids
Case 1	2.8	Straight	• Water • Air • Nano-Fluids
		Helical	
Case 2	1.5	Straight	
		Helical	

A. *Case 1 (straight tube-Water)*

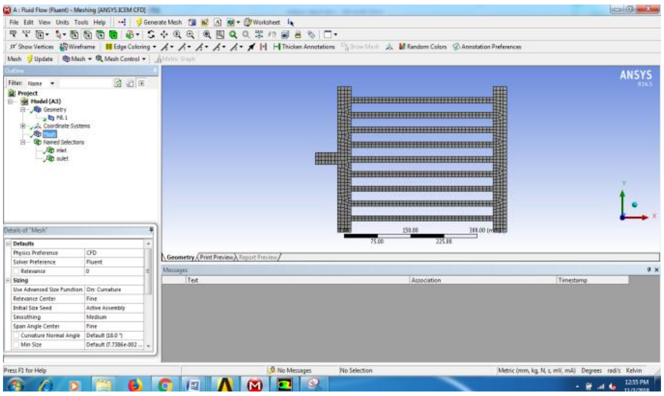
Geometric model of the design was designed in CREO and was imported in Ansys fluent the geometric model of the radiator with straight tube is shown below in Fig. 3



Definition
Source C:\Users\lenovo\Desktop\HELICAL RADIA...
Type Iges
Length Unit Meters

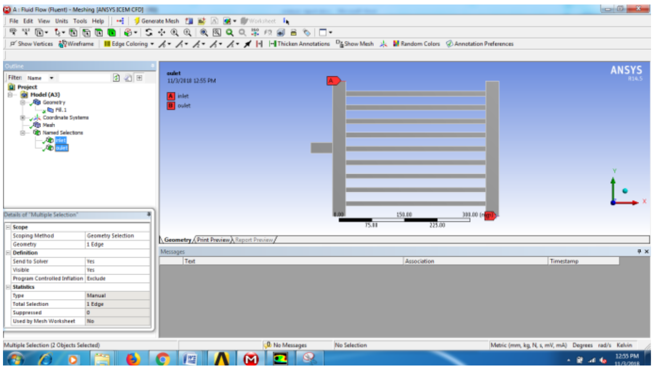
Fig. 3 Geometry considered for CFD simulation

The mass flow rate considered for the simulation, is 2.8 m/s and turbulent flow was assumed. The domain was discretized using 2-d quad elements, in Ansys fluent using Ansys auto mesh feature. The discretized domain is shown in**Fig. *4***. In the presetting, the advanced meshing feature with on curvature option along with fine meshing option was adopted. Once meshing is done, named selections are made with inlet and outlet as named faces to provide a name tag to be used in Ansys fluent, while applying boundary conditions as shown in **Fig. *5***.



Details of "Mesh"
Defaults
Physics Preference: CFD
Solver Preference: Fluent
Relevance: 0
Sizing
Use Advanced Size Function: On: Curvature
Relevance Center: Fine
Initial Size Seed: Active Assembly
Smoothing: Medium
Span Angle Center: Fine
Curvature Normal Angle: Default (18.0 °)
Min Size: Default (7.7386e-002 ...

Fig. 4 Discretized geometry of the radiator



Details of "Multiple Selection"
Scope
Scoping Method: Geometry Selection
Geometry: 1 Edge
Definition
Send to Solver: Yes
Visible: Yes
Program Controlled Inflation: Exclude
Statistics
Type: Manual
Total Selection: 1 Edge
Suppressed: 0
Used by Mesh Worksheet: No

Fig. 5 Shows named selections created in Ansys fluent

The static temperature and temperature plots are shown in **Fig.** ***6*** and **Fig.** ***7***

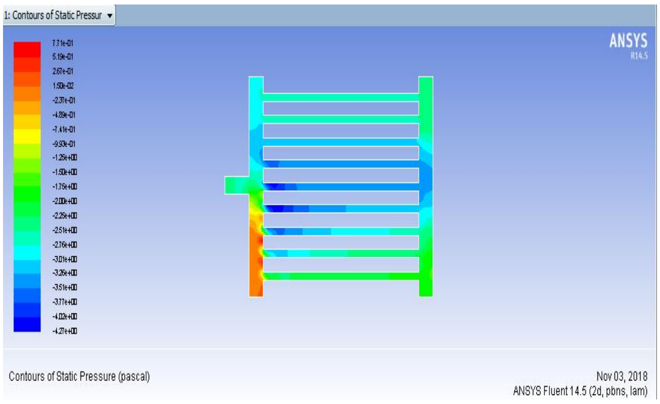


Fig. 6 Static temperature contour for straight tube radiator

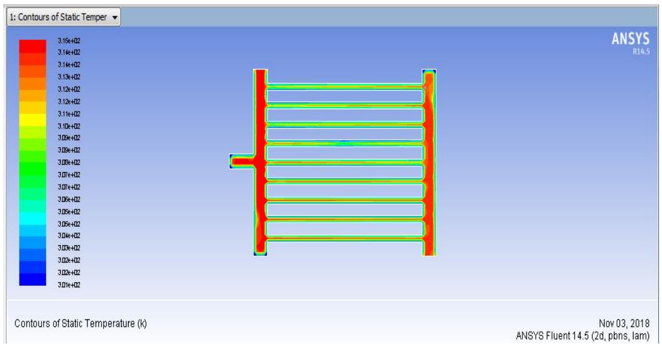
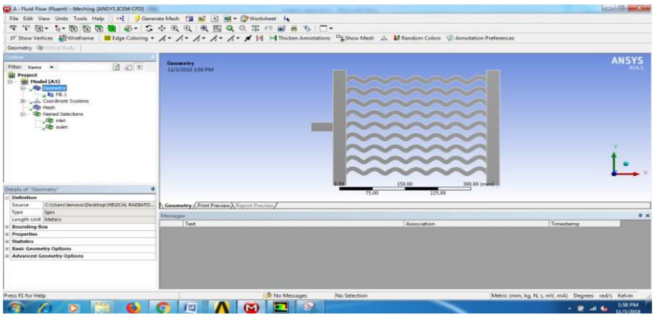


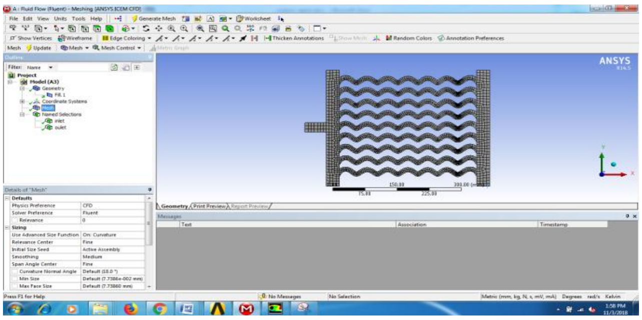
Fig. 7 Temperature contour for straight tube radiator

From **Fig.** ***7***, it can be seen that from a hot temperature of 315K, the coolant gets cooled to 301K as it passes through the straight tubes of the radiator. Similar analysis was performed with helical tubes arrangement. The geometry and the mesh model as shown below in



Details of "Geometry"
Definition
Source: C:\Users\lenovo\Desktop\HELICAL RADIATO...
Type: Iges
Length Unit: Meters
Bounding Box
Properties
Statistics
Basic Geometry Options
Advanced Geometry Options

Fig. 8 Geometric model with helical tubes



Details of "Mesh"
Defaults
Physics Preference: CFD
Solver Preference: Fluent
Relevance: 0
Sizing
Use Advanced Size Function: On: Curvature
Relevance Center: Fine
Initial Size Seed: Active Assembly
Smoothing: Medium
Span Angle Center: Fine
Curvature Normal Angle: Default (18.0 °)
Min Size: Default (7.7386e-002 mm)
Max Face Size: Default (7.73860 mm)

Fig. 9 Discretized model of the helical tube

With the specified mass flow rate and with the above stated boundary conditions, the simulation was performed once again. The temperature and pressure contours are shown in**Fig.** ***10*** and **Fig.** ***11*** respectively.

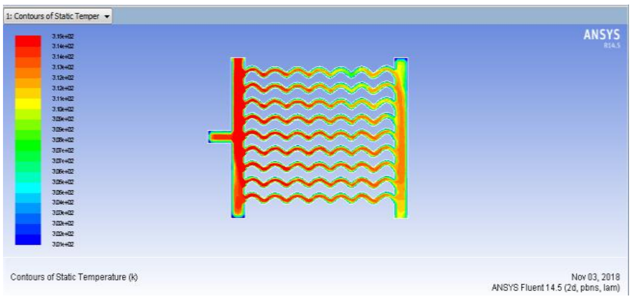


Fig. 10 Static temperature contour for helical tube

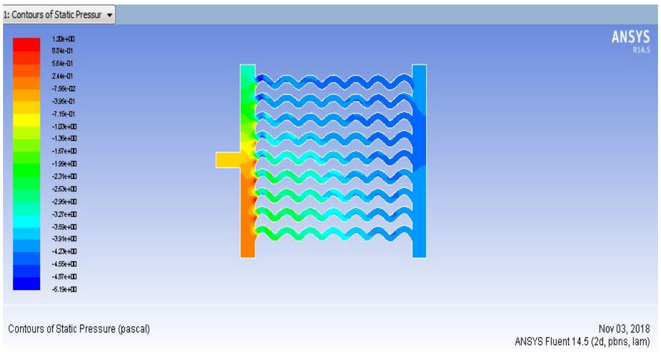


Fig. 11Static pressure contour for helical tube

The above simulation procedure was performed on all the cases stated in Table II. For the Nano-fluids (SiC-Mgo$_2$), two volume fractions were considered to analyze the effect of volume fractions and their thermo-physical properties stated in Table I were used in the simulation. The cumulative results for all the cases considered is elaborated in **Table** ***III*** and **Table** ***IV***. From the **Table** ***III***, it can be interpreted that for case 2 stated in Table II with mass flow rate of 1.5

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

kg/sec, the lowest heat transfer coefficient occurs in the case of air as coolant.

Table III Cumulative results for straight pipe radiator

Fluids	At mass flow rate (kg/s)	Pressure (pa)	Temperature (k)	Heat transfer Co-efficient (w/m^2k)	Mass flow rate (kg/sec)	Heat transfer rate (w)
Water	2.8	$7.71e^{-01}$	$3.13e^{+02}$	$2.14e^{+02}$	2.8002	180391.23
	1.5	$1.86e^{-01}$	$3.15e^{+02}$	$2.138e^{+02}$	1.5001	90263.803
air	2.8	$3.50e^{+02}$	$3.151e^{+02}$	$8.62e^{+00}$	2.800102	46727.32
	1.5	$1.03e^{+02}$	$3.151e^{+02}$	$8.621e^{+00}$	1.50001	24695.932
Mgo_2(0.2)	2.8	$4.75e^{-01}$	$3.15e^{+02}$	$3.68e^{+02}$	2.8000002	101150.24
	1.5	$9.87e^{-02}$	$3.1531e^{+02}$	$3.67e^{+02}$	1.500001	47008.793
Mgo_2(0.4)	2.8	$3.09e\text{-}^{01}$	$3.154e^{+02}$	$6.58e^{+02}$	2.800002	58253.379
	1.5	$6.17e^{-02}$	$3.151e^{+02}$	$6.57e^{+02}$	1.500011	25074.977
Sic(0.2)	2.8	$4.02e^{-01}$	$3.149e^{+02}$	$4.45e^{+02}$	2.8000021	80192.953
	1.5	$8.43e^{-02}$	$3.152e^{+02}$	$4.444e^{+02}$	1.5000014	36127.811
Sic(0.4)	2.8	$2.43e^{-01}$	$3.1511e^{+02}$	$9.36e^{+02}$	2.80000012	194364.36
	1.5	$4.86e^{-02}$	$3.1512e^{+02}$	$9.35e^{+02}$	1.50000012	88308.172

Table IVCumulative results for helical pipe radiator

Fluids	At Mass flow rates (kg/sec)	Pressure (pa)	Temperature (k)	Heat transfer coefficient ($w/m^{2K)}$	Mass flow rate (kg/sec)	Heat transfer rate(w)
Water	2.8	$1.20e^{+00}$	$3.1501e^{+02}$	$2.36e^{+03}$	2.8000002	168575.27
	1.5	$3.58e^{-01}$	$3.151e^{+02}$	$2.364e^{+03}$	1.500015	81321.451
Air	2.8	$7.09e^{+02}$	$3.1502e^{+02}$	$9.54e^{+01}$	2.8000021	46067.237
	1.5	$2.06e^{+02}$	$3.1503e^{+02}$	$8.33e^{+01}$	1.5000111	24075.719
Mgo_2(0.2)	2.8	$8.17e^{-01}$	$3.1503e^{+02}$	$4.07e^{+03}$	2.8000021	88253.705
	1.5	$2.39e^{-01}$	$3.150e^{+02}$	$4.071e^{+03}$	1.50001	39209.043
Mgo2(0.4)	2.8	$6.17e^{-01}$	$3.151e^{+02}$	$7.27e^{+03}$	2.8000021	46923.734
	1.5	$1.74e^{-01}$	$3.1521e^{+02}$	$7.561e^{+03}$	1.5000123	19695.16
Sic(0.2)	2.8	$6.91e^{-01}$	$3.1522e^{+02}$	$4.92e^{+03}$	1.500021	29224.613
	1.5	$2.03e^{-01}$	$3.1510e^{+02}$	$4.921e^{+03}$	1.500021	29224.613
Sic(0.4)	2.8	$4.85e^{-01}$	$3.1510e^{+02}$	$1.03e^{+04}$	2.8000002	166734.98
	1.5	$1.37e^{-01}$	$3.151e^{+02}$	$1.031e^{+04}$	1.500013	72788.58

Further, it can also be seen that maximum heat transfer coefficient occurs for Sic with 0.4 volume fraction. This happens as the addition of Nano-particles to the water enhances the heat transfer capability by increasing the effective volume of the molecules. Further from **Table *IV*** it can be seen that, utilization of Nano-fluids enhances the heat transfer characteristics. But for a given mass flow rate, the heat transfer coefficient is higher when Nano-fluid is used in a radiator with helical tubes.

V. CONCLUSIONS

CFD analysis was performed on a radiator model with varying mass flow rates, and varying tube designs. In order to enhance the heat transfer, Nano-fluids SIC and Mgo_2 were used with 2 different volume fractions.

It is observed that with increase in mass flow rate the heat transfer coefficient increases thereby increasing the coolant efficiency. Further it was found that Nano-fluids have much better tendency to enhance heat transfer than orthodox fluids air and water. In design considerations between straight and helical tubes helical tubes stands way a head in terms of heat transfer and nominal in pressure drop.

REFERENCES

1. A.Witry M.H. Al-Hajeri and Ali A. Bondac, “CFD analysis of fluid flow and heat transfer in patterned roll bonded aluminium radiator”, 3rd International conference on CFD, CSIRO, Melborne, Australia, pp. 12-19, 2003.
2. Hilde Van Der Vyer, Jaco Dirker and Jousoa P Meyer, “Validation of a CFD model of a three dimensional tube-in-tube heat exchanger”, Third International Conference on CFD in the Minerals and Process Industry, CSIRO, Melborne, Australia. pp. 25-32, 2003.
3. J A Chen, D F Wang and L Z Zheng, “Experimental study of operating performance of a tube-and-fin radiator for vehicles”, Proceedings of Institution of Mechanical Engineers, Republic of China, 215: pp. 2-8, 2001.
4. Changhua Lin and Jeffrey Saunders, “The Effect of Changes in Ambient and Coolant Radiator Inlet Temperatures and Coolant Flowrate on Specific Dissipation”, SAE Technical Papers, 2000.
5. Sridhar Maddipatla, “Coupling of CFD and shape optimization for radiator design”, Oakland University. Ph.D. thesis,2001
6. J.P.Holman, Heat transfer, Tata-McGraw-Hill Publications, 2000.
7. Seth Daniel Oduro, “Assessing the effect of dirt on the performance of an engine cooling system”, Kwame Nkrumah University of Science and Technology, PG thesis, 2000.
8. Beard, R. A. and Smith, G. J., "A Method of Calculating the Heat Dissipation from Radiators to Cool Vehicle Engines", SAE Technical Paper 710208, 1971.