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 of radiator 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.

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 performing parametric 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 in Fig. 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.

Fig. 1 Water air convective cooling radiator

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*, Postgraduate student, Department of Mechanical Engineering, Koneru Lakshmaiah Education Foundation (KLEF), Guntur, India

Published By:
Blue Eyes Intelligence Engineering & Sciences Publication
Heat transfer Analysis of Automobile Radiator with Helical Tubes using CFD

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.

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
\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho \omega_i)}{\partial e_i} = 0; \quad i = 1, 2
\]

Where ‘t’ is the time, \( e_i \) is the coordinate axes and \( \omega_i \) is the velocity in the direction of \( e_i \) and \( \rho \) 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:
\[
\frac{\partial (\rho \omega_i)}{\partial t} + \frac{\partial (\rho \omega_i \omega_j)}{\partial e_j} = - \frac{\partial P}{\partial e_i} + \frac{\partial}{\partial e_j}\left(\mu \frac{\partial \omega_i}{\partial e_j}\right) + \rho H_i, \quad i, j = 1, 2
\]

where \( P \) is the pressure distribution, \( \mu \) 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
\[
\frac{\partial \bar{\omega}_i}{\partial t} + \bar{\omega}_j \frac{\partial (\bar{\omega}_i)}{\partial e_j} = - \frac{1}{\lambda} \frac{\partial P}{\partial e_i} + \frac{\partial}{\partial e_j}\left(\frac{\partial \bar{\omega}_i}{\partial e_j} - \tau_{ij}\right)
\]

\[ i, j = 1, 2 \]

In the Eq. (3), the caps on the variable represents the time averaged quantities and \( \tau_{ij} \) 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 \( \varepsilon \) and is given as
\[
\mu_t = \frac{4 k^2}{\varepsilon}
\]

From [1], the complete mathematical description of \( k - \varepsilon \) is given as
\[ \frac{\partial(\lambda k)}{\partial t} + u_i \frac{\partial(\lambda k)}{\partial e_i} = \frac{\partial}{\partial e_i} \left( \mu_i \frac{\partial k}{\partial e_i} \right) + Y - \lambda \varepsilon \]  
Eq. (5)

\[ \frac{\partial(\varepsilon)}{\partial t} + u_i \frac{\partial(\varepsilon)}{\partial e_i} = \frac{\varepsilon}{\rho} \left( \frac{\partial u_i}{\partial e_i} \right)^2 + C_{1\varepsilon} \varepsilon \frac{\partial \varepsilon}{\partial t} + \frac{\partial}{\partial e_i} \left( \frac{\varepsilon}{\rho} \frac{\partial (k)}{\partial e_i} \right) \]  
Eq. (6)

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

\[ C_{2\varepsilon} = C_{2\varepsilon} + C_{2s} \]  
Eq. (7)

\[ C_{2s} = C_{2s} \frac{(1 - \eta)}{\eta^3} \]  
Eq. (8)

\[ \eta = \frac{S}{\varepsilon} \]  
Eq. (9)

\[ S = \sqrt{2S_{ij}S_{ij}} \]  
Eq. (10)

Where, \( S_{ij} \) 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

\[ \rho_{nf} = \phi \times \rho_e + [(1 - \phi) \times \rho_w] \]  
Eq. (12)

Where,

- \( \rho_{nf} \) is density of the Nano fluid
- \( \rho_e \) is density of the Nano particles
- \( \phi \) is the volume fraction
- \( \rho_w \) is density of the fluid

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

\[ C_{p_{nf}} = \frac{\phi \times C_{p_e} + (1 - \phi)(\rho_w \times C_{p_w})}{\phi \times C_{p_e} + (1 - \phi) \times \rho_w} \]  
Eq. (13)

\[ \mu_{nf} = \mu_w (1 + 2.5\phi) \]  
Eq. (14)

Where,

- \( \mu_{nf} \) is the viscosity of Nano-fluid
- \( \mu_w \) 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>
<thead>
<tr>
<th>Fluid</th>
<th>Volume Fraction(( \phi ))</th>
<th>Density(( J/\text{Kg} ))</th>
<th>Specific Heat (J/Kg-( K ))</th>
<th>Viscosity (Kg/( M )-( S ))</th>
<th>Thermal Conductivity (W/M-( K ))</th>
</tr>
</thead>
<tbody>
<tr>
<td>MAGIC</td>
<td>0.2</td>
<td>1510.56</td>
<td>2600.95</td>
<td>0.0015</td>
<td>0.0045</td>
</tr>
<tr>
<td>SUM</td>
<td>0.4</td>
<td>2022.92</td>
<td>1910.40</td>
<td>0.0015</td>
<td>0.0045</td>
</tr>
<tr>
<td>OXI DE</td>
<td>0.4</td>
<td>2570.92</td>
<td>5357.01</td>
<td>0.0020</td>
<td>0.0061</td>
</tr>
</tbody>
</table>

Cases considered for the simulation are

<table>
<thead>
<tr>
<th>Case/ Parameter</th>
<th>Mass flow rate of the coolant</th>
<th>Tube used</th>
<th>Fluids</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1</td>
<td>2.8</td>
<td>Straight</td>
<td>Water</td>
</tr>
<tr>
<td>Case 2</td>
<td>1.5</td>
<td>Straight</td>
<td>Air</td>
</tr>
<tr>
<td>Case 3</td>
<td>1.5</td>
<td>Helical</td>
<td>Nano-Fluids</td>
</tr>
</tbody>
</table>

**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

![Geometry considered for CFD simulation](image-url)

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.
The static temperature and temperature plots are shown in Fig. 6 and Fig. 7. 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 Fig. 8 and Fig. 9 respectively.

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.

The above simulation procedure was performed on all the cases stated in Table II. For the Nano-fluids (SiC-Mgo₂), 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
kg/sec, the lowest heat transfer coefficient occurs in the case of air as coolant.

### Table III Cumulative results for straight pipe radiator

<table>
<thead>
<tr>
<th>Fluids</th>
<th>At mass flow rate (kg/sec)</th>
<th>Pressure (pa)</th>
<th>Temperature (k)</th>
<th>Heat transfer Co-efficient (w/m²k)</th>
<th>Mass flow rate (kg/sec)</th>
<th>Heat transfer rate (w)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Water</td>
<td>2.8</td>
<td>7.71e⁻⁰⁷</td>
<td>3.13e⁺⁰⁰</td>
<td>2.14e⁻⁰⁴</td>
<td>2.80002</td>
<td>1.8039123</td>
</tr>
<tr>
<td></td>
<td>1.5</td>
<td>1.86e⁻⁰⁷</td>
<td>3.15e⁺⁰⁰</td>
<td>2.138e⁻⁰⁴</td>
<td>1.5001</td>
<td>9.0263803</td>
</tr>
<tr>
<td>Air</td>
<td>2.8</td>
<td>3.50e⁻⁰⁷</td>
<td>3.151e⁺⁰⁰</td>
<td>8.62e⁻⁰⁰</td>
<td>2.800102</td>
<td>4.672732</td>
</tr>
<tr>
<td></td>
<td>1.5</td>
<td>1.03e⁻⁰⁷</td>
<td>3.151e⁺⁰⁰</td>
<td>8.621e⁻⁰⁰</td>
<td>1.50001</td>
<td>24.695932</td>
</tr>
<tr>
<td>MgO₂(0.2)</td>
<td>2.8</td>
<td>4.75e⁻⁰⁷</td>
<td>3.151e⁺⁰⁰</td>
<td>3.68e⁻⁰⁰</td>
<td>2.800002</td>
<td>1011590.24</td>
</tr>
<tr>
<td></td>
<td>1.5</td>
<td>9.87e⁻⁰⁷</td>
<td>3.1531e⁺⁰⁰</td>
<td>3.67e⁻⁰⁰</td>
<td>1.50001</td>
<td>47.008793</td>
</tr>
<tr>
<td>MgO₂(0.4)</td>
<td>2.8</td>
<td>3.09e⁻⁰⁷</td>
<td>3.154e⁺⁰⁰</td>
<td>6.58e⁻⁰⁰</td>
<td>2.800002</td>
<td>58.25379</td>
</tr>
<tr>
<td></td>
<td>1.5</td>
<td>6.17e⁻⁰⁷</td>
<td>3.151e⁺⁰⁰</td>
<td>6.57e⁻⁰⁰</td>
<td>1.500011</td>
<td>25074.977</td>
</tr>
<tr>
<td>Sic(0.2)</td>
<td>2.8</td>
<td>4.02e⁻⁰⁷</td>
<td>3.149e⁺⁰⁰</td>
<td>4.45e⁻⁰⁰</td>
<td>2.8000021</td>
<td>80192.953</td>
</tr>
<tr>
<td></td>
<td>1.5</td>
<td>8.43e⁻⁰⁷</td>
<td>3.152e⁺⁰⁰</td>
<td>4.444e⁻⁰⁰</td>
<td>1.5000014</td>
<td>36127.811</td>
</tr>
<tr>
<td>Sic(0.4)</td>
<td>2.8</td>
<td>2.43e⁻⁰⁷</td>
<td>3.1511e⁺⁰⁰</td>
<td>9.36e⁻⁰⁰</td>
<td>2.80000012</td>
<td>194364.36</td>
</tr>
<tr>
<td></td>
<td>1.5</td>
<td>4.86e⁻⁰⁷</td>
<td>3.152e⁺⁰⁰</td>
<td>9.35e⁻⁰⁰</td>
<td>1.5000012</td>
<td>88308.172</td>
</tr>
</tbody>
</table>

### Table IV Cumulative results for helical pipe radiator

<table>
<thead>
<tr>
<th>Fluids</th>
<th>At Mass flow rate (kg/sec)</th>
<th>Pressure (pa)</th>
<th>Temperature (k)</th>
<th>Heat transfer coefficient (w/m²k)</th>
<th>Mass flow rate (kg/sec)</th>
<th>Heat transfer rate (w)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Water</td>
<td>2.8</td>
<td>1.20e⁻⁰⁷</td>
<td>3.1501e⁺⁰⁰</td>
<td>2.36e⁻⁰⁷</td>
<td>2.800002</td>
<td>168575.27</td>
</tr>
<tr>
<td></td>
<td>1.5</td>
<td>3.58e⁻⁰⁷</td>
<td>3.151e⁺⁰⁰</td>
<td>2.364e⁻⁰⁷</td>
<td>1.500015</td>
<td>81321.451</td>
</tr>
<tr>
<td>Air</td>
<td>2.8</td>
<td>7.09e⁻⁰⁷</td>
<td>3.1502e⁺⁰⁰</td>
<td>9.54e⁻⁰⁷</td>
<td>2.8000021</td>
<td>46067.237</td>
</tr>
<tr>
<td></td>
<td>1.5</td>
<td>2.06e⁻⁰⁷</td>
<td>3.1503e⁺⁰⁰</td>
<td>8.33e⁻⁰⁷</td>
<td>1.500011</td>
<td>24075.719</td>
</tr>
<tr>
<td>MgO₂(0.2)</td>
<td>2.8</td>
<td>8.17e⁻⁰⁷</td>
<td>3.1503e⁺⁰⁰</td>
<td>4.07e⁻⁰⁷</td>
<td>2.8000021</td>
<td>88251.705</td>
</tr>
<tr>
<td></td>
<td>1.5</td>
<td>2.39e⁻⁰⁷</td>
<td>3.150e⁺⁰⁰</td>
<td>4.071e⁻⁰⁷</td>
<td>1.50001</td>
<td>39209.043</td>
</tr>
<tr>
<td>MgO₂(0.4)</td>
<td>2.8</td>
<td>6.17e⁻⁰⁷</td>
<td>3.151e⁺⁰⁰</td>
<td>7.27e⁻⁰⁷</td>
<td>2.8000021</td>
<td>46923.734</td>
</tr>
<tr>
<td></td>
<td>1.5</td>
<td>1.74e⁻⁰⁷</td>
<td>3.1521e⁺⁰⁰</td>
<td>7.561e⁻⁰⁷</td>
<td>1.5000123</td>
<td>19895.16</td>
</tr>
<tr>
<td>Sic(0.2)</td>
<td>2.8</td>
<td>6.91e⁻⁰⁷</td>
<td>3.1522e⁺⁰⁰</td>
<td>4.92e⁻⁰⁷</td>
<td>1.500021</td>
<td>29224.613</td>
</tr>
<tr>
<td></td>
<td>1.5</td>
<td>2.03e⁻⁰⁷</td>
<td>3.1510e⁺⁰⁰</td>
<td>4.92e⁻⁰⁷</td>
<td>1.500021</td>
<td>29224.613</td>
</tr>
<tr>
<td>Sic(0.4)</td>
<td>2.8</td>
<td>4.85e⁻⁰⁷</td>
<td>3.1510e⁺⁰⁰</td>
<td>1.03e⁻⁰⁷</td>
<td>2.8000002</td>
<td>166734.98</td>
</tr>
<tr>
<td></td>
<td>1.5</td>
<td>1.37e⁻⁰⁷</td>
<td>3.151e⁺⁰⁰</td>
<td>1.031e⁻⁰⁷</td>
<td>1.500015</td>
<td>72788.58</td>
</tr>
</tbody>
</table>

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 MgO₂ 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 stand way a head in terms of heat transfer and nominal in pressure drop.

### REFERENCES