CFD ANALYSIS OF RADIATOR

C. Uma Maheswari¹, C. Suresh Reddy², C. Rajeshwara Reddy², A.T. Kiran Sai Naidu², C. Saradhi², A. Chenna kesavulu²

¹(Asst. Professor Mechanical Engineering, Sri Venkateswara College of Engineering and Technology, India)
²(Department of Mechanical Engineering, Sri Venkateswara College of Engineering and Technology, India)

Abstract: This paper deals with the thermal and CFD analysis of automobile radiator. The theoretical calculation has been done in MATLAB by varying the mass flow rate of coolant. Modeling has been done in Solidworks and exported to Ansys for CFD analysis. The temperature distribution, heat transfer rate for different velocities of coolant has been done for different tube materials such as copper, aluminium and stainless steel. The numerical results were compared and found that copper has best heat transfer rate and has better efficiency than the others.

1. Introduction

A lot of technology research work has been carried out on CFD Analysis of Automobile Radiator. In the year 2014, ChavanDK[1] had done the work on CFD Analysis of an air cooled radiator for diesel engine. In this work they had considered only one mass flow rate of coolant to find the temperature distribution of the coolant. M. Carl[2] at all had done on the theoretical and experimental investigation of the heat transfer process of an automobile radiator. C. Oliet [3] at different factors which influence radiator performance. It includes air and coolant flow, fin density and air inlet temperature. The main purpose of this work is done for various mass flow rate of coolant to find the temperature distribution.

2. Theoretical Analysis

The theoretical analysis is carried out with the help of MATLAB, in which the input parameters are mass flow rate of coolant, inlet temperature of air and coolant, properties of water at room temperature and dimensions and material properties of radiator. A suitable program is coded and the following results are obtained.

Table 2.1: Simulation results from MATLAB

<table>
<thead>
<tr>
<th>Mass flow rate of coolant in kg/s</th>
<th>Temperature drop of coolant in tube. °C</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Aluminium</td>
</tr>
<tr>
<td>0.23</td>
<td>27.6</td>
</tr>
<tr>
<td>0.33</td>
<td>23.2</td>
</tr>
<tr>
<td>0.43</td>
<td>20.5</td>
</tr>
<tr>
<td>0.53</td>
<td>16</td>
</tr>
<tr>
<td>0.63</td>
<td>13.2</td>
</tr>
</tbody>
</table>

3. Numerical Analysis

3.1 Modeling

The figure 1(a) shows the model of water domain in the Radiator. The water domain has been modelled in solid works as per the dimensions. Like the figure 1(b) shows the tube material domain. This one also modeled in solidworks as per the dimensions. After completion of the modeling the domain are save as in IGES file for further proceedings for CFD analysis in Ansys.
3.2 MESHING

ANSYS ICEM CFD meshing technologies provide physics preferences that help to automate the meshing process. For an initial design, a mesh (Tetrahedral) can often be generated in batch with an initial solution run to locate regions of interest. Further refinement can then be made to the mesh to improve the accuracy of the solution. There are physics preferences for structural, fluid, explicit simulations. By setting physics preferences, the software adapts to more logical defaults in the meshing.

Initially, IGES files of air domain, water domain and tube material domain of radiator has been imported to Ansys. the domain is meshed by using ICEMCFD with tetrahedral elements and the global element scale factor 1. In the mesh generation 1135922 nodes, 6015488 elements are generated. Near the wall of the radiator, the elements were created so as to capture the fine boundary layers. Patch dependent method is used for meshing. After application of smoothing the mesh quality of 0.3 was obtained.

4. Boundary Conditions

In this mesh files of created models for radiator and then exported to CFX-Pre. In the CFX-Pre physic and boundary equations are applied on the domain is explained to solve the continuity and momentum equation. The water domain and tube domain are created. The water domain is obtained for liquid water and tube domain is obtained for solid with material properties of aluminium, copper, and steel. Free slip boundary is gives to liquid domain and no slip boundary is given to the outer wall of the tube material. Inlet mass flow rate is given to the inlet boundary and outlet static presser is given. The heat transfer model of isothermal is selected.
5. Results and Discussions

5.1 MATLAB Results

The above figure 3 shows the overall heat transfer of coolant in the radiator for different mass flow rate of coolant for different tube materials. The coolant heat is transferred from coolant to the air as the temperature of coolant decreases and temperature of air increases.

5.2 CFD results

Computational Fluid Dynamic Simulations the performance of radiator depends upon the mass flow rate of coolant and air, temperature of coolant, thickness of tube, material use for tube and fin ect., which could be modified for improving the performance of radiator. Hence, computational fluid dynamic simulation approach is adopted to analyze the effect of different temperature distributions for coolant on the varying mass flow rate.

5.2.1 Velocity Distribution

The figure 4(a) shows velocity of coolant in the tubes at the mass flow rate 0.43 m/s and temperature 359.5 k. The velocity coolant is maximum at the inlet and goes on decreasing till the exit of the tube. The velocity of coolant at inlet is 0.7850 m/s.
5...2.2 Temperature Distributions

The figure 4 (a) and (b) shows the temperature distribution in the radiator at mass flow rate of coolant is 0.43 m/s and inlet temperature is 359.5 K. The temperature is maximum at the inlet and goes on decreasing till the outlet. The magnitude of temperature at the outlet is 347.5 K.

6. Conclusion

In this work, the theoretical calculation has been done by the MATLAB software by coding. The theoretical calculation of velocity of coolant, convective heat transfer of water and air, and the outlet temperature of water and air is found. The radiator domains like water domain and material domain has been created in the SolidWorks which are imported into ICEMCFD. In the ICEMCFD the domain are meshed with tetrahedral mesh. These mesh parts are imported into CFX 14.0. In CFX the interface and boundary conditions are applied. By varying the inlet mass flow rate of coolant the temperature distribution was analyzed. It has been founded that, when the coolant mass flow rate increases the outlet temperature of coolant increases.

The tube of different material is varied as copper, aluminium and steel for different mass flow rates of coolants. Finally the results shows that copper has the highest and aluminium is better than stainless steel. Due to corrosion resistant copper is best used for tube materials.

7. Nomenclature

Flow area=(\(\pi/4\)\(*\)\(D_i^2\)) m^2

Velocity of water=(\(ma/ (\rho_{wa}\,*\,V_{wa}\,*\,D_i)/\mu\)) m/s

Reynolds number for water=((\(\rho_{wa}\,*\,V_{wa}\,*\,D_i)/\mu\))

Nusselt number for water=0.023*((\(RE_{wa}\))^0.8)*((\(PR_{wa}\))^0.3)

convective heat transfer coefficient of water=(\(NU_{wa}\,*\,K_{wa}\))/\(D_i\) watt/m^2 K

Velocity of air=\(m_{air}/(2\,*\,\,V_{air}\,*\,(\pi/4)\,*\,D_f)\) m/s

Maximum velocity of air=\((S_f/(S_f\,-\,D_f))*\,V_{air}\) m/s

Reynolds number air=\((\rho_{air}\,*\,\,Max V_{air}\,*\,D_f)/\,\mu_{air}\)

Nusselt number for air=0.664*\((RE_{air}\)^0.5*\((PR_{air}\)^0.333

convective heat transfer coefficient of air=(\(NU_{air}\,*\,K_{air}\))/(\(D_o\)) watt/m^2 K

Corrected fin length \(L_c=\,\,L_f+(H_f/2)\) m

Coefficient for calculating efficiency \(m=(2\,*\,h_{air})/(K_{alu}\,*\,H_f)\)^0.5

efficiency of fin \(\eta=(\tan(h_{alu}\,L_c))/(m\,*\,L_c)\)

Surface area of fin \(A_f=2\,*\,W_f\,*\,L_c\) m^2
Base surface area $A_b = (2* L_r*W_t -(H_f*W_f*N_f)) \ m^2$
Total surface area of fin $A_{finbase} = (N_f*A_f) + A_b \ m^2$
Overall efficiency $\eta_{over} = 1 - ((N_f*A_f)/(A_{finbase})*(1-\eta_f))$
Total internal surface area $= 2*3.14*L_r*N_t \ m^2$
Overall heat transfer coefficient $= \frac{1}{((1/(\eta_{over} * h_{air} * A_{ex})) + (1/(h_{wa} * A_{in})))} \ \text{Watt/m2-K}$
Number of transfer units $= (UA/C_{min})$
effectiveness $= \exp((NTU/0.22)(C_r) \times (\exp(C_r \times NTU/0.78)-1))$
$q_{max} = C_{min} \times (T_{waterin} - T_{airin}) \ \text{watts}$
predicted heat transfer rate $= ($effectiveness$) \times q_{max} \ \text{watts}$
water outlet temperature $= T_{waterin} - (q_{predicted}/C_{max}) \ ^\circ\text{C}$
air outlet temperature $= T_{airin} + (q_{predicted}/C_{min}) \ ^\circ\text{C}$

References