Proceedings of the
International Conference on Mechanical Engineering 2005
(ICME2005) 28- 30 December 2005, Dhaka, Bangladesh

ICME05-FL-24

CFD ANALYSIS OF FLUID FLOW AND HEAT TRANSFER IN A SINGLE TUBE-FIN ARRANGEMENT OF AN AUTOMOTIVE RADIATOR

S.N. Sridhara[1*], S.R. Shankapal[2] and V. Umesh Babu[3]

[1] Center for Rotary Machinery Design,
[2, 3] SAS-Technosolutions,
MS Ramaiah School of Advanced Studies, New BEL Road, Bangalore-560 054, India.
* sns@msrsas.org , sridhara_sn@yahoo.co.uk

ABSTRACT

Power plants in the automobiles are becoming highly power-packed with increasing power to weight and/or volume ratio. Further, the space available under the bonnet is also decreasing due to the ever increasing demand of small cars by the customers. This has lead to the increased demand on the power packed radiators, which can dissipate maximum amount of heat for any given space. The flow behaviour and temperature profile prediction in the radiator tubes are very useful information and is of great importance to the designer. The geometry of the finned-tube heat exchanger is an intricate one and there are no analytical optimization schemes available to optimize their design, while experimental trial and error is far too time consuming. The radiator designs at present depend on the empirical methods, wherein existing experimental data is used as the thumb rules for the design process. However, for any preliminary design the performance of the radiator can be assessed through Computational Fluid Dynamics (CFD) in priori to the fabrication and testing. In the current study a tube fin arrangement of an existing radiator is analyzed for evaluating the fluid flow and heat transfer characteristics. The overall pressure, temperature and mass flow rate distribution of the coolant and air in and around the single tube-fin arrangement with 32 fins are evaluated. The fluid flow simulation is conducted using commercial software FLUENT 6.1. The pressure and temperature distribution along the tube length and tube width are presented and analysed. The results obtained serve as good database for the future investigations.

Keywords: CFD, radiator design, Fin-tube convective heat transfer.

1. INTRODUCTION

The thrust on automobile manufacturers for developing compact and energy efficient cars warrants a thorough optimization process in the design of all engine components, including radiators. Radiators are installed in automobiles to remove heat from the under hood which include engine cooling and heat removal during air-conditioning process. The use of higher output engines with tightly packed under hood packaging, the addition of new emission control components and the requirement of aerodynamic front end styling with narrower openings are decreasing the space available for circulation of under-hood cooling air. These conditions demand a better understanding of the complex cooling fluid flow characteristics and resulting thermal performance of the radiator.

About 30% of the thermal energy generated due to combustion of fuel is dissipated to the coolant that circulates in the engine-cooling jacket. The hot coolant coming out of engine cooling jacket is to be cooled in a radiator and circulated again in the jacket. In an automobile, energy dissipated from the engine through radiator is not utilized but lost to the atmosphere. The engine performance, safety and engine life depends on effective engine cooling.

CFD Analysis of the radiator as a whole is seldom reported in the literature. However, the technical information about fin-tube heat exchangers are available in bits and pieces as individual analysis carried out experimentally and/or numerically.

Hilde Van Der Vyer et al. [1] conducted a CFD simulation of a three dimensional tube-in-tube heat exchanger using Star-CD CFD software and made a validation test with the experimental work. The authors were fairly successful to simulate the heat transfer characteristics of the tube-in-tube heat exchanger. Though this work is not directly related to our study, this has been used as the base for the procedures of CFD code validation of a heat exchanger.

Witry et. al., [2] carried out CFD analysis of fluid flow and heat transfer in patterned roll bonded aluminium radiator, in which FLUENT’s segregated implicit 3-D steady solver with incompressible heat transfer is used as the tool. In this study, shell side airflow pattern and tube side water flow pattern are studied. The authors presented the variation of overall heat transfer coefficients across the radiator ranging from 75 to 560 W/m^2-K. This study established the capability of FLUENT code to handle such problems.

Chen et al., [3] made an experimental investigation of the heat transfer characteristics of a tube-and-fin radiator for vehicles using an experimental optimization design technique on a wind tunnel test rig of the radiator. The authors have developed the regression equations of heat dissipation rate, coolant pressure drop and air pressure drop. The influences of the air velocity, inlet coolant temperature and volume flow rate of coolant on heat dissipation rate, coolant pressure drop and air pressure drop have been discussed in detail by means of the numerical analyses. The results published in this paper provide a basis for the theoretical analysis of heat performances and structural refinement of the tube-and-fin radiator.

Coupling of CFD and shape optimization for radiator design, a paper by Sridhar Maddipatla, [4] forms the benchmark for the present work. This paper presents a method to design automobile radiator by coupling CFD with a shape optimization algorithm on a simplified 2D model. It includes automated mesh generation using Gambit, CFD analysis using Fluent and an in-house C-code implementing a numerical shape optimization algorithm are discussed. All of the flow simulations reported in this paper were performed using the classical simple algorithm with a k- ε turbulent model and second order upwind scheme. Despite the developments reported in the literature a clear gap of basic understanding of the adoption of CFD procedure to the analysis of fluid flow and heat transfer mechanisms in a complex fin-tube radiator is unattended. The present work is an attempt to fill this gap and get an insight on this complex phenomena. The study is aimed to analyze the fluid flow characteristics in a commercially existing radiator and understand the flow phenomenon to establish better design. It involves calculating the overall pressure drop and mass flow rate distribution of the coolant and air in and around the single tube arrangement of an automotive radiator. The fluid flow simulation is conducted using commercial software FLUENT.

2. NUMERICAL EXPERIMENTATION

The Numerical experimentation of the work carried out involved reverse engineering of a commercial automotive radiator for the required fluid domain, discretising the fluid domain, simulation of the fluid flow and heat transfer at steady state and post processing the results and drawing suitable conclusions.

The radiator of a commercially existing vehicle is chosen for the analysis to bring in the practicality to the study. The details of the geometry of the radiator were obtained by the process of reverse engineering. The dimensions of individual components of the radiator were measured using suitable measuring instruments (see Table 1). The measurements obtained were used to generate the CAD model in CATIA V5 R9 (see Fig. 1).

Table 1: Specifications of the radiator chosen

Radiator type	Cross flow, single row core, forced air cooled radiator
In hose barb (inlet)	6.81mm
Out hose barb (outlet)	6.81mm
Core rows	35
Core tube	1.84mm
Fin density	1.5mm/fin
Core dimensions (L x H x W)	510mm x 395mm x 40mm
Tube and fin material	Aluminium
End tube material	Nylon 6,6

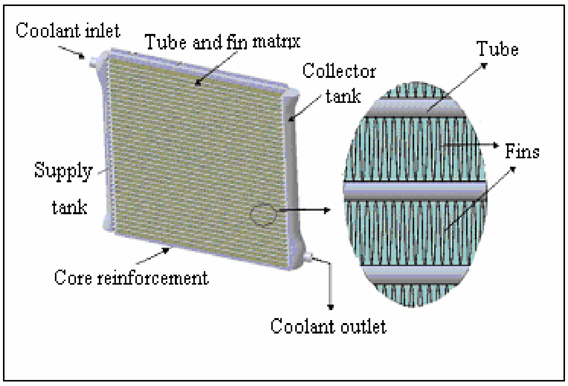

Fig 1. Assembled CATIA model of the radiator

2.1 Discretisation of the fluid domain

The geometric similarity between the rows of tube and fin helps us in limiting the computational domain to a single tube and adjoining fin arrangement as shown in Fig. 2. Hence, the fluid domain is created for a single fin and tube assembly and numerical analysis is carried out.

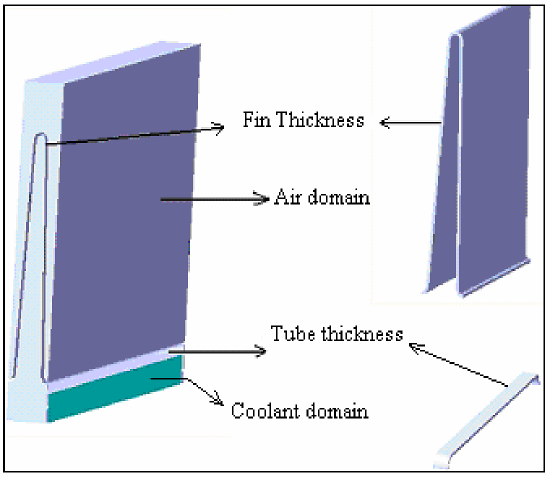

Fig 2. Fluid domain of the single tube fin arrangement

The fluid domain includes the airflow volume and the coolant flow volume. The problem is solved as a conjugate heat transfer requiring the thickness of the tube and fin also to be modeled.

The surfaces in the air domain, coolant domain, tube thickness and fin thickness are discritised with varying mesh density in accordance to the physics of fluid flow and heat transfer. Denser mesh size is used at critical volumes in the fluid flow and heat transfer domain. The grid independence study of the simulation is carried out to arrive at the minimum number of elements required to maintain the required stability and accuracy in the computation. The finalised element count and the related aspects are listed in Table 2. The meshed geometry for a single tube and fin is then extended to the entire 32 fin assembly of the radiator as shown in Fig. 3.

Table 2: Element count in tube-fin assembly with 32 fins

Number of hexahedral elements	469568
Number of prismatic elements	10304
Number of 2D elements	168320
Total Number of elements	648192
Aspect ratio	2%
Skewness	1%

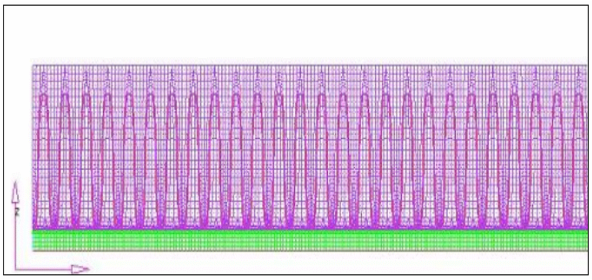

Fig 3. Isometric view of the completely meshed tube-fin assembly with 32 fins

2.2 CFD Simulation for fluid flow and conjugate heat transfer

The physics of conjugate heat transfer in radiator is simplified with the following technically valid assumptions.

- Velocity and temperature at the entrance of the radiator core for air and coolant is uniform.
- No phase change occurs in fluid streams.
- Fluid flow rate is uniformly distributed through the core in each pass on each fluid side. No flow leakages occur in any stream. The flow condition is characterized by the bulk speed at any cross section.
- The thermal conductivity of the solid material is constant.
- No internal source exists for thermal-energy generation
- Properties of the fluids and the wall, such as specific heat, thermal conductivity, and density are only dependent on temperature.

The input data and boundary conditions are chosen from the study of Changhua Lin and Jeffrey Saunders [5], in which the effect of varied air and coolant inlet temperatures on the specific dissipation is discussed. The air inlet velocity (V_{ai}) is chosen to be 4.4m/s air at ambient temperature [4,5]. With reference to the specifications of the coolant pump used for the vehicle chosen, coolant inlet velocity (V_{ci}) is 0.0063 kg/s in each tube. The properties of air and coolant were defined for standard conditions and kept constant throughout the analysis.

The solver chosen for the analysis is segregated, implicit, 3D, steady state solver in FLUENT 6.1. The segregated solver has been used for incompressible and mildly compressible flows by many investigators [3,4] and has been shown to depict the results with better accuracy. It solves the energy and flow equations sequentially. The continuity, momentum and energy equations (equations not shown in this paper for their generality) of fluid flow are solved in the process of obtaining temperature profiles. The segregated approach solves for a single variable field (e.g., pressure, p) by considering all cells at the same time. It then solves for the next variable field by again considering all cells at the same time, and so on [6]. This approach is well documented in literature and commonly adopted by the scientists in complex problems such as conjugate heat transfer dealt in this paper. Standard k-ε model is chosen to account for turbulent flow. Robustness, economy, and reasonable accuracy for a wide range of turbulent flows and heat transfer simulations are the reasons for choosing this model.

3. RESULTS AND DISCUSSION

The simulation results obtained show reasonable variation in the temperature and pressure as expected (see Figs. 4, 5, 6 and 7). A drop in temperature of the coolant from 371.6 K to 365.6 K and an increase in the temperature of the air from 301.1 K to 310.6 K is observed in these plots. The effectiveness of the heat exchanger is computed to be 80%. The drop in the pressure is 52.3Pa. The energy transfer/unit weight in the coolant domain is 303.6449 kJ/kg and of air domain is 37.1 kJ/kg indicating the compactness of the heat exchanger.

An initial slackness is observed in the coolant temperature profile (Fig. 4) and pressure profile (Fig. 6) indicating the ineffectiveness of the fins at the entry to the radiator. This is an indirect indication of slow thermal response of the radiator at the beginning stages of the fin. This suggests the need for a deeper study in this area in the direction to improve the initial thermal slackness.

The air side temperature (Fig. 4) and pressure show a steady variation across the length of the tube (Fig. 7). This trend is expected in any usual radiator performance. The pressure contours of air and coolant along their directions of flow (Fig. 8), that of the air along the direction of airflow (Fig. 9) and temperature contours of coolant and air along their directions of flow (Fig. 10) add to the data base knowledge towards a better design of compact radiator.

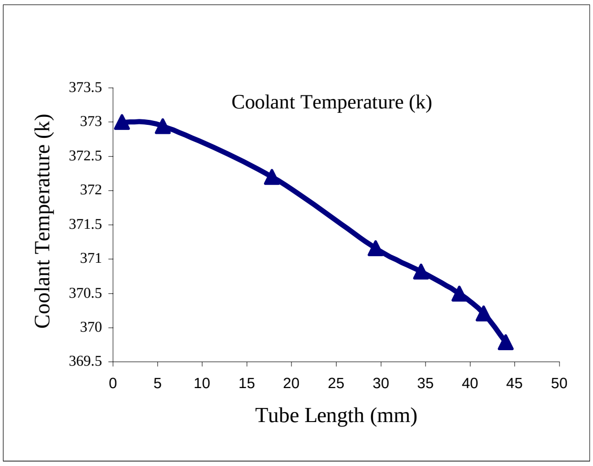

Fig 4. The variation of coolant temperature (K) across the tube length

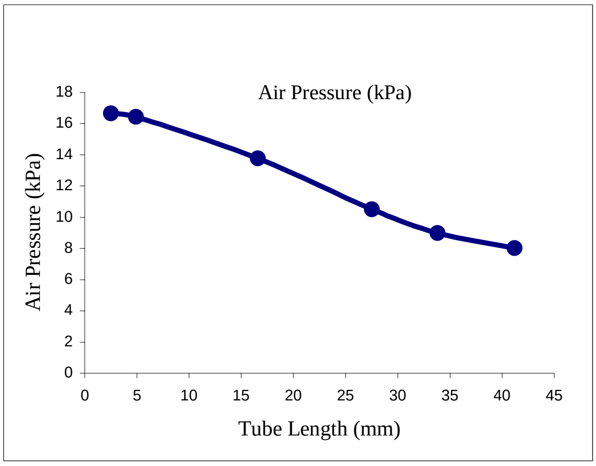

Fig 7. The variation of total pressure (kPa) on airside across the tube length

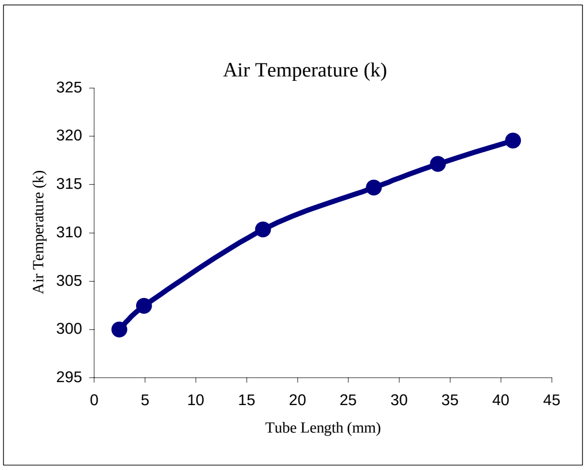

Fig 5. The variation of air temperature (K) across the tube length

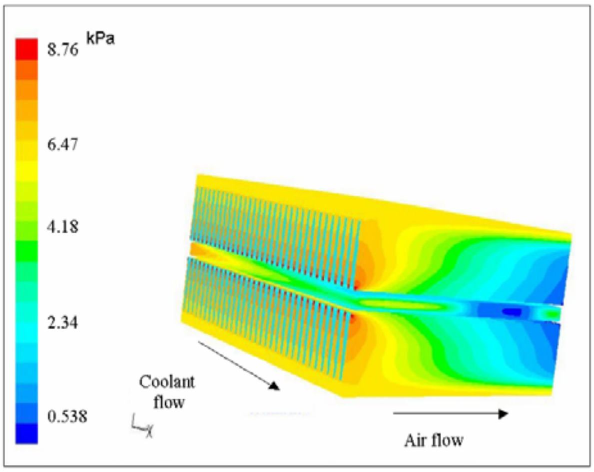

Fig 8. Total Pressure Contours (kPa) of air and coolant along their directions of flow

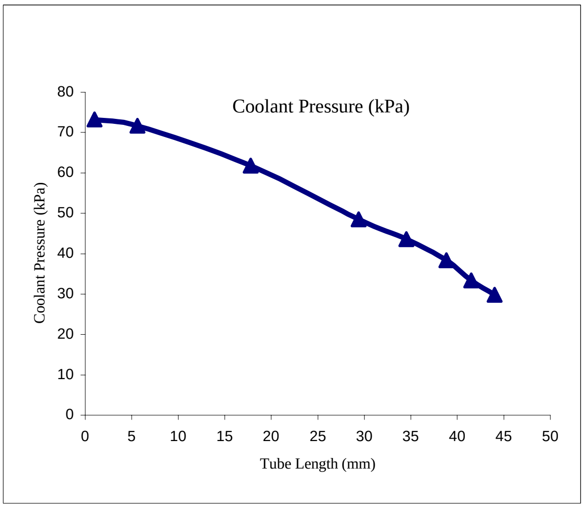

Fig 6. The variation of total pressure (kPa) on coolant side across the tube length.

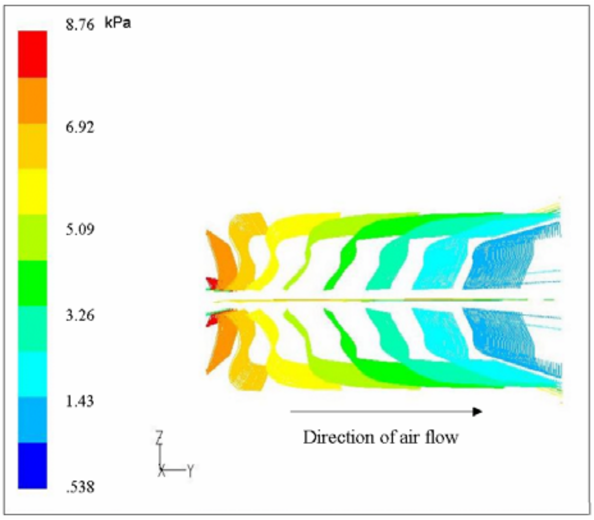

Fig 9. Pressure contours (kPa) of the air along the direction of airflow

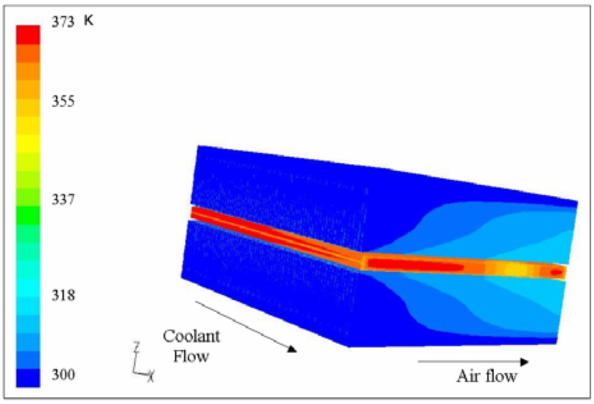

Fig 10. Temperature (K) contours of coolant and air along their directions of flow

The performance of the radiator is usually expressed in terms of effectiveness and compactness. Effectiveness is the measure of heat transfer rate of the system, given by ε = (actual heat transfer)/(maximum heat transfer possible),

$$\varepsilon = \frac{m_c c_{pc}(T_{ci} - T_{co})}{m_a c_{pa}(T_{ci} - T_{ao})} \qquad (1)$$

The effectiveness of the system is found to be 81% from the computational analysis, which reflects the moderate performance of the radiator as a heat exchanger.

Compactness of the system is the energy transfer that takes place per unit weight. From the analysis, the values obtained for compactness of the coolant and air is, $C_{COOLANT}$ = 303.6kJ/kg and C_{AIR} = 37 kJ/kg. Though these figures are encouraging, a continued work on the size optimization is suggested for this radiator.

4. CONCLUSIONS

The fluid flow and heat transfer analysis of a single tube-fin arrangement of an automotive radiator is successfully carried out using numerical simulation built in commercial software FLUENT. The variations in the pressure, temperature and velocity in the direction of coolant flow and airflow are presented and analysed. It is observed that the temperature of coolant drops by 6 K and a pressure drop of 52.3 Pa in the coolant. The air that absorbs the heat due to forced convection gains an in increase in temperature by 9.5 K.

The study forms a foundation for the fluid flow analysis of an automotive radiator. With the computational time and resources available, the results obtained were found to be satisfactory. However, a continued study in various aspects towards a better design of the radiator is suggested as shown below.

- To account for the variation of the inlet conditions with time as in practical cases, transient analysis can be done.
- Optimizing the values of the flow rates and the dimensions of the radiator for a given power rating of the vehicle, by generating CFD codes.

5. REFERENCES

1. Hilde Van Der Vyer, Jaco Dirker and Jousoa P Meyer, 2003, "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.
2. A.Witry M.H. Al-Hajeri and Ali A. Bondac, 2003, "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.
3. 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 of Institution of Mechanical Engineers*, Republic of China, 215: pp. 2-8.
4. Sridhar Maddipatla, 2001, "Coupling of CFD and shape optimization for radiator design", Oakland University. Ph.D. thesis.
5. Changhua Lin and Jeffrey Saunders, 2000, "The Effect of Changes in Ambient and Coolant Radiator Inlet Temperatures and Coolant Flowrate on Specific Dissipation", SAE Technical Papers, 2000-01-0579.
6. Fluent company, 2000, *FLUENT Help manual.*
7. J.P.Holman, 2002, *Heat transfer*, Tata-McGraw-Hill Publications.

6. NOMENCLATURE

Symbol	Meaning	Units
T_{ci}, T_{co}	Coolant inlet and outlet temperatures	(K)
T_{ai}, T_{ao}	Air inlet and outlet temperatures	(K)
L	Length of the radiator	(mm)
H	Height of the radiator	(mm)
W	Width of the radiator	(mm)
V_a	Velocity of air flow	(m/s)
V_c	Velocity of coolant flow	(m/s)
P	Pressure of the fluids	(Pa)
m_c, m_a	Mass flow rate of coolant	(m^3/s)
c_{pc}, c_{pa}	Specific heat of the coolant	(KJ/Kg K)
ρ_a	Density of air	(kg/m^3)
a_a	Area of air flow	m^2
E	Effectiveness of the radiator	(-)