International Journal of Mechanical and Production Engineering Research and Development (IJMPERD)
ISSN(P): 2249-6890; ISSN(E): 2249-8001
Vol. 8, Special Issue 8, Aug 2018, 26-32

EFFECT OF FIN GEOMETRY FOR VARIOUS TUBES SHAPE USING CFD SIMULATION ON MULTI-CHANNEL HEAT EXCHANGER IN MOBILE AIR CONDITIONING (MAC)

V. ASHOK KUMAR & N. SATHISHKUMAR

Department of Mechanical Engineering, St. Joseph's college of engineering, Chennai, Tamil Nadu, India

ABSTRACT

This study deals with air-side Heat Transfer and Pressure drop characteristics of Louvered fin automotive condenser for different geometrical parameters of louvers, including louver pitch, louver angle and louver length were investigated using Computational Fluid Dynamics (CFD) simulation. The 3-dimension (3-D) CFD model was established to predict air-side flow and heat transfer characteristics. The comparison between CFD simulations and theoretical data were established. After finding out the optimal geometrical parameters, three different tube profiles (Hexagonal, Oval, and Rectangular) were also investigated using 3-D CFD model.

KEYWORDS: CFD Simulation & Automotive Air-Conditioning Condenser

Original Article

INTRODUCTION

Recently, the compact heat exchanger has been paid more and more attention by HVAC&R (heating, ventilating, air-conditioning, and refrigeration) industries and used in some commercial and residential air conditioning system. In compact heat exchangers, heat-resistance is majorly considered on air-side and account for 80% or more of the total thermal resistance. The heat transfer surface area is 10 times larger in air-side compare with refrigerant-side. Enhancement of heat transfer on air-side increases the overall performance of the heat exchanger. Because of this high thermal resistance on the air-side, the optimization of such fins is predominant to enhance the performance of the heat exchangers in thermal systems. Fin louvers geometry optimization is essential to increase the heat transfer performance and reduce weight, packaging, and cost requirements. Many studies on louver fin have been reported Davenport [1] through flow visualization study with smoke traces or dye injection techniques, showed that the air flow had two directions depended on Reynolds number based on the louver pitch and maximum air velocity. The air did not pass through louvers called duct directed flow at low Reynolds numbers. Achaichia and Cowell [2] further confirmed the phenomena. Based on their researches, some air-side heat transfer coefficient and pressure drop correlations for louver fin have been proposed based on a huge database[3], also including some results of their own experimental studies on heat transfer characteristics of different louver fin geometries by Chang and Wang [4] and Kim and Bullard [6].

The objective of this study is to choose an optimal geometrical parameter combination of louvers and tube geometry, required for the current work, which gives maximum heat transfer coefficient using CFD simulations.

EXPERIMENTATION

The experiments were carried out in Air-conditioning and Heat Exchangers Laboratory directly under our incorporation, consisting of an unattached Air, water, and refrigerant Enthalpy Method Calorimeter. The Automotive air-conditioning condenser is installed in the inlet of the wind tunnel with surrounded insulation to protect it from heat loss and air leakage. Tests are performed for face air velocity from 2 to 8 m/s, corresponding to the normal operating range of conditions in automotive air-conditioning. The air inlet temperature is 35˚ for all tests. On the oval tube side, a refrigerant inlet is given as 83˚C temperature. The refrigerant mass flow rate is maintained at 54g/s. The air-side thermal performance data are determined using the effectiveness-NTU method for cross-flow heat exchangers with both fluids unmixed. Measurement of air-side pressure drops directly measured with the help of static pressure device. Experimental results for test sample are given below:

Table 1: Geometrical Data of Test Sample

Tube and fin depth	16 mm
Tube height	1.8 mm
Fin height	8.05 mm
Louver angle	30˚
Louver pitch	1.5 mm
Louver length	6.2 mm

Table 2: Experimental Results

Face Velocity (m/s)	Heat Transfer Co-Efficient (W/m^2K)	Pressure Drop (Pa)	Overall HTCW/m^2 K
2	60	25	148
5	102	150	221
8	135	410	277

GEOMETRY AND BOUNDARY CONDITIONS

The 3D computational domain as shown in Figure 1 is the representation of one convolution i.e., It contains two fins of the actual air-side surface for different automotive heat exchangers like radiators, condensers. Two inlet and outlet rectangular channels are added to establish the flow before the entrance in the fins and to avoid backflow downstream of the convolution. Symmetry conditions are assumed on both sides of the domain. A tetrahedral mesh is used and the total number of cells for the 3D model is 6 million (without tube) and 8 million (with tube) as shown in Figure 2. Tube and fins have been meshed to take into account the material thermal resistance. Air velocity is considered as inlet condition for simulation. For fin, non-slip boundary conditions were applied, as well as constant temperature boundary condition.

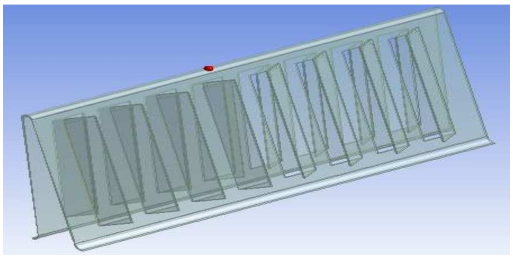


Figure 1: 3D Geometrical Model of Louvered Fin

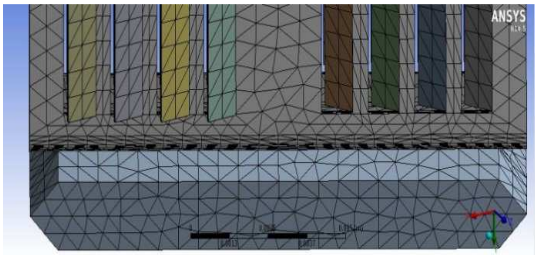
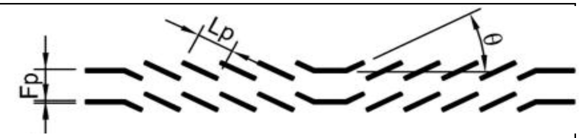


Figure 2: Mesh Connectivity Diagram

A flow of air on the louver side is assumed as laminar and steady. The Model is governed by the conservation of mass, momentum and energy equation. The fin wall temperature is 80˚C, air velocity is varied from 2-8 m/s and air inlet temperature is 35˚C. This is the common condition of an automotive air-conditioning condenser.

Table 2: Geometrical Parameters and Operating Conditions of the Louvered Fin

Fin Pitch (mm)	1.2 mm	
Fin depth(mm)	46 mm	
Fin thickness(mm)	0.08 mm	
Louver Pitch (mm)	1.3/1.4 /1.5 /1.6 /1.7	**Where, Fp – Fin Pitch, Lp – Louver Pitch,**
Louver angle (deg)	24/ 27/30	**θ- Louver Angle**

CFD SIMULATION ON FIN SIDE

The performance of automotive air-conditioning condenser is highly affected by the louver fin geometry. Here the analysis mainly evaluates the influence of louver angle, louver pitch, and louver length.

Figure 3 shows the louver fin pictorial results from the CFD simulation, temperature counter diagram illustrates the temperature increment of the fluid when it has a contact with the louvers. As well as pressure counter diagram explains the pressure difference throughout the fin. The Vector distribution of the fluid is also illustrated in the picture. Due to the turbulence created near the louvers in the fluid domain is the major cause of pressure drop.

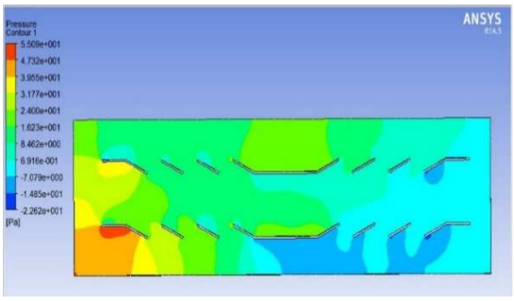


Figure 3(a): CFD Results of Pressure Counter

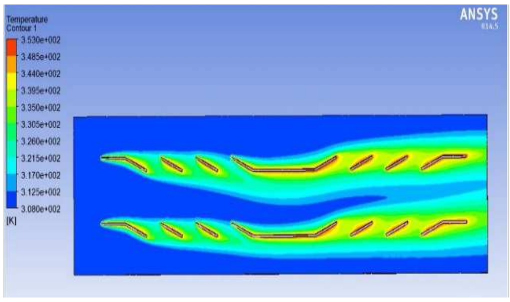


Figure 3(b): CFD Results of Temperature Counter

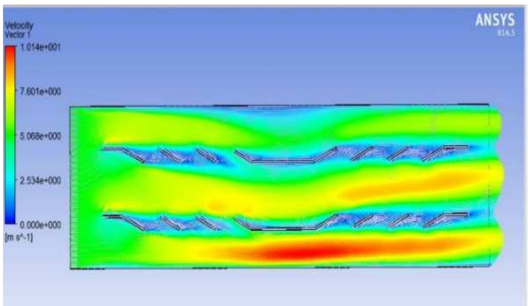


Figure 3(c): CFD Results of Velocity Vector

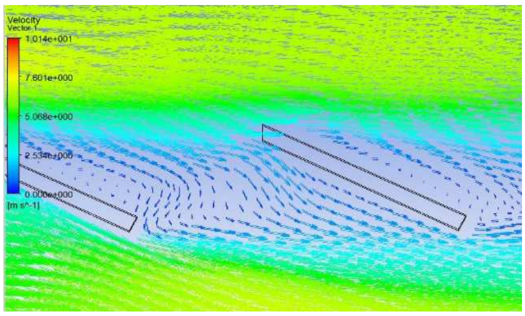


Figure 3(d): Magnified View of Velocity Vector

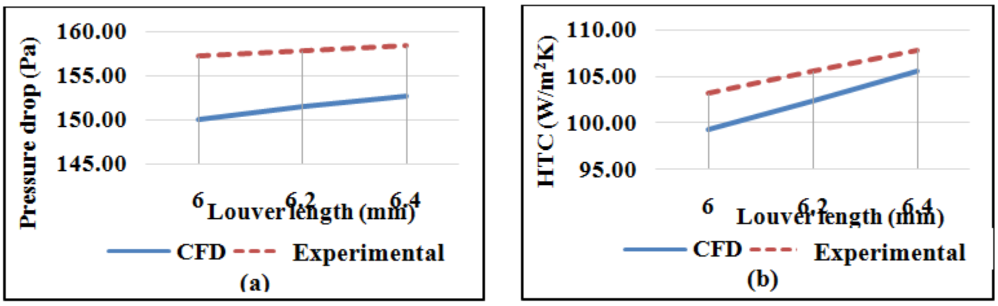


Figure 4: Comparison of CFD and Experimental Values: (a) Pressure Drop (b) Heat Transfer Coefficient

Figure 4 depicts the effect of louver length on the heat transfer and pressure drop characteristics for Lp=1.5mm and ϴ=27 deg. The heat transfer coefficient increase with louver length and the pressure drop is nearly linearly rising. An increase of heat transfer coefficient is not that much noticeable when louver length increases, so there is no point in increasing the louver length. Therefore the optimal value of louver length is fixed at 6.2 mm.

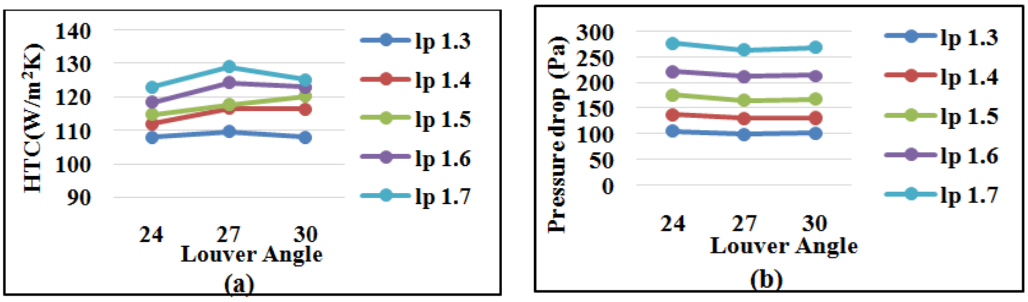


Figure 5: Variation of (a) Heat Transfer Co-Efficient and (b) Pressure Drop

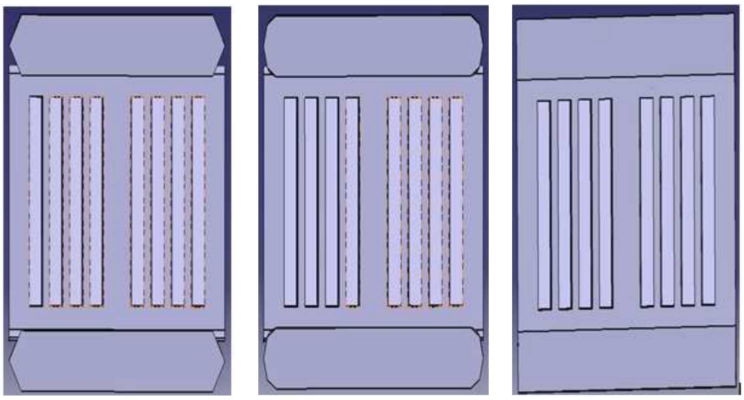
Figure 5, illustrates the variations of heat transfer coefficient and pressure drop with louver pitch and louver angle for Fp=1.2 mm. The heat transfer coefficient increases with increasing louver pitch. The heat transfer coefficient of large louver angle is higher when small louver pitch is preferred. When louver pitch reaches 1.5mm, the heat transfer coefficient of large louver angle is higher. The pressure drop increases with increasing louver pitch and decreases with increasing louver angle. Hence, when heat transfer rate meets the actual requirement, the combination of 1.5 mm louver pitch and 27° louver angle is a better choice**.**

CFD SIMULATION OF FIN WITH TUBE

To predict the overall performance of the condenser, fin with tube model is also to be analyzed. Three different tube geometry models (i.e., Hexagonal, Oval, and Rectangular) were investigated. To reduce the simulation complexity, only one convolution of the fin and a small part of the tube is taken for the simulation work.

Table 2: Geometrical and Operating Parameters of the Louvered Fin

Tube depth(mm)	16
Tube height (mm)	1.8
Fin Pitch(mm)	1.2
Louver Pitch(mm)	1.5
Louver angle(deg)	27
Louver length(mm)	6.2
Air velocity (m/s)	2,5,8



Hexagonal Tube Oval Tube Profile Rectangular Tube

Figure 6: Geometrical Models of 3 different Tube with Fin Profile

RESULTS AND DISCUSSIONS

Here the analysis mainly evaluates the influence of tube profiles as well as louver geometry in the automotive air conditioning condenser using CFD simulation on the flow pressure drop, heat transfer and Overall heat transfer coefficient of the condenser. Figure 7 shows the CFD simulation velocity vector flow of the fluid over the tube and the fin of the condenser.

Figure 8 shows the variations of Overall Heat transfer coefficient with air velocity and comparison of CFD results with the experimental values, for three different tube profiles. Here the Overall Heat transfer coefficient has a deviation of only around 10% with the experimental results, which reveals the good reliability of CFD geometry profile.

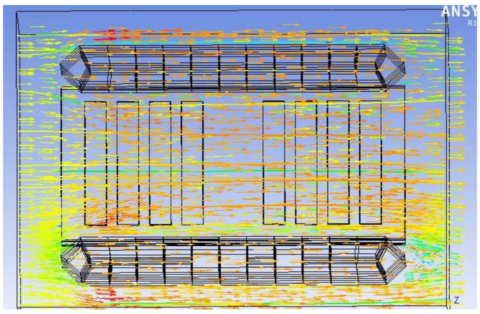


Figure 7(a): Velocity Vector of Air

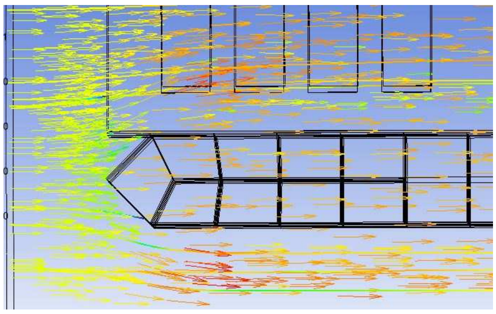


Figure 7(b): Magnified View of Vector Counter

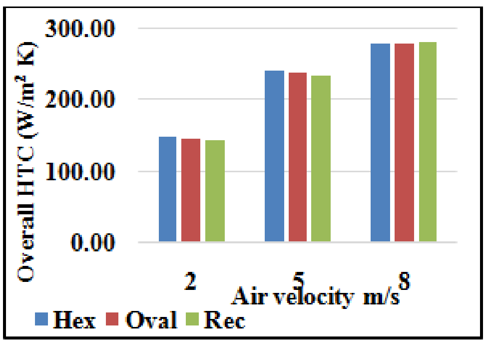


Figure 8(a): Variations of Overall Heat Transfer Coefficient with Air Velocity

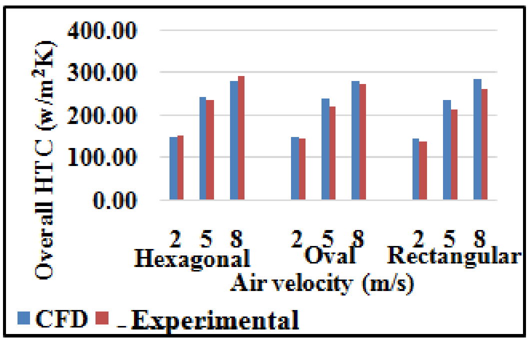


Figure 8(b): Comparison of Overall Heat Transfer Coefficient with CFD and Experimental Results

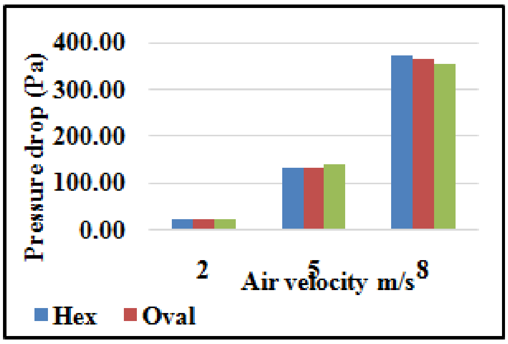


Figure 9(a): Variations of Pressure Drop (CFD Results) with Air Velocity for 3 Tube Profile

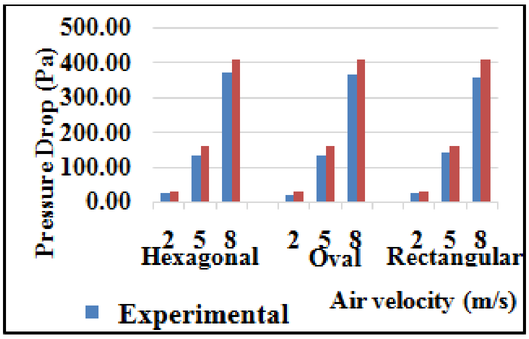


Figure 9(b): Comparison of Pressure Drop with CFD Results and Experimental Results

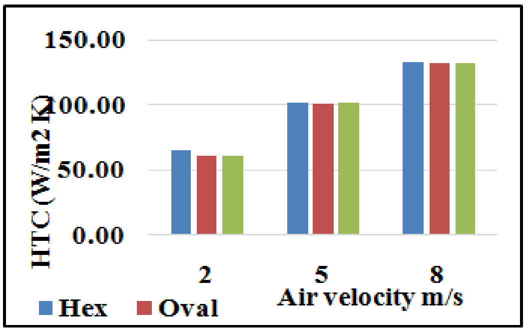


Figure 10(a): Variations of Heat Transfer Co-Efficient (CFD Results) with Air Velocity for 3 Tube Profile

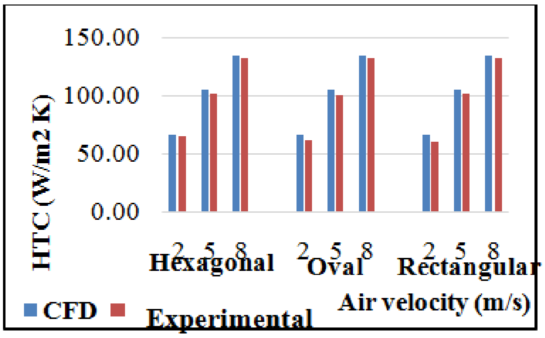


Figure 10(b): Comparison of Heat Transfer Co-Efficient with CFD Results and Experimental Results

It is shown in figure 9 and 10, the heat transfer coefficient increases with air velocity and the pressure drop is also linearly increase with air velocity. The Air velocity influences the pressure drop and heat transfer coefficient significantly. Hence the heat transfer rate meets its actual requirement when the hexagonal tube profile is considered.

CONCLUSIONS

The air-side heat transfer and pressure drop characteristics for Automotive Air-conditioning condenser with three different tube profile with louver fin were studied. The 3-D CFD simulation model was established and validated to predict air-side flow, heat transfer and overall heat transfer characteristics. Several important geometry configuration parameters of condenser louver fin, involving louver angle, louver pitch, louver length and three different tube profile have been analyzed with CFD simulation method under commonly used Automotive air conditioning condenser conditions. Major conclusions are summarized as follows:

When louver pitch reaches 1.5mm, the heat transfer coefficient of large louver angle is higher. The pressure drop increases with increasing louver pitch and decreases with increasing louver angle. Therefore, the heat transfer rate meets the actual requirement, when the combination of 1.5 mm louver pitch and 27˚ louver angle is preferred. With this fixed Louver parameter, the three different tube profile has been analyzed in CFD simulation for different air velocity. The results for this tube and fin simulation is summarized below.

The Air velocity influences the pressure drop and heat transfer coefficient significantly. Hence the heat transfer rate meets its actual requirement when the hexagonal tube profile is considered. In addition, the higher Overall Heat transfer coefficient for hexagonal tube profile also recommended for the optimal working condition. From the present study, the Hexagonal tube and (1.5 mm Louver pitch, 27 louver angle, 6.2 mm louver length) louver fin profile is the best optimal condenser geometry profile for the different air velocities on the basis of Overall HTC, Pressure drop and Heat transfer Co-efficient of the Automotive Air-conditioner Condenser.

REFERENCES

1. *Davenport C. J., Heat Transfer and Fluid Flow in Louvered Triangular Ducts: PhD thesis. CNAA, Lanchester Polytechnic, 1980.*
2. *Achaichia A., Cowell T. A., (1988), Heat Transfer and Pressure Drop Characteristics of Flat Tube and Louvered Plate Fin Surfaces. Experimental Thermal and Fluid Science, vol. 1: pp. 147-157.*
3. *Atkinson K. N., Drakulin R., Heikal M. R., Cowell T. A., (1998), Two- and Three-Dimensional Numerical Models of Flow and Heat Transfer over Louver Fin Arrays in Compact Heat Exchangers, International Journal of Heat Mass Transfer, vol.41: pp. 4063-4080*
4. *Tafti D.K., Wang G., Lin. W., (2000), Flow transition in a multi louvered fin array. International Journal of Heat and Mass Transfer, vol. 43, pp.901-919.*
5. *Chang Y. J., Wang C. C., (1997), A Generalized Heat Transfer Correlation for Louver Fin Geometry, International Journal of Heat Mass Transfer, vol. 40: pp. 533-544.*
6. *Kim M. H., Bullard C. W., (2002), Air-side Thermal Hydraulic Performance of Multi-louvered Fin Aluminum Heat Exchangers, International Journal of Refrigeration, vol. 25: pp. 390-400.*
7. *Hsieh C. T., Jang J. Y., 3-D Thermal-hydraulic Analysis for Louver Fin Heat Exchangers with Variable Louver Angle in Applied thermal engineering, vol. 26, 2006 pp. 1629-1639.*
8. *Saleem, A.; Kim, M.-H. CFD Analysis on the Air-Side Thermal-Hydraulic Performance of Multi-Louvered Fin Heat Exchangers at Low Reynolds Numbers. Energies 2017,* 10*, 823*