Sabari et al. 2017. *Int. J. Vehicle Structures & Systems*, 9(3), 145-148
ISSN: 0975-3060 (Print), 0975-3540 (Online)
doi: 10.4273/ijvss.9.3.03


**International Journal of Vehicle Structures & Systems**
Available online at www.maftree.org/eja

# CFD Simulation of Fin-and-Tube Heat Exchanger with Louvered Fin Configuration: Technical Note

**M. Sabari[a], Channankaiah[b] and D. Shivalingappa[c]**

*Dept. of Mech. Engg., Adhiyamaan College of Engg., Hosur, India*
*[a]Corresponding Author, Email: sabarimec93@gmail.com*
*[b]Email: hod_mech@adhiyamaan.ac.in*
*[c]Email: dsivadvg@gmail.com*

**ABSTRACT:**

*Heat exchanger plays a major role in almost all mechanical industries. Enhancement of heat transfer surface plays major role in numerous applications such as in heat exchangers, refrigeration and air conditioning systems etc. This paper examines the fluid flow and heat exchange on the air side of a multi-row fin-and-tube heat exchanger. A brief comparison is given between fin-and-tube heat exchanger attributes with louvered fins in a wider range of operating conditions defined by inlet air velocities. The brief representation on the calculated data for the louvered heat exchanger shows better heat transfer characteristics with a slightly higher pressure drop. The CFD procedure is validated by comparing the numerical simulation results with different inlet air velocities. Best combination of higher heat transfer and minimum pressure drop are occurred in inlet air velocity of 2.5 m/s.*

**KEYWORDS:**

*Fin-and-tube heat exchanger; Louvered fin; Heat transfer analysis; Computational fluid dynamics*

**CITATION:**

M. Sabari, Channankiah and D. Shivalingappa. 2017. CFD Simulation of Fin-and-Tube Heat Exchanger with Louvered Fin Configuration: Technical Note, *Int. J. Vehicle Structures & Systems*, 9(3), 145-148. doi:10.4273/ijvss.9.3.03.

**ACRONYMS AND NOMENCLATURE:**

| | |
|---|---|
| D | Tube outside diameter (mm) |
| $\mu$ | Coefficient of friction |
| H | Fin pitch (mm) |
| $P_l$ ,$P_t$ | Longitudinal and transverse tube pitch (mm) |
| t | Fin thickness (mm) |
| $T_{a,i}$ | Air inlet temperature (K) |
| $T_w$ | Wall surface temperature (K) |

## 1. Introduction

Fin-and-tube heat exchangers are being widely used in water chillers, heat pumps and heating, ventilating, air conditioning and refrigeration (HVAC&R). The usage of fin-and-tube heat exchangers in water chillers and heat pumps is increasing. The production of heat pumps and chillers are having constant growth since the last few years, and considering these devices energy efficiency is of paramount importance for achieving low energy consumption and sustainability. This large amount of production justifies that even the minimum improvement in heat exchanger. Although many automotive companies and heat exchanger manufacturers have performed a lot of experimental research in fin heat exchangers, very little of the experimental data is publicly available owing to its viable value. However some experimental data for louvered fins was reported by Yan et al [1] and Dong et al [2]. Expensive and long-term experimental research allows analysis of only a restricted number of geometrical shapes. In addition to the experimental study with the becoming larger processing power of modern computers, numerical fluid flow simulations have recently used in order to improve airside performance of fin-and-tube heat exchangers as described by Malapure et al [3] and Huihan et al [4] and also by using CFD as described by Carija et al [5].

Numerous papers report an improvement in air-side performance by varying the fin geometry. One of the very popular enhancements is the louvered fin surface. Louvered fin surfaces can provide higher average heat transfer coefficients due to renewal of the boundary layer improvement. The most common interrupted surfaces are louvered fins, which are analysed in this paper. In fin-and-tube heat exchangers, the fin shape has a main part in the heat exchange. The fin is subject to alteration with such enriched features as vortex generators, micro-grooved fins, or the use of other fin shapes for increasing heat transfer as reported by a number of authors including Tiwari et al [6], Wua et al [7], and Pesteei et al [8]. Design alteration and improvement with the technique of stiffener pattern repeatition explored with design of experiments and found to be highly efficient technique [9]. CFD is a powerful tool for calculating performance and succeeding the optimal design of fin-and-tube heat exchangers for various operating conditions as described by Lyman et al [10], Xia and Jacobi [11], Taler and Oclan [13]. Technique of material variations been used in stiffness and crash analysis achieving increased metrics can also be validated for CFD applications by Rajasekaran et al [12,14]. This paper is intended to examine the influence of the focal geometric parameters of the louvered fins on heat exchange and pressure drop to improve the overall performance of the fin-and-tube heat exchanger.

## 2. Geometrical model and grid generation

The intermittent geometry along the tube and the symmetry along the air flow direction allow for simplification of the model geometry. The dashed line in Fig. 1 designates the computational domain. The plan view representation of the computational field and the section cut are shown in figure along with annotations for variable parameters that affect the performance of the fin-and-tube heat exchanger which was analysed in the present work. The dimensions of the model are given in Table 1. ANSYS ICEM CFD was utilized to mesh the airfoil model and control volume. The control volume is created on the boundary of around 14C to clearly study about the far field variations. Unstructured triangular meshes are used and the region around the airfoil surface is fine meshed to acquire precise results. The mesh consists of 3,03,692 nodes and 3,03,468 elements with the minimum sizing of 3.8574 mm and the growth rate is set to 1.2. Fig. 2 shows the meshed airfoil that is fully covered with ice bubbles accretion on the upper surface. The mesh smoothing process is done near the solid.

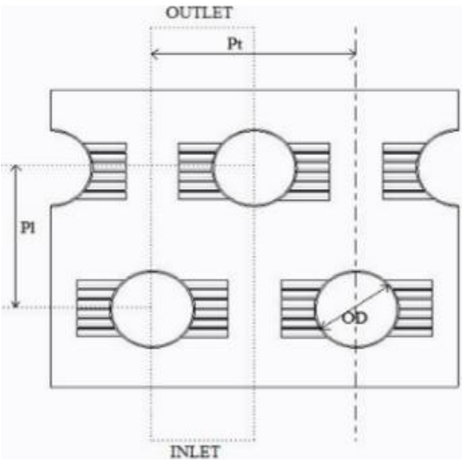



**Fig. 1: Plan view of louvered fins and definitions of geometric parameters**

**Table 1: Dimension of the model**

| Parameter | Value |
|---|---|
| Tube outside diameter | 10.42 mm |
| The number of tube row | 2 |
| Longitudinal tube pitch (Pl) | 19.05 mm |
| Transverse tube pitch (Pt) | 25.4 mm |
| Fin thickness (t) | 0.115 mm |
| Fin pitch (H) | 2.06 mm |
| Number of louvers (N) | 6 |
| Louver angle (θ) | 25° |

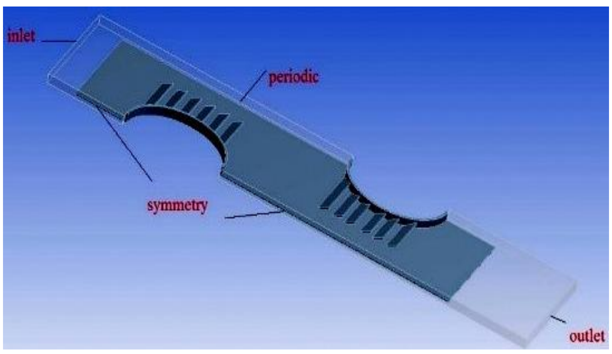



**Fig. 2: Fluid zone (shaded) around the louvered fin (grey)**

## 3. Computational models

### 3.1. Conservation equations

The flow over the louvered fin is assumed to be laminar and steady. In the 3-D models, variations in flow properties along all three coordinate directions are anticipated to be significant. The equations representing the conservation of mass, momentum and energy for the three dimensional models are therefore as follows:

$$\frac{\partial \rho}{\partial t} + \nabla(\rho \vec{u}) = 0 \quad (1)$$

$$\frac{\partial}{\partial t}(\rho \vec{u}) + \nabla(\rho \vec{u}\vec{u}) = -\nabla p + \nabla(\mu(\nabla \vec{u} + \nabla \vec{u}^T) \quad (2)$$

$$\frac{\partial}{\partial t}(\rho E) + \nabla.\left(\vec{u}(\rho E + P)\right) = \nabla.(k.\nabla T) \quad (3)$$

In order to complete this set of equations, additional relations are essential to link thermodynamic and transport properties of air. Air properties can be assumed to change according to ideal gas law.

$$\frac{P}{\rho} = RT \quad (4)$$

The dynamic viscosity of the air is a function of temperature and is obtained from Sutherland's law as,

$$\mu = 1.45 \times 10^{-6} \frac{T^{1.5}}{T+110} \quad (5)$$

Due to a small change in air temperature over the fin, specific heat can be assumed constant and is evaluated at the mean air temperature ($T_{a,i}$ +$T_{a,o}$)/2. The buoyancy and radiation effects have been neglected.

### 3.2. Solution algorithm

Fluid flow simulations are accomplished using commercial fluid flow solver fluent, where the above set of equations is solved using finite volume techniques. Equations are integrated over the individual computational cells and over a finite time increment in event of unsteady simulations. The second-order upwind scheme is used to calculate convective flux on the boundary surfaces of control volumes. This 2$^{nd}$ order scheme is the least sensitive to mesh structure. The SIMPLE algorithm, which utilizes a relationship between velocity and pressure corrections, is used to enforce mass conservation and to obtain the pressure field.

### 3.3. Boundary conditions

The analysed fluid zone consists of an inlet, an outlet, the periodic top and bottom boundaries and the symmetric boundaries on the sides (Fig. 2). The centre part of the domain is either a flat or louvered fin, depending on the analysed case. The fluid flow along the upper and lower surfaces is considered periodic and the one along the sides is considered symmetrical. Uniform dry air flow with constant velocity in the range of from 0.5 - 3.0 m/s and constant temperature $T_{a,i}$ = 330 K is assumed on the upstream boundary. The stream wise gradient for all variables is set to zero at the downstream end of the computational domain which is located at twice the tube diameter length from the last downstream tube row. No-slip conditions for velocity and constant wall temperature ($T_w$ = 333 K) are specified on all solid surfaces. The normal velocity components and normal gradients on the plane of symmetry are set to zero as well as the heat flux.

### *3.4. Numerical mesh*

The influence of cell shape and number is examined in a test case prior to this one. Computational performance of meshes containing tetrahedral, polyhedral and mapped, is analysed separately. The secondary polyhedral meshes are made up of primary tetrahedral meshes of equal cell sizes. Five layers of inflation with a total thickness of 0.16 mm are created on all wall surfaces in order to increase the accuracy. These meshes allow the usage of a minimal number of cells while keeping the same level of accuracy, with four times less volume elements than in tetrahedral meshes as shown in Fig. 3. Mesh independent solution is achieved using a non-conformal mesh with 286433 cells, 850969 faces and 1603556 elements.

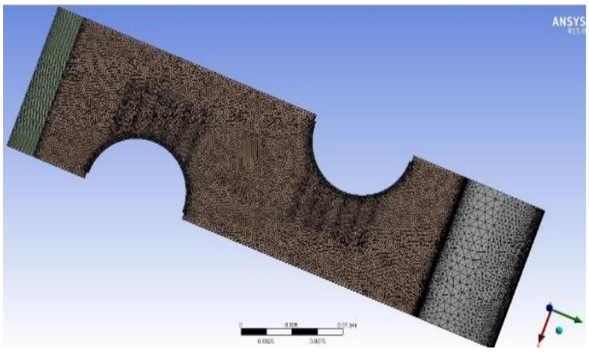


**Fig. 3: Meshed louvered fin**

### *3.5. Convergence criteria*

Fluid flow simulations in all examined cases are performed iterating towards a final stable solution. It is observed that 400 iterations are sufficient to obtain the solution in all examined cases. The convergence criteria used are residuals (a measure of discretization equation solution inaccuracy; a perfectly converged case would approach a measure equal to the computer round-off error) and slight change through further iteration in the area weighted average total pressure variance between the problem inlet and outlet section as shown in Fig. 4.

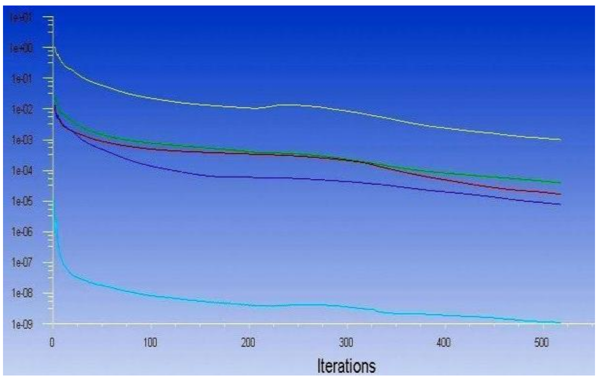
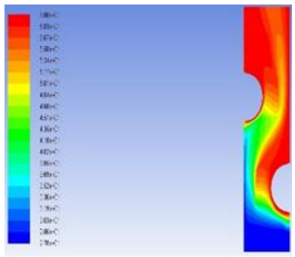



**Fig. 4: Residuals history during calculation**

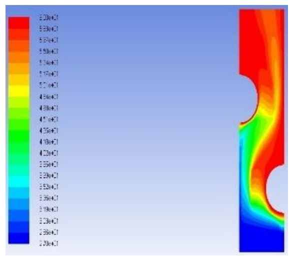
## 4. Results and discussion

This paper mainly evaluates the influence of inlet air velocity on heat transfer characteristics of louvered fin and tube heat exchangers. The louvered fin heat exchanger is analysed with fin parameters set as shown in Table 1. Temperature distribution on the louvered fin surface with different air inlet velocities (0.5 - 3.0 m/s) is shown Fig. 5. Louvered fin temperature is reduced from 333 K to 300 K and pressure remains constant as observed in Fig. 5(a). The pressure is constant and louvered fin temperature reduced from 333 K to 328 K at inlet air velocity of 1 m/s (Fig. 5(b)). The louvered fin temperature reduced from 333 K to 326 K at the inlet air velocity 1.5 m/s (Fig. 5(c)). As illustrated in Fig. 5(d), louvered fin temperature reduced from 333 K to 323 K and pressure is constant at inlet air velocity of 2 m/s.

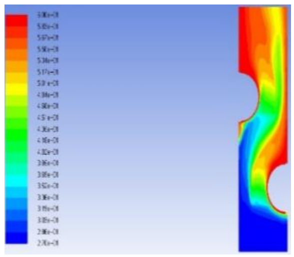
Fig. 5(e) shows that louvered fin temperature reduced from 333 K to 318 K and slight variation of the pressure at inlet air velocity of 2.5 m/s. Fig. 5(f) shows that louvered fin temperature reduced from 333 K to 315 K and maximum pressure drop at inlet air velocity of 3.0 m/s. The influence of louvered fin parameters on convective heat transfer is analysed over the examined range of inlet air velocities (0.5 - 3.0 m/s). The maximum heat transfer is achieved at the inlet air velocity of 3.0 m/s. However at this inlet air velocity pressure drop is also maximum compared to other inlet velocities. Preferred inlet air velocity is 2.5 m/s because even though heat transfer is slightly lesser than at inlet air velocity 3 m/s but pressure drop is minimum.



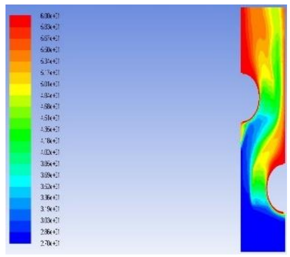
**a) Inlet air velocity 0.5 m/s**



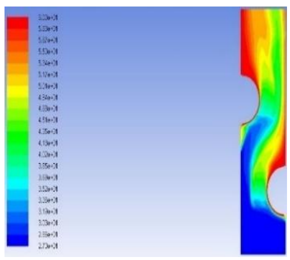
**b) Inlet air velocity 1 m/s**



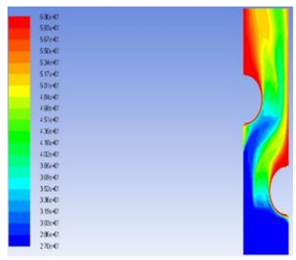
**c) Inlet air velocity 1.5 m/s**



**d) Inlet air velocity 2 m/s**



**e) Inlet air velocity 2.5 m/s**



**f) Inlet air velocity 3 m/s**

**Fig. 5: Temperature distribution on the louvered fin surface for various air inlet velocities (Legend, Min.: 270 K, Max.: 600 K)**

## 5. Conclusions

A fluid flow solver is used to compare the performance of heat exchanger with different inlet air velocities in the range (0.5 - 3.0 m/s). Best combination of higher heat transfer and minimum pressure drop are occurring at inlet air velocity 2.5 m/s. The louvered fin surface can then provide a higher average heat transfer coefficient owing to the periodical renewal of the boundary layer development when the heat transfer from a solid boundary to the moving fluid achieves its peak intensity. Care should be taken when choosing an optimal shape for louvered fins as the louvers remarkably increase the pressure drop.

**REFERENCES:**

[1] W.M. Yan and P.J. Sheen. 2000. Heat transfer and friction characteristics of fin and tube heat exchangers, *Int. J. Heat and Mass Transfer*, 43(9), 1651-1659. https://doi.org/10.1016/S0017-9310(99)00229-X.

[2] J. Dong, J. Chen, Z. Chen, W. Zhang and Y. Zhou. 2007. Heat transfer and pressure drop correlations for the multi-louvered fin compact heat exchangers, *Energy Convers. Manag.*, 48(5), 1506-1515. https://doi.org/10.1016/j.enconman.2006.11.023.

[3] V.P. Malapure, S.K. Mitra and A. Bhattacharya. 2007. Numerical investigation of fluid flow and heat transfer over louvered fins in compact heat exchanger, *Int. J. Therm. Sci*., 46(2), 199-211. https://doi.org/10.1016/j.ijthermalsci.2006.04.010.

[4] H. Han, Y-L. He, Y-S. Li, Y. Wang and M. Wu. 2013. A numerical study on compact enhanced fin and tube heat exchangers with oval and circular tube configurations, *Int. J. Heat and Mass Transfer*, 65, 686-695. https://doi.org/10.1016/j.ijheatmasstransfer.2013.06.049.

[5] Z. Carija and B. Frankovic. 2008. Heat transfer analysis of flat and louvered fin-and-tube heat exchangers using CFD, *Int. J. Heat and Mass Transfer*, 43, 1651-1659.

[6] S. Tiwari, D. Maurya and V. Eswaran. 2003. Heat transfer enhancement in cross-flow heat exchangers using oval tubes and multiple delta winglets, *Int. J. Heat and Mass Transfer*, 46(15), 2841-2856. https://doi.org/10.1016/S0017-9310(03)00047-4.

[7] J.M. Wua and W.Q. Tao. 2008. Numerical study on laminar convection heat transfer in a rectangular channel with longitudinal vortex generator, *Int. J. Heat and Mass Transfer*, 51(5-6), 1179-1191. https://doi.org/10.1016/j.ijheatmasstransfer.2007.03.032.

[8] S.M. Pesteei, P.M.V. Subbarao and R.S. Agarwal. 2005. Experimental study of the effect of winglet location on heat transfer enhancement and pressure drop in fin-tube heat exchangers, *Applied Thermal Engineering*. 25(11-12), 1684-1696. https://doi.org/10.1016/j.applthermaleng.2004.10.013.

[9] M. Rajasekaran, V. Hari Ram and M. Subramanian. 2016. New methodology for light weight solutions to improve BIW structural performance using bulk head optimization, *J. Mech. Science and Technology*, 30(12), 3533-3537. https://doi.org/10.1007/s12206-016-1130-5

[10] C. Lyman, R. Stephan, K. Thole, L.W. Zhang and S.B. Memory. 2002. Scaling of heat transfer coefficients along louvered fins, *Experimental Thermal and Fluid Science*, 26(5), 547-563. https://doi.org/10.1016/S0894-1777(02)00163-2.

[11] Y. Xia and M. Jacobi. 2010. A model for predicting the thermal-hydraulic performance of louvered-fin, flat-tube heat exchangers under frosting conditions, *Int. J. Refrigeration*, 33(2), 321-333. https://doi.org/10.1016/j.ijrefrig.2009.09.021.

[12] M. Rajasekaran, V. Hari Ram and M. Subramanian. 2016. Multiobjective optimization of material layout for body-in-white using design of experiments, *Int. J. Vehicle Structures & Systems*, 8(1)(2016) 17-22, doi:10.4273/ijvss.8.1.04.

[13] D. Taler and P. Oclan. 2014. Determination of heat transfer formulas for gas flow in fin-and-tube heat exchanger with oval tubes using CFD simulations, *Chem. Eng. Process*, 83, 1-11. https://doi.org/10.1016/j.cep.2014.06.011.

[14] M. Rajasekaran, V. Hari Ram and M. Subramanian. 2016. New methodology for light weight solutions to improve BIW structural performance using bulk head optimization, *J. Mech. Science and Technology*, 30(12), 5617-5623. https://doi.org/ 10.1007/s12206-016-1130-5