Paper ID: 391

Analysis of Plate-Fin-Tube Heat Exchanger

Sultan Mahmud[1]
[1]Khulna University of Engineering and Technology, Khulna, Bangladesh
E-mail: sultanmahmudkuetme19@gmail.com

Abstract

Computational fluid dynamics (CFD) simulation has been conducted In order to examine the heat transfer properties of a plain fin tube heat exchanger, ANSYS 2020R1 simulation software package. The heat transfer characteristics have been examined for Reynolds numbers between 500 and to 1500. In this paper, results obtained from the simulation has been compared to the experimental data and found to be almost similar. The Reynolds numbers effects on heat transfer rate has been found to be linear proportional. The pressure drop and contour of velocity magnitude has not been covered in this paper.

Keywords: ANSYS, CFD, Fin, Heat exchanger.

1. Introduction

Plate-fin-tube heat-transfer devices are among the most popular devices in HVAC systems. This type of heat exchanger primarily uses air as its cooling agent. One major error that has been occurring while designing the tube fin type of heat exchangers is that it has been assumed to have a uniform flow of air and uniform pressure distribution of flow across the heat exchangers [1]. Taler *et al.* [2] formed a plate-fin and tube numerical model automotive radiator that was created and used in a digital control system. to determine the number of fan rotations per minute required to keep a constant water temperature at the heat exchanger outlet. Bhuiyan *et al.*[3] investigated that fins improve heat transfer. rates and control the cooling system. 85% or more heat resistance for HVAC heat exchanger is mainly on the air side. Wang *et al.* [4] discovered that the heat-transfer coefficient is unaffected by the fin spacing. Anna *et al.* [5] calculated the transient temperature distribution in each fin. Lindqvist *et al.*[6] developed a CFD model and validated to study plate fin-and-tube heat exchangers' thermal-hydraulic performance, with a focus on the influence of the tube array angle on heat transfer and pressure drop. Yaïci *et al.* [7] investigated using 3D CFD models, the thermo-hydraulic performance of heat exchangers utilized in air handling units was impacted by the inlet airflow maldistribution. The experimental findings were compared to the CFD model for validation. The study of Chen *et al.* [8] shows that on four circular tubes for the staggered arrangement of two-row plate finned-tube heat exchangers with a vertical square fin, the average heat transfer coefficient and the heat transfer coefficient under isothermal conditions, different air velocities, and fin spacings are determined using experimental and numerical inverse methods. Saboya *et al.* [9] conducted experiential research on elliptic sections in heat exchangers with plate fins and tubes arranged in one and two rows to calculate average transfer coefficients and compare with circular tube exchangers. In this paper, it has been investigated the pressure and temperature distribution, using the ANSYS 2020R1 Fluent software package. The work of this paper involves CFD modeling, validation, and simulation for four different Reynold numbers. The outcome of this paper is expected to be valuable for improving the heat exchanger's design.

2. Methodology

Physical Model

In this paper, Fig.1(a) presents the schematic figure of the heat exchanger considered for CFD simulation. Fig.1 (b) shows the considered 2D section for simulation. The staggered arrangements have been taken into consideration in this paper. The CFD simulation will be carried out by using ANSYS 2020R1 Fluent software package.

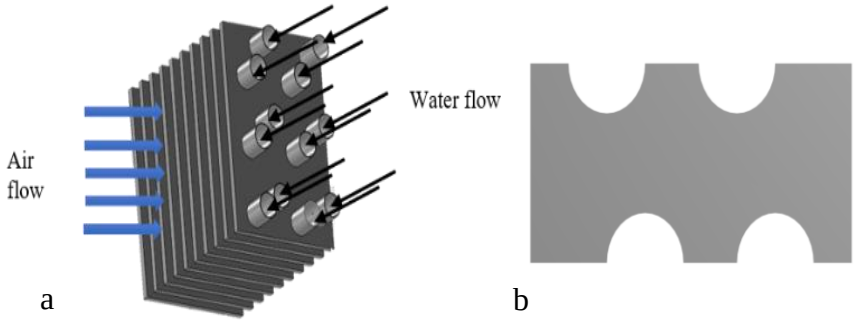


Fig.1. Staggered arrangement of heat exchanger with plate-fin-tube.

The values of geometry have been listed in Table 1[3]. Specifically, simulation has been conducted for Reynolds number 500, 800,1000,1500.

Governing Equations

The governing equations for the simulation has been given as follows[10]:

Nusselt Number as $Nu_D = C_1 C_2 Re^n Pr^m (\frac{Pr}{Pr_s})^{0.25}$

Heat Transfer Rate as - $Q = mC_p(T_{out} - T_{in})$

Logarithmic mean temperature difference, $LMTD = \frac{(T_w - T_{in}) - (T_w - T_{out})}{\ln \frac{(T_w - T_{in})}{(T_w - T_{out})}}$

The Fanning *f*-friction factor is stated as- $f = \frac{(P_{in} - P)H}{\frac{1}{2}\rho u_{in}^2 4L}$

The Colburn *j*-factor is stated as- $j = \frac{Nu}{RePr^{\frac{1}{3}}}$

Reynolds number is stated as- $Re = \frac{\rho u_{in} H}{\mu}$

Model Validation

Mesh independency test has been conducted to find out the optimum number of divisions for meshing which will save time and give results independent of mesh number. It has been observed that after 500 number of divisions the value of Total heat transfer rate varies between 79 and 78.9. In this simulation, the number of divisions has been set to 500 as the variation of results is less than 0.1%. As the number of divisions increases, the variation in results tends towards zero. But for optimum result, 500 number of divisions is considered suitable for this simulation.

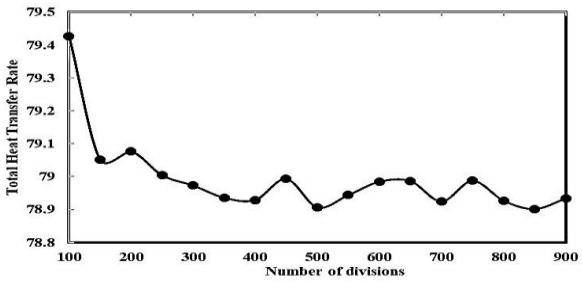


Fig.2. Mesh independency test.

After meshing operation, total generated elements was 159759 and nodes were 146369. Around the tube the mesh was more in number. First layer thickness has been set to 0.003 and 65 layers have been created around the tube wall for better result.

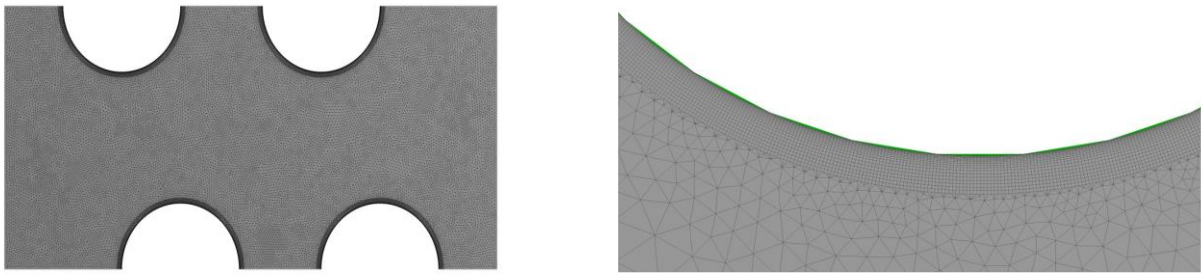


Fig.3. Mesh of the considered section and around the tube wall.

The experimental data from the experiment of Wang et al. [4] has been used to validate the model created in ANSYS Fluent. The geometrical data reported by Wang et al. [4] has been taken into consideration while creating the model, and all the parameters have been set nearly the same as their experiment. For example, L_l=28.00 mm, L_t=32.00mm, and D=10.00mm. The air temperature is 298 K , and the wall temperature is 373 K. Air has been used as the working fluid in the simulation, and its properties have been set as follows: ρ=1.1831 kg/m^3, μ=1.831*10^{-5} Pa s, k=0.0261 W/m K, Pr=0.736 [4]. The Reynolds number values have been set to 500, 800, 1000 and 1500. The present study data has been plotted with the numerical study of Bhuiyan *et al.* [3] and experimental study of Wang *et al.* [4]. It has been observed that the graph of the present study is following their results which validate the present simulation.

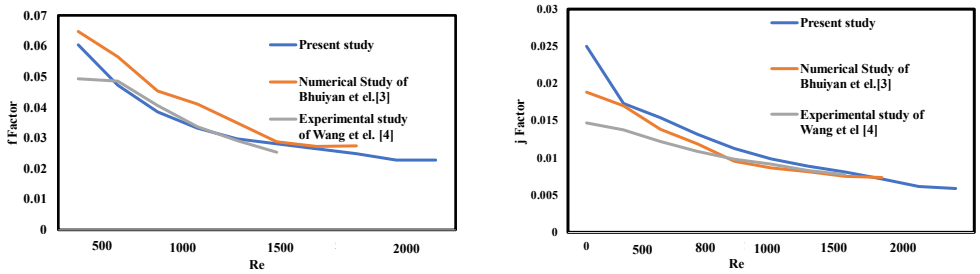


Fig.4. Current analysis is compared to the experimental and numerical values of the Fanning f factor and Colburn factor j in relation to Reynolds number.

Fig.4 illustrates the experimental and numerical comparison with the present study between friction factor *f* and Colburn factor *j* against the Reynolds number [3,4]. It has been observed from the graph that the line for present study is following the trend of the line of experimental results and numerical results. The negligible amounts of variations of these lines validate the current simulation for this paper. In this paper, the range of Reynolds number has been considered from 500 to 1500.

Boundary Conditions

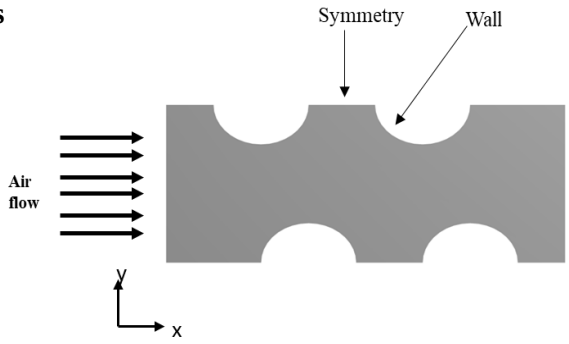


Fig.5. Considered domain with boundary conditions.

The boundary conditions have been showed in Fig.5. Symmetry condition has been imposed between two fins on the midplane. For making the computation a little bit easier only one half of the domain has taken in consideration.

At inlet, Dirichlet boundary conditions has been imposed with uniform flow velocity u_{in} and constant temperature T_{in}[11]. At the outlet, the boundary condition has been set to be pressure outlet. No slip boundary conditions have been set at the fin and tube surfaces.

$$U=u_{in,}\ v=w=0,\ T_{in} = 293\ K$$

It has been assumed that the working fluid in the flow model is air and the flow is of laminar type. The fluid has been taken as to be Newtonian fluid, constant inlet properties with incompressibility. The inlet air temperature has been set to 298 K. No body force has been considered. The geometrical parameters are set as mentioned in the Table 1.

Table 1. Geometrical Parameters of the model

Parameters	Value
Tube diameters, D(mm)	10.00
L_I (mm)	28.00
L_t(mm)	32.00
Tube number	4
Arrangements of tube	Staggered
Value of Re	500,800,1000,1500

4. Results and discussion

The temperature distribution, total pressure and total heat transfer rate of staggered configuration has been analyzed in this paper, The effect of different Reynolds numbers (500,800,1000 & 1500) on flow of air has been investigated on the temperature. The temperature distribution from for Reynolds number 500,800,1000 and 1500 respectively, allows to observe that the highest temperature is imposed only after the first tube and the successive tube encounters less temperature. As the Reynolds number increases, it has been observed that there is a turbulence after the first column of tube.

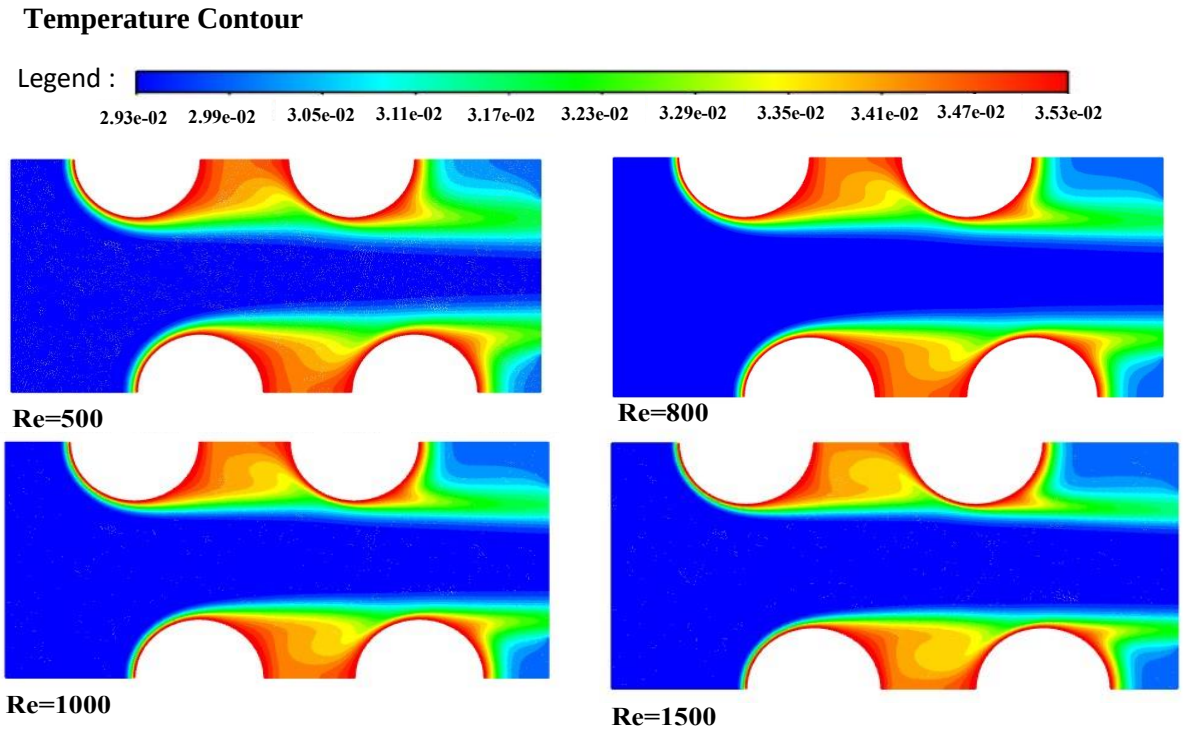


Fig.6. Temperature contour for Re=500,800,1000,1500.

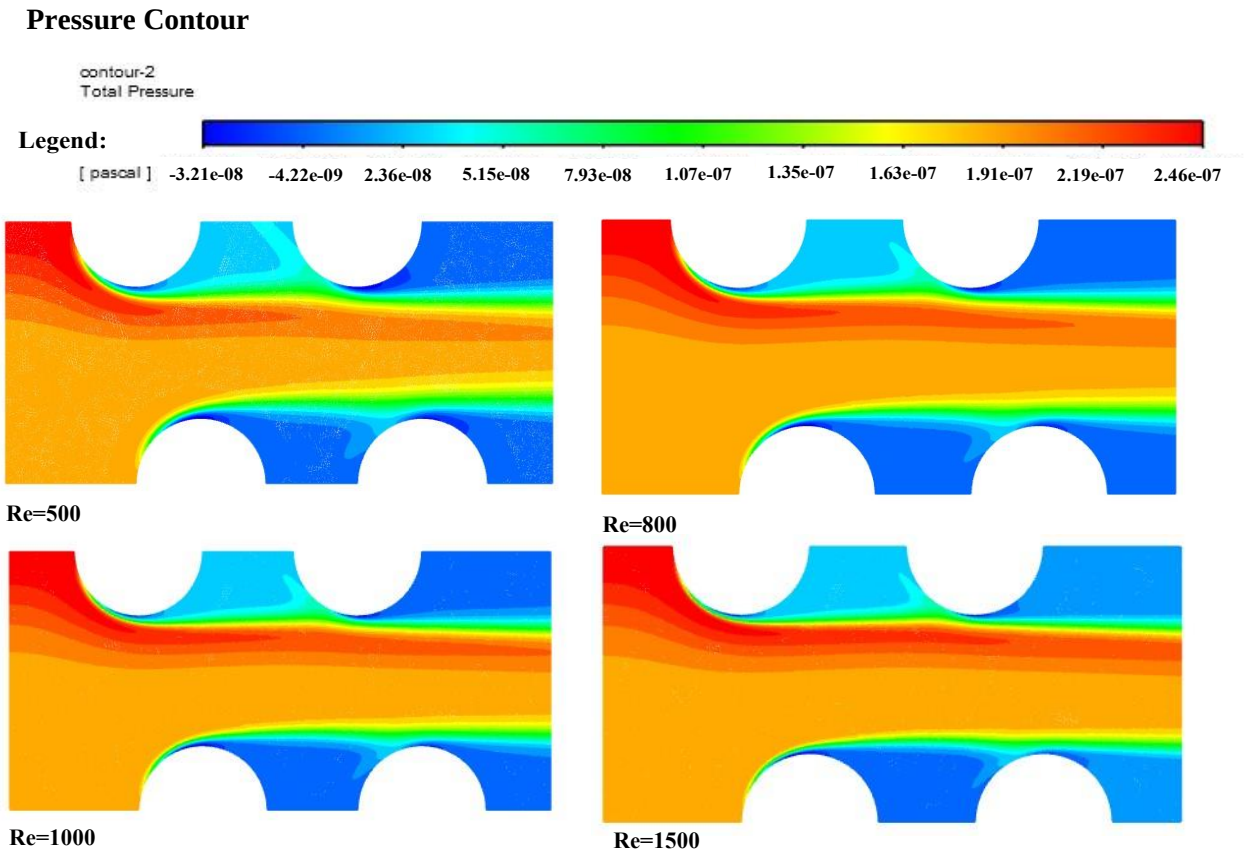


Fig.7. Total pressure contour with legend.

In the Fig.7, pressure contour for Reynolds number 500,800,1000 & 1500 have been generated from the solution module of ANSYS Fluent. From the pressure contour, it is evident that the top right most first tube faces the maximum pressure of flow. For the rest of the column of tube, the pressure due to the flow is minimum. There is no significant changes found in pressure due the changes in flow velocity.

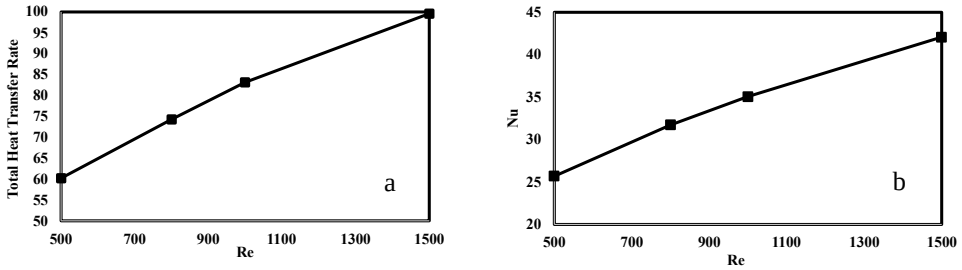


Fig.8(a). Total Heat Transfer Rate Vs Re and **8(b).** Nu Vs Re.

The Reynolds number influences the Nusselt number and the total heat transfer rate. It can be shown from Fig. 8(a) that the total heat transfer rate increases nearly linearly from Re=500 to Re=1500 as Re increases. Figure 8(b) illustrates how, within the area covered in this work, the Nusselt number has also increased linearly as the Reynolds number has increased. From Fig.9, it has been observed that there is a rapid slowdown in the velocity magnitude against the position and then some slight pulsating magnitude shows up with almost similar differences. There is no considerable difference has been observed according to the change of Reynolds number except the magnitude of the velocity. It is also observed in this paper that the velocity magnitude is a function of Reynolds number foe fluid flowing in heat exchangers [10].

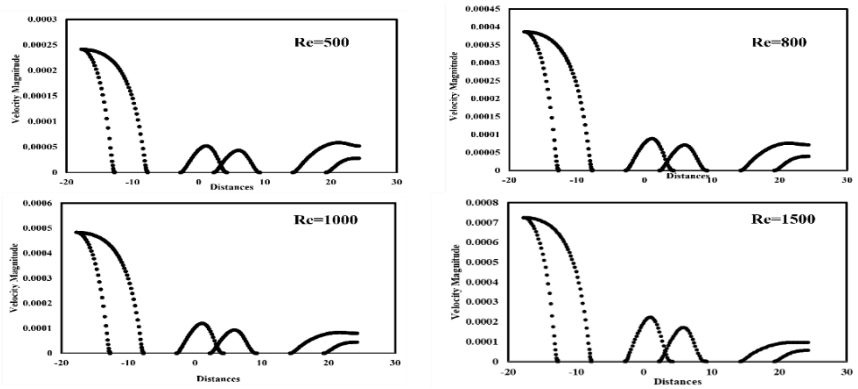


Fig.9. Velocity magnitude Vs Distance.

5. Conclusion

The staggered plate-fin-tube heat exchangers have been simulated for Re=500,800,1000, and 1500 using ANSYS Fluent software, assuming that the air flow is laminar and incompressible. The current study illustrates the relationship between heat transfer rate, Nu, and Re as well as the temperature and total pressure contour. This study found—

- The total heat transfer rate increases in tandem with the Reynolds number.
- The higher the Reynold number, the higher the Nusselt number within the range of Re=500 to Re=1500.
- The flow velocity has no significant effect on the total pressure after the first column of tube. So in case of, laminar flow with Reynolds number between 500 to 1500, the effect of pressure drop and flow maldistribution can be ignored only if there is huge number of tube in staggered heat exchangers.

6. References

[1] Taler, D. (2015). Mathematical modeling and control of plate fin and tube heat exchangers. Energy Conversion and Management, 96, 452–462. https://doi.org/10.1016/j.enconman.2015.03.015 .

[2] Korzeń, A., & Taler, D. (2015). Modeling of transient response of a plate fin and tube heat exchanger. International Journal of Thermal Sciences, 92, 188-198. https://doi.org/10.1016/j.ijthermalsci.2015.01.036 .

[3] Bhuiyan, A. A., Amin, M. R., & Islam, A. S. (2013). Three-dimensional performance analysis of plain fin tube heat exchangers in transitional regime. Applied Thermal Engineering, 50(1), 445-454. https://doi.org/10.1016/j.applthermaleng.2012.07.034

[4] Wang, C., Chang, Y., Hsieh, Y., & Lin, Y. (1996). Sensible heat and friction characteristics of plate fin-and-tube heat exchangers having plane fins. International Journal of Refrigeration, 19(4), 223-230. https://doi.org/10.1016/0140-7007(96)00021-7

[5] Moffat, R. J. (1987). Describing the uncertainties in experimental results. Experimental Thermal and Fluid Science, 1(1), 3-17. https://doi.org/10.1016/0894-1777(88)90043-X.

[6] Lindqvist, K., Skaugen, G., & Meyer, O. H. (2021). Plate fin-and-tube heat exchanger computational fluid dynamics model. Applied Thermal Engineering, 189, 116669. https://doi.org/10.1016/j.applthermaleng.2021.116669

[7] Yaïci, W., Ghorab, M., & Entchev, E. (2014). 3D CFD analysis of the effect of inlet air flow maldistribution on the fluid flow and heat transfer performances of plate-fin-and-tube laminar heat exchangers. International Journal of Heat and Mass Transfer, 74, 490-500. https://doi.org/10.1016/j.ijheatmasstransfer.2014.03.034

[8] Chen, H., & Lai, J. (2012). Study of heat-transfer characteristics on the fin of two-row plate finned-tube heat exchangers. International Journal of Heat and Mass Transfer, 55(15-16), 4088-4095. https://doi.org/10.1016/j.ijheatmasstransfer.2012.03.050

[9] Saboya, S. M., & Saboya, F. E. (2001). Experiments on elliptic sections in one- and two-row arrangements of plate fin and tube heat exchangers. Experimental Thermal and Fluid Science, 24(1-2), 67-75. https://doi.org/10.1016/S0894-1777(00)00059-5

[10] Jayavel, S., & Tiwari, S. (2010). Finite Volume Algorithm to Study the Effect of Tube Separation in Flow Past Channel Confined Tube Banks. Engineering Applications of Computational Fluid Mechanics, 4(1), 39–57. https://doi.org/10.1080/19942060.2010.11015298

[11] Wen, J., & Li, Y. (2004). Study of flow distribution and its improvement on the header of plate-fin heat exchanger. Cryogenics, 44(11), 823-831. https://doi.org/10.1016/j.cryogenics.2004.04.009