Journal of Complex Flow, Vol. 6 No. 1 (2024) p. 23-28

FAZ Publishing

JCF

Journal of Complex Flow

Journal homepage: www.fazpublishing.com/jcf

e-ISSN : 2672-7374

Numerical Study of Convective Heat Transfer from Tube Banks with In-Lined Arrangement in Crossflows

Haiqal Haziq Nor Sham[1], Suzairin Md Seri[1,*]

[1] Faculty of Mechanical and Manufacturing Engineering
Universiti Tun Hussein Onn Malaysia, Parit Raja, Batu Pahat, Johor, MALAYSIA

*Corresponding Author

Received 17 Nov 2023;
Accepted 12 Dec 2023;
Available online 15 April 2024

Abstract: The inline crossflow heat exchanger is commonly used in the process industry due to optimum heat transfer efficiency and lower price than other heat exchangers. The design of the unit is frequently based on modelling of flow arrangement. In this study, tube pitch distance ratio, Reynolds number, and fluid velocity entering heat exchangers vary. The spacing between tubes in a heat exchanger is directly related to the resistance to air flowing around it. Smaller spacing increases the surface contact area per volume, promoting better heat transfer. However, it requires higher pumping power to push air through the tight space. Optimum tube spacing will give the best ratio of heat transfer over the required pumping power. A numerical investigation was made on the flow-through-in-line arrangements heat model consisting of 6 tubes per row in a range of Reynolds numbers 1,000 to 20,000. The distribution of local static pressure was determined around the tubes in several ratios of tube pitch distance. Furthermore, the total pressure drop was measured. This study uses Computational Fluid Dynamics (CFD) software, which is ANSYS FLUENT, to review its pressure drop and velocity vectors when the fluid flows across the vertical tube.

Keywords: Heat Transfer, Experimental Study, Tube Bank, Heat Exchanger, Turbulence, Reynold Number, Pressure Coefficient

1. Introduction

The number of heat transfers per unit of pressure decreases the heat exchanger's efficiency. Smaller distances between tubes can increase flow friction and, unfortunately, pumping power. The most efficient heat exchanger design requires the least pumping power and has the lowest power consumption [1,2]. Therefore, the optimal tube spacing must be identified to reduce pump energy consumption. This study examines the effect of tube pitch spacing on the performance of In-line cross-flow heat exchangers. The governing equations for velocity and pressure drop field are solved using the Ansys Fluent programme. The flow behaviours that result in the best heat exchanger performance may be determined using the result and the optimal tube pitch distance.

The research indicates that the type of tube bank architecture in a heat exchanger substantially affects its performance. When creating or selecting the appropriate heat exchanger, thermodynamic parameters such as heat transmission are crucial [3,4]. Among the criteria is the fluid flow over the tube banks. Additionally, this idealises several other industrially relevant processes [5]. As a result, it is essential to analyse the characteristics of fluid flow and heat transfer for tube banks in crossflow, as this will aid in predicting heat transfer and ensuring that heat exchangers are constructed or designed appropriately.

From the problem statement stated the objectives of this research are to investigate the characteristics of the thermal and flow field of crossflow tubular heat exchanger inline arrangement and to obtain a correlation between flow geometries and the thermal performance of crossflow tubular heat exchangers in in-lined arrangements. This study was handled in simulation through ANSYS with some scopes that need to be emphasised to build the desired model of heat exchangers. The models were built with eight rows of inline cylindrical tubes. The longitudinal and transverse ratio of pitch distance to diameter is 1.2 to 2. The last one is the most important to set the velocity in the ANSYS: to set the Reynold numbers, *Re*, between 1000 and 20 000.

**Corresponding author: suzairin@uthm.edu.my*

2. Design and Methodology

This research is required to acquire the results of the analysed data correlation between flow geometries and the thermal performance of crossflow tubular heat exchangers in an inline-square configuration. Aside from that, this study aims to examine the thermal and flow field properties of an inline-square crossflow tubular heat exchanger. This research is important due to its wide range of uses, including educational equipment and power plants. The mathematical relationship between all the listed variables has not been exhaustively studied [6-8]. The thermal performance of the contemporary crossflow heat exchanger system is affected by numerous variables.

The previous study from Shinya et al. [9] used a tube bank of inline tubes with a diameter of 25 mm, seven rows, and four columns. In this experiment, a wind tunnel that was 120 mm high and 225 mm wide was also used. It is indicated in Figure 1.

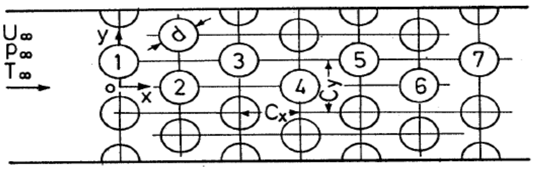

Fig. 1 – Inline tube banks; $C_y/d = C_x/d = 1.6$ [9]

Since *k-kl-omega* has the best value among the other models regarding the normalised root Mean Square of Error, it was chosen as the model with the best outcome. Validate the result with previously published experimental data. The local pressure coefficient, C_p vs angle from the forward stagnation point, θ shown in Figure 2 for data validation.

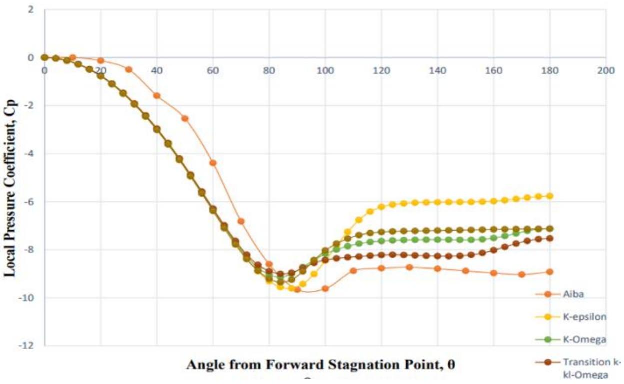

Fig. 2 – Local pressure coefficient, C_p vs angle from forward stagnation point, θ

The result from S. Aiba et al. [10] was compared to the simulation result. Various values of Reynold numbers, 1000, 5000, 10000, 15000, and 20000, have been used to handle the simulation. The Reynold number formula can be used to determine the value of mean velocity, then can be used to calculate the inlet velocity, U_∞.

$$\mathrm{Re} = \frac{\rho U_T D}{\mu} \tag{1}$$

Value from previous U_t can be used to calculate inlet velocity, $\boldsymbol{U_\infty}$ by using the following equation.

$$U_\infty = \frac{U_T(s-D)}{s} \tag{2}$$

Other equations used to obtain the correlation for the convective heat transfer from tube banks of 8 rows with in-lined arrangements in crossflow, such as the pumping power per area, efficiency and overall performance, were shown in the following equations.

$$P_{pump} = P_{inlet} - (P_{outlet} \times Vel_{inlet}) \tag{3}$$

$$\eta = \frac{\text{desired output}}{\text{required input}} \tag{4}$$

$$\eta_{overall} = \frac{\text{average heat flux}}{P_{pump}} \times 100\% \tag{5}$$

2.1 Geometry Modelling

The geometry for this project is modelled in Design Modeler before the simulation begins. It has four main components: inlet, outlet, circular tubes, and walls. Figure 3 shows the geometry that is modelled and used throughout the project. The parameters used for modelling the geometry of Figure 3 include the tube diameter (d = 0.025 mm) and the pitch distance-to-diameter ratio (P = 1.2d, 1.5d, 1.8d, 2.1d).

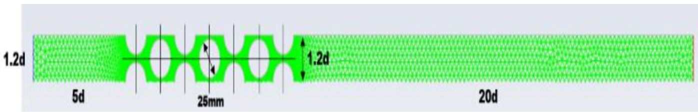

Fig. 3 – Dimension sketching of 2-dimensional model

2.2 Grid Independence Test (GIT)

This research performed a grid-independent test before initiating the ANSYS Fluent simulation to improve results and increase convergence. A coarse mesh was created by reducing the number of nodes and elements. As the mesh of the crossflow heat exchanger becomes finer, the number of nodes and elements increases. The grid-independence test was implemented using several mesh sizes, from 1.0 to 1.2. Table 1 below summarises the number of elements with different mesh sizes.

Table 1 – Number of elements of different mesh grid

Mesh grid	Grid 1.0	Grid 1.01	Grid 1.02	Grid 1.05	Grid 1.1
No of elements	62114	20369	15768	12188	10942

Figure 4 shows the fluid domain's pressure coefficient graph around cylindrical tubes with different meshing grids simultaneously. In contrast, Figure 5 shows the mesh of the model with a grid size of 1.0 to 1.1, respectively.

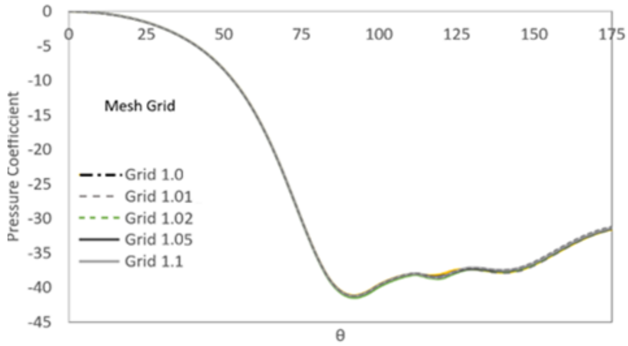

Fig. 4 – Local Pressure Coefficient (C_p) against Angle from Forward Stagnation Point, θ

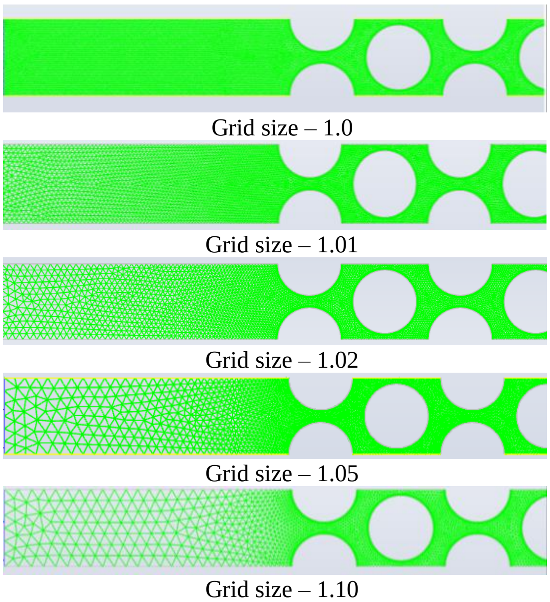

Fig. 5 – Meshing size for the model with different meshing sizes.

The grid independence test will produce more accurate results if calculations are performed with lower cell sizes, and as the mesh gets finer, calculations should yield the right answer. The meshing images are shown in Figure 5. Grid 1.0, Grid 1.01, Grid 1.02, Grid 1.05, and Grid 1.1 systems first implemented the grid independence study. Studies on grid independence are based on stacked configurations and staggered arrangements of cylinder tubes with various grid independents.

2.3 Root Mean Square Error (RMSE)

The root mean square error (RMSE) is the residual standard deviation (prediction errors). The residuals of data points indicate how far they deviate from the regression line, and the RMSE indicates how spread these residuals are. In other words, it provides information on how tightly the data is concentrated around the line of best fit. Root means square error is frequently used to validate experimental results in climatology, forecasting, and regression analysis. RMSE comparison based on the turbulent model is shown in Figure 6.

From the figure, Transition *k-kl-omega* was chosen for the best result because, based on the calculated normalised root mean square of error, Transition *k-kl-omega* has a good value among other models.

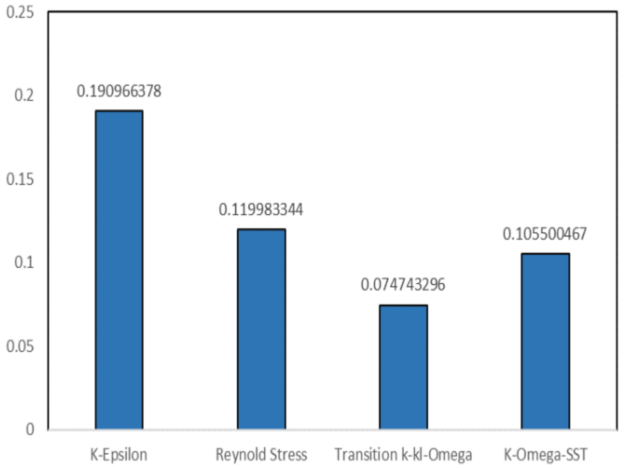

Fig. 6 – Graph of Normalized Root Mean Square of Error (NRSME)

3. Results and Discussion

The simulation results are reported in this section using Ansys Workbench. This section discusses the results of the Ansys Workbench simulation using Fluent. The crossflow tubular heat exchanger with an in-line formation was achieved and displayed in a meshing configuration. Twenty simulations were conducted, each with a unique pitch distance to diameter ratio and a Reynolds number ranging from 1,000 to 20,000. Two-dimensional, steady-state, incompressible conditions were the primary focus of the numerical analysis. All twenty simulations have used appropriate meshing configurations to achieve excellent accuracy and validity. The numerical test represents each model's pressure drops, distribution, and velocity vectors.

3.1 Average Heat Flux

The relationship between the average heat flux and the Reynolds number is displayed in Table 2 and Figure 7. Between distance-to-diameter ratios of 1.8 and 2.0, there is little variation in heat flux, as depicted by the graph. Large gaps exist between values of 1.2 and 1.5 for pitch distance to diameter, while the gap narrows between 1.8 and 2.0. It was predicted there would be no difference if the pitch distance-to-diameter ratio increased from 3.0 to 5.0. Consequently, the Reynolds number affects surface heat flux values with varied pitch distance-to-diameter ratios.

Table 5 – Average heat flux at different spacing diameter

	Spacing Diameter			
Re	1.2	1.5	1.8	2.0
1 000	584.74	0.177	2.389	0.0034
5 000	2168.17	0.176	2.360	2.722
10 000	4053.98	0.749	2.448	2.704
15 000	13402.04	2.390	2.462	2.688
20 000	16652.85	4.929	2.367	2.699

Published by FAZ Publishing
http://www.fazpublishing.com/jcf

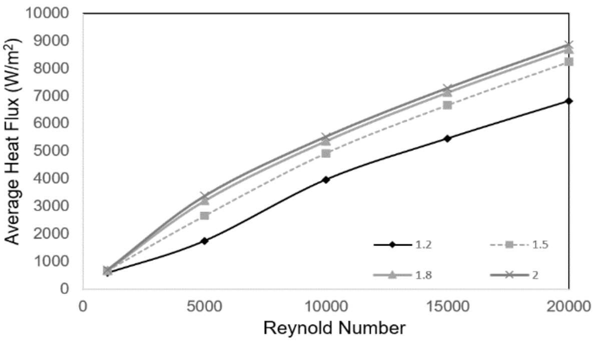

Fig. 7 – Average Heat Flux vs Reynolds Number

3.2 Average Nusselt Number

The graph of the average Nusselt number vs the Reynolds number is shown in Table 6 and Figure 8. The graph reveals that there is not much difference in the heat flow between values of 1.8 and 2.0 for the pitch distance-to-diameter ratio. The difference is significant when comparing values of 1.2 and 1.5 for the pitch distance-to-diameter ratio. However, it is less pronounced when comparing 1.8 and 2.0. According to my forecast, there will be no change even if the ratio of the pitch's distance to its diameter grows from 3.0 to 5.0. As a direct consequence, The Reynolds number impacts the values of the surface heat flux for a variety of pitch distance-to-diameter.

Table 6 – Average Nusselt number at different spacing diameter

	Spacing Diameter			
Re	1.2	1.5	1.8	2.0
1 000	584.74	0.177	2.389	0.0034
5 000	2168.17	0.176	2.360	2.722
10 000	4053.98	0.749	2.448	2.704
15 000	13402.04	2.390	2.462	2.688
20 000	16652.85	4.929	2.367	2.699

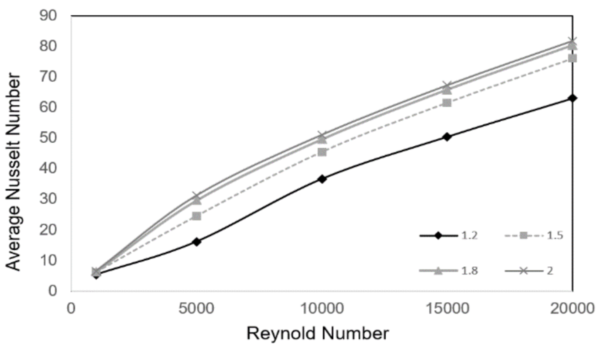

Fig. 8 – Average Nusselt Number vs Reynolds Number

3.3 Euler Number

Table 7 together with Figure 9 shows the graph of the Euler Number. The Euler number represents the pressure differential. If we raise the pumping power, kinetic energy will also rise. However, the Euler number describes a tube's pressure drop or head loss rather than the fluid flow behaviour. The Euler number is the pressure drop or head loss ratio to the fluid kinetic energy. The relationship between the Euler and Reynolds numbers is complex and depends on many factors, such as the fluid 45 properties, tube geometry, and flow conditions. The Euler number generally decreases as the Reynolds number increases, indicating that the head loss or pressure drop decreases as the fluid flow becomes more turbulent. The fluid loses less energy to friction.

Table 7 – Euler Number

	Spacing Diameter			
Re	1.2	1.5	1.8	2.0
1 000	4.25E+05	3.37E+13	3.26E+19	3.89E+23
5 000	1.69E+04	6.76E+12	1.20E+18	4.72E+25
10 000	3.54E+03	5.07E+11	3.05E+17	1.15E+25
15 000	1.10E+02	1.70E+10	1.17E+16	3.61E+23
20 000	2.32E+01	3.70E+09	2.90E+15	8.37E+22

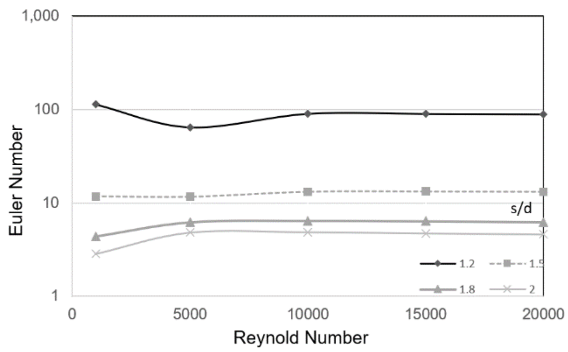

Fig. 9 – Euler Number vs Reynolds Number

3.4 Pump Power Efficiency

Table 8 and Figure 10 show the graph for pumping power per area, showing that when the ratio of pitch distance to diameter is 1.2, the inlet pressure will rise with an increase in pitch distance to diameter. The velocity of the fluid entering the pump is what determines its power. Even if it reduces overall performance, use a high pumping power to remove a lot of heat transfer.

$$Power_{Pump} = 4.3*10^{-11}*(S/d)^{0.83}*Re^{3}$$

Table 8 – Pump power efficiency

	Spacing Diameter			
Re	1.2	1.5	1.8	2.0
1 000	4.25E+05	3.37E+13	3.26E+19	3.89E+23
5 000	1.69E+04	6.76E+12	1.20E+18	4.72E+25
10 000	3.54E+03	5.07E+11	3.05E+17	1.15E+25
15 000	1.10E+02	1.70E+10	1.17E+16	3.61E+23
20 000	2.32E+01	3.70E+09	2.90E+15	8.37E+22

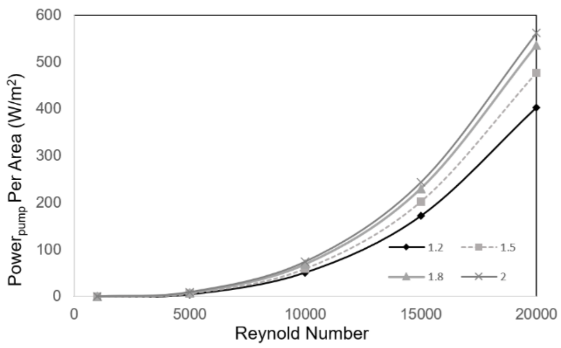

Fig. 10 – Power Pump/Area vs Reynolds Number

Published by FAZ Publishing
http://www.fazpublishing.com/jcf

Figure 11 and the equation below can represent a four-line graph. If the dotted line crosses the red line, the correlation and numerical result values will be identical. The correlation value is virtually identical to the numerical result of the parity plot and the numerical result of the simulation.

$$\text{Power}_{Pump} = 4.3*10^{-11}*(S/d)^{0.83}*Re^{3}$$

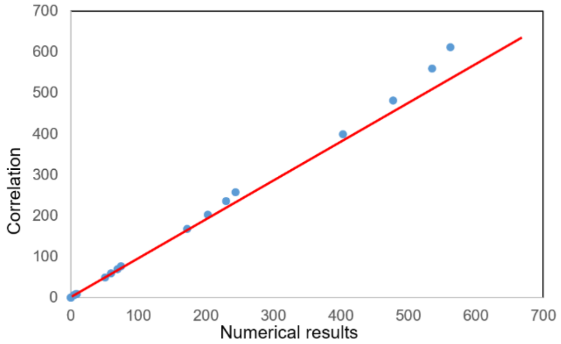

Fig. 11 – Parity Plot Pumping power

3.5 Overall Performance

Table 9 shows the data for the overall performance of the tube bank with the different spacing diameters. The pressure decreases from 1000 to 5000 and is very large. The pressure drops from 5000 to 20000, which is not much different. The overall performance remains unchanged as the Reynolds number rises. Overall performance will be unchanged. A new equation we create can represent a four-line graph.

Table 9 – Overall performance of the tube bank

	Spacing Diameter			
Re	1.2	1.5	1.8	2.0
1 000	4.25E+05	3.37E+13	3.26E+19	3.89E+23
5 000	1.69E+04	6.76E+12	1.20E+18	4.72E+25
10 000	3.54E+03	5.07E+11	3.05E+17	1.15E+25
15 000	1.10E+02	1.70E+10	1.17E+16	3.61E+23
20 000	2.32E+01	3.70E+09	2.90E+15	8.37E+22

The equation below shows that the correlation value is virtually identical to the numerical result of the parity plot and the numerical result of the simulation. Figure 12 is a plot of overall performance using the equation for correlation. A parity plot's correlation and numerical result are identical if a dot is on the red line.

$$\text{Overall, } \eta = 7.66*10^{9}*(S/d)^{1.1}*Re^{-2}$$

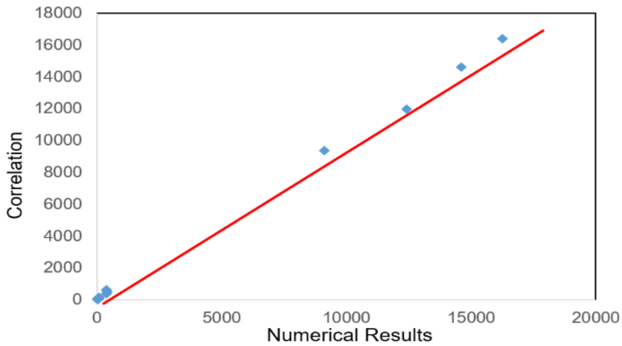

Fig. 12 – Parity plot for overall performance

4. Conclusion

An inline arrangement for a cross-flow heat exchanger has optimum thermal efficiency and costs less than other heat exchangers. Besides, its performance is determined by the arrangement of the cross-flow heat exchanger. From this study, it can be concluded that the numerical study on the effect of tube pitch distance on the performance of cross-flow heat exchangers with inline arrangement has been successfully achieved. The numerical result of pressure drops and Euler number when the fluid flows through the tube have been calculated by using Computational Fluid Dynamic (CFD) software.

The objective of this study has been successfully achieved, and the numerical prediction of pressure drops when fluid flows across tubes has been conducted and compared with experimental results to obtain good results. Based on the study, the conclusions are as shown below:

1. The pressure drop depends on the Reynolds number and the tube banks' pitch distance ratio; the relationship between the Euler number and the pitch distance-to-diameter ratio was discovered to have no effect.
2. A new finding found that for pitch distance-to-diameter ratios of 1.5 to 2.0, the effect of the distance itself does not affect the Euler number.

Acknowledgement

The authors sincerely thank the Faculty of Mechanical and Manufacturing Engineering at Universiti Tun Hussein Onn Malaysia for their continuous support throughout the research process, including access to facilities, provision of materials, and analysis of findings, which has been instrumental in completing this research paper.

References

[1] A. Bejan, R. Re. (1995) The optimal spacing for cylinders in crossflow forced convection, *Journal of Heat Transfer*, 117(3) 767–770.

[2] Achenbach, E., (1991) Heat transfer from smooth and rough in-line tube banks at high Reynolds number, *Int. J. Heat Mass Transfer*, 34, 199–207.

[3] Amir Safwan Noor Alhady, Suzairin Md. Seri (2023) Convective Heat Transfer from Tube Banks of 8 Rows with In-Lined Arrangements in Crossflow, *Journal of Complex Flow*, 5(2) 46-50.

[4] R.L.S. Mainardes, R.S. Matos, J.V.C. Vargas, J.C. Ordonez (2013) Pumping power minimization in staggered finned circular and elliptic-tube heat exchangers in turbulent flow, *Exp. Heat Transf.*, 26(4),397–411.

[5] F. B. Ismail, M. Ali, M. Simic, & S. S. Teoa. (2017). Numerical Analysis and Improvement of the Heat Exchanger Designed for Co-Generating Units. *Procedia Comput. Sci*., 112, 2240-2248.

[6] Hamed Tahmooressi, Alibakhsh Kasaeian, Adel Yavarinasa., (2021) Numerical simulation of nanoparticles size/aspect ratio effect on thermal conductivity of nano liquids using lattice Boltzmann method, *Int. Commun. Heat and Mass Transf.*, 120, 105033.

[7] K.N. Shah, et al., (2014) CFD analysis of heat exchanger over a staggered tube bank for different angle arrangement of tube bundles, *Int. J. Eng*. 2, 1–5.

[8] L. Gu, J. Min, X. Wu, L. Yang, (2017) Airside heat transfer and pressure loss characteristics of bare and finned tube heat exchangers used for aero engine cooling considering variable air properties, *Int. J. Heat Mass Transf*. 108, 1839–1849.

[9] Shinya Aiba, Hajime Tsuchida, Terukazu Ota, (1982) Heat Transfer Around Tubes in In-line Tube Banks, *Bulletin of JSME*, 25(204), 919-926

[10] S. Aiba, H. Tsuchida, T. Ota, O. Terukazu, (1982) Heat transfer around tubes in staggered tube banks, , *Bulletin of JSME*, 25 (204), 927–933.

Published by FAZ Publishing
http://www.fazpublishing.com/jcf