JCF

Journal of Complex Flow, Vol. 6 No. 1 (2024) p. 11-15

Journal of Complex Flow

Journal homepage: www.fazpublishing.com/jcf

e-ISSN : 2672-7374

Numerical Study of Convective Heat Transfer from Tube Banks with Staggered Arrangements in Crossflows

Wan Muhammad Azmi Wan Yusuf[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 13 August 2023;
Accepted 26 Nov. 2023;
Available online 15 April. 2024

Abstract: The purpose of this 2-dimensional numerical research of tubes in crossflows in staggered formation is to evaluate the thermal and flow characteristics of crossflow tubular heat exchangers in staggered formation. Prediction of heat transfer and flow around tube banks is critical in many engineering applications. Six tubes with equilateral triangle layouts were built up, with pitch distance-to-diameter ratios ranging from 1.2 to 2.0. Reynolds numbers ranging from 1000 to 100,000 are utilized. The coefficient of performance (*COP*) of the settings was determined in this study. At pitch distance-to-diameter of 1.8 and 2.0, the Reynolds number of 5000 has the maximum value of the coefficient of performance. Pitch distance to diameter 1.2 at Reynolds number 100 000 receives the highest average of Nusselt number value. Then, the Average heat flux at 1.2 pitch distance to diameter has the highest Average heat flux. Lastly, Normal Root Mean Square Error (*NRSME*) has the lowest value at 1.2 pitch distance to diameter, compared with other pitch distance to diameter.

Keywords: Coefficient of Performance, Average Heat Flux, Normal Root Mean Square Error (NRSME), Nusselt Number

1. Introduction

The importance of predicting heat transfer and flow around tube bank cylinders in various areas of engineering and how high-performance heat exchangers have been developed to maximise efficiency in response to the recent energy crisis. It emphasises the importance of defining a crossflow heat exchanger's specific heat transfer properties with a staggered formation to achieve this goal. It also mentions that previous studies have focused on pressure loss and heat transfer through the bank, but there is a lack of detail on the connection between fluid flow behaviours and heat transfer [1-4]. Additionally, there may be a shortage of data on closely packed tube banks, which could be important for improving transportable heat exchangers. The current authors conducted extensive research on the heat transfer and flow around three to four cylinders in a straight line or spaced-apart arrangement subjected to a crossflow of air and found that to learn more about heat transfer in tube banks, it was necessary to modify the distance of the tube to diameter significantly [5-7].

In a staggered arrangement, the heat transfer parameters at close spacing are like those of a cylinder in a two-dimensional impinging jet. Still, they become like a single cylinder in a uniform flow at wide separation. However, it also notes that the impacts of adjacent cylinders on heat transfer qualities may be substantial in a staggered tube bank made up of many cylinders, especially at tight spacing [8-10]. Study, the research questions, the problem statements, the hypotheses, the objectives, as well as the expected outcome.

Crossflow tubular heat exchangers are used in various devices, including gas-cooled reactors, indirect-fired heaters, air-conditioning coils, waste heat recovery systems, and steam boiler economisers, to mention a few [11]. When constructing or choosing the ideal heat exchanger, the geometry of the tube arrangement is very important because of the flow and heat characteristics of the heat exchanger [12]. A better knowledge of the thermal properties and fluid flow for staggered configuration in the crossflow heat exchanger at various Reynolds numbers is provided by this work.

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

This study was conducted to achieve several objectives: investigate the characteristics of the thermal and flow fields of a staggered crossflow tubular heat exchanger and determine the relationship between flow geometry and the thermal performance of a crossflow tubular heat exchanger with a staggered arrangement. 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 8 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 100 000.

2. Methodology and Geometry Modelling

The method and process for this research are illustrated in Figure 1. The first step involved defining the problem statement, gathering information on relevant parameters from the heat exchanger's tubes, and reducing the complexity of the CAD drawing to focus on the problem. Parameters that are measured and analysed are specified to guide the research. The method also includes using ANSYS-FLUENT software, starting with pre-processing by establishing the structure of the flow system and components to be investigated in Design Modeler. The design is then transmitted to ANSYS-FLUENT to construct the system mesh and set boundary conditions. A calculation solver produces the mesh, and a grid independence test is performed to ensure appropriate meshing and accurate results. The results are verified for validity and accuracy before proceeding to further parameters. The final step is post-processing, where the simulation results are shown in different modes, including a graph analysis of the selected parameters, and used to inform the discussion and conclusion of the research.

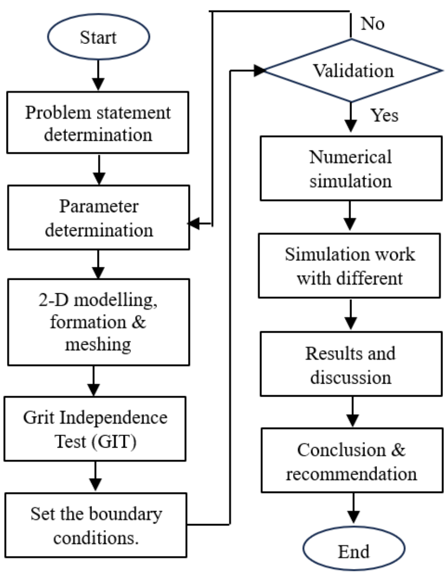

Fig. 1 – Flowchart of the project

2.1 Numerical Validation

A study by Shinya Aiba et al. [13] compared the accuracy of various turbulence models by fixing the pitch distance-to-diameter ratio to $C_y = 1.2d$ and comparing the NRSME values of the models with experimental data. The study found that the standard-k-epsilon model deviated slightly from the study results. In contrast, the results of several other turbulence models were compared to the Local Pressure Coefficient at $Re = 3.0 \times 10^4$.

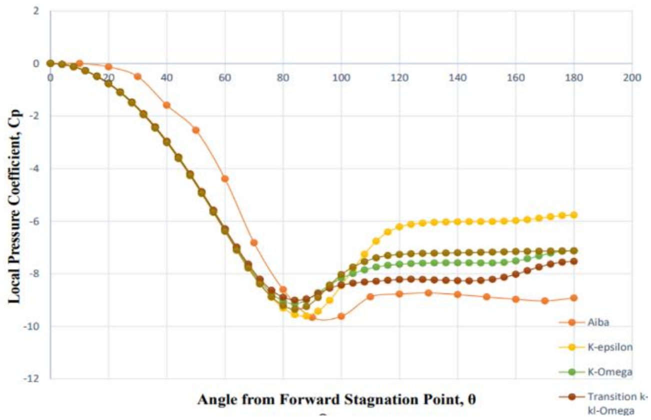

Fig. 2 – Graph of local pressure coefficient, C_p against angle from the forward stagnation point

2.2 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 process forsketchesetry Modeling includes running the Workbench, navigating to the Geometry Cell, changing the analysis type to 2D, using Design Modeler to sketch the *XY* surface, and setting up the units for design. 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).

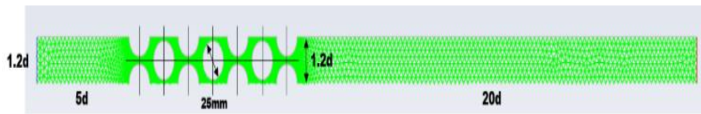

Fig. 3 – Dimension sketching of 2-dimensional model

2.3 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	Grid 1.2
No of elements	62114	20369	15768	12188	10942	9830

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 1.05, respectively.

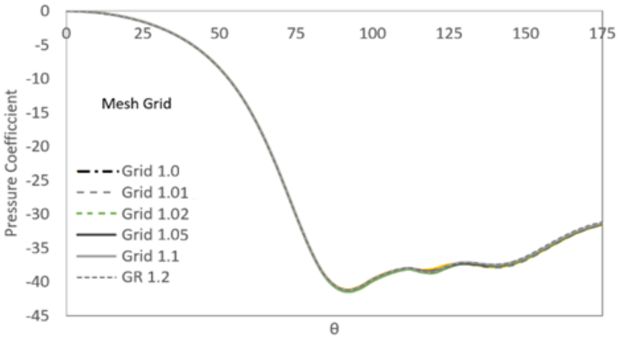

Fig. 4 – Local Pressure Coefficient (C_p) against Angle from Forward Stagnation Point, θ

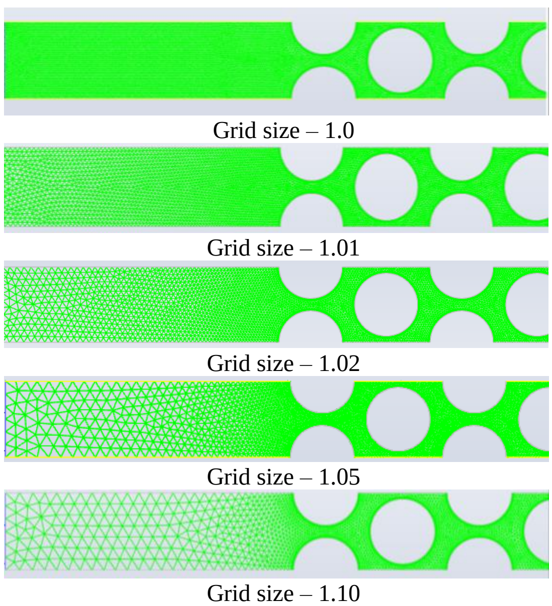

Fig. 5 – Meshing size for the model with different meshing sizes.

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, Grid 1.1, and Grid 1.2 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.

3. Results and Discussion

The summary of data obtained from the simulation is shown in Tables 2 and 3. Table 2 shows the inlet velocity for each ratio used in the simulation, while Table 3 shows the total value for area distribution obtained from the simulation result. The graph of pumping power/area distributions for the 1.2 pitch distance-to-diameter ratio is shown in Figure 6. According to Table 4, the pumping power per area with the highest value is 60356.23 W/m^2. It has a Reynolds number 100000. If the value of the Reynolds number increases, pumping power per area also increases.

Table 2 – Velocity inlet with different Reynolds number

Reynolds Number	1.2 Spacing	1.5 Spacing	1.8 Spacing	2.0 Spacing
1000	9.74E-02	1.90E-05	4.92E-09	1.44E-12
5000	4.87E-01	9.48E-05	2.46E-08	7.19E-12
10 000	9.74E-01	1.90E-04	4.92E-08	1.44E-11
50 000	4.87E+00	9.48E-04	2.46E-07	7.19E-11
100 000	9.74E+00	1.90E-03	4.92E-07	1.44E-10

Table 3 – Total value for Area Distribution

Reynolds Number	1.2 Spacing	1.5 Spacing	1.8 Spacing	2.0 Spacing
1000	0.471143	0.471143	0.471143	0.471143
5000	0.471143	0.471143	0.471143	0.471143
10 000	0.471143	0.471143	0.471143	0.471143
50 000	0.471143	0.471143	0.471143	0.471143
100 000	0.471143	0.471143	0.471143	0.471143

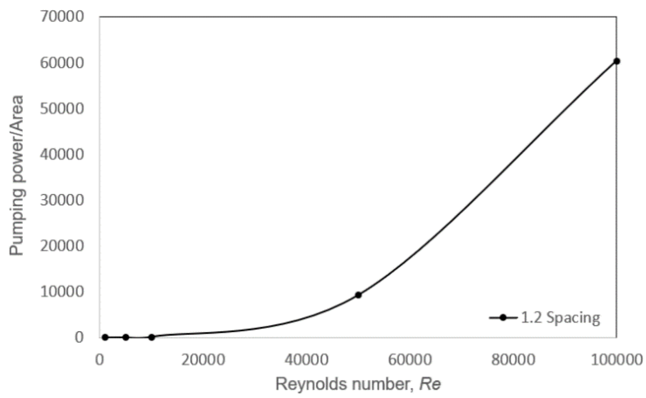

Fig. 6 - Reynolds Number Vs Pumping Power/Area for spacing 1.2d

Table 4 – Total value for Pumping power/Area

Reynolds Number	1.2 Spacing	1.5 Spacing	1.8 Spacing	2.0 Spacing
1000	0.22899	4.09E-10	8.69E-18	4.37E-25
5000	18.17213	1.04E-08	2.3E-16	1.03E-23
10 000	120.4889	4.3E-08	9.35E-16	4.42E-23
50 000	9206.959	2.13E-07	2.39E-14	1.21E-21
100 000	60356.23	4.06E-06	9.56E-14	4.66E-21

Table 5 shows the local pressure coefficient, C_p and stagnation point value, θ, for pitch distance diameter 1.2. The highest value for the local pressure coefficient is at row 6, which is 47.04227. Then, the lowest value for the local pressure coefficient is at the 1st row, which is -0.1104. The data summary is shown in Figure 6 for local pressure coefficient vs stagnation point, θ for pitch distance diameter 1.2.

Table 5 – Total value local pressure coefficient vs stagnation point, θ for pitch distance diameter 1.2

θ	1st Tube Row	2nd Tube Row	3rd Tube Row	4th Tube Row	5th Tube Row	6th Tube Row
0	0	0	0	0	0	0
20	-1.971	-1.882	-1.894	-1.909	-1.881	-0.700
40	-3.481	-3.653	-3.692	-3.733	-3.682	-1.283
60	-9.524	-9.660	-9.634	-9.694	-9.597	-4.846
80	-21.58	-21.29	-21.36	-21.34	-21.32	-16.36
100	-12.44	-13.86	-14.00	-14.03	-13.96	-9.142
120	13.142	7.484	7.286	7.271	7.344	11.965
140	22.646	11.304	10.903	10.876	10.982	15.554
160	25.823	18.544	17.754	17.882	17.896	22.441
180	26.749	42.160	43.026	42.352	42.815	47.042

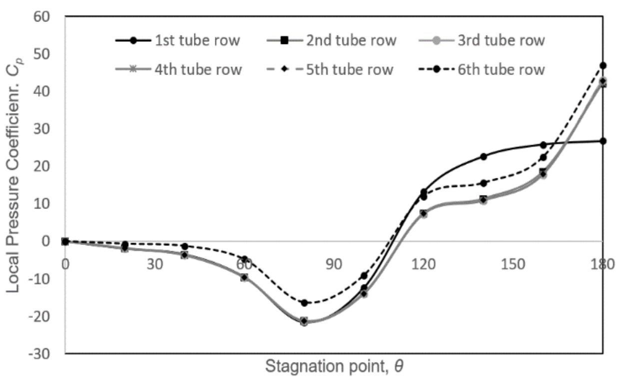

Fig. 6 - Local Pressure Coefficient vs Stagnation point, θ for pitch distance diameter 1.2

Another interesting result obtained from the simulation was an average of the Nusselt number and heat flux. Table 6 shows the data for the Nusselt number for each spacing simulated. The highest value for the Nusselt number value is 1.00576 at 2.0 spacing. Then, the lowest value for the Nusselt number is at 1.8 spacing, which is 0.8524789. Figure 7 shows the plotted data for the average Nusselt versus Reynolds number. The data for average heat flux is shown in Table 7, plotted versus the Reynolds number at spacing 1.2d, as in Figure 8. The highest average heat flux is 28332.18 at Reynolds number 100 000. Then, the lowest average heat flux number is at 1.2 spacing, which is 2.033666 at Reynolds number, which equals 1000.

Table 6 – Nusselt Number, *Nu*

Reynolds Number	1.2 Spacing	1.5 Spacing	1.8 Spacing	2.0 Spacing
1000	6.88014	0.751392	0.852478	0.996510
5000	30.4267	0.844916	0.852478	0.951253
10 000	30.4267	0.844916	0.860888	0.951253
50 000	30.4267	3.42796	0.880510	0.995124
100 000	10468.0	6.87137	0.921807	0.946239

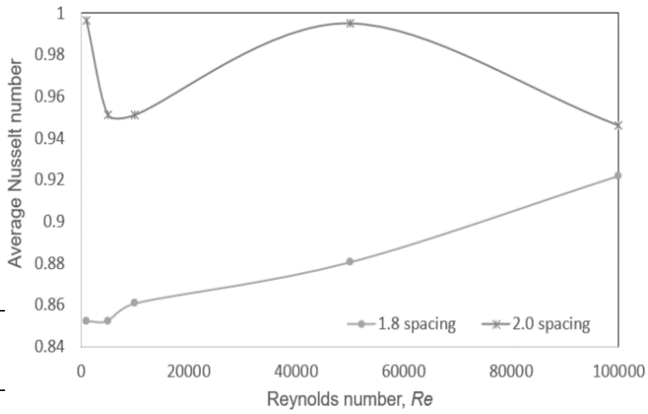

Fig. 7 - Average Nusselt number vs Reynolds number

Table 7 – Avg. heat flux vs Reynolds number, *Re*

Reynolds Number	1.2 Spacing	1.5 Spacing	1.8 Spacing	2.0 Spacing
1000	744.8523	2.033666	2.307258	2.697084
5000	3294.028	2.286791	2.313789	2.722117
10 000	3294.028	2.286791	2.33002	2.574595
50 000	3294.028	9.27787	2.383127	2.693333
100 000	28332.18	18.59757	2.494896	2.561023

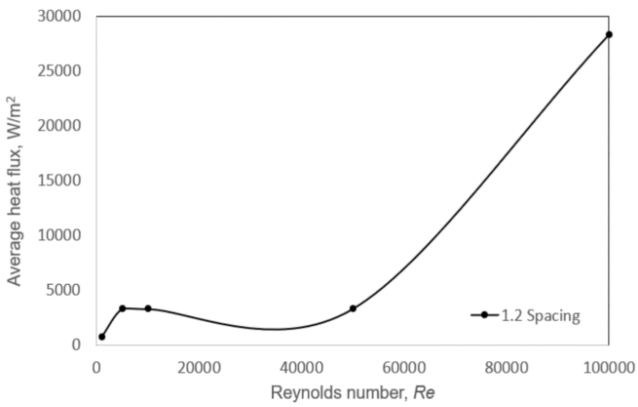

Fig. 8 - Average Heat Flux vs Reynolds Number for spacing 1.2d

Lastly, the values for Normal Root Mean Square Error (NRSME) are shown in Table 7. The highest value for NRSME is 0.68938067 at spacing 1.5 on the 1st row. Then, the lowest value of NRSME is at 1.2 spacing, which is 0.264442802 at the 3rd row. It shows that if the pitch diameter ratio increases, the Normal Root Mean Square value also increases, according to the graph in Figure 9.

Table 7 – Normal Root Mean Square Error (*NRSME*)

Tube row	1.2 Spacing	1.5 Spacing	1.8 Spacing	2.0 Spacing
1st	0.33263	0.68938	0.68190	0.68190
2nd	0.26682	0.60725	0.61410	0.61410
3rd	0.26500	0.60691	0.61380	0.61380
4th	0.26500	0.60691	0.61380	0.61380
5th	0.26473	0.60620	0.60726	0.60726
6th	0.28434	0.56073	0.57581	0.57581

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

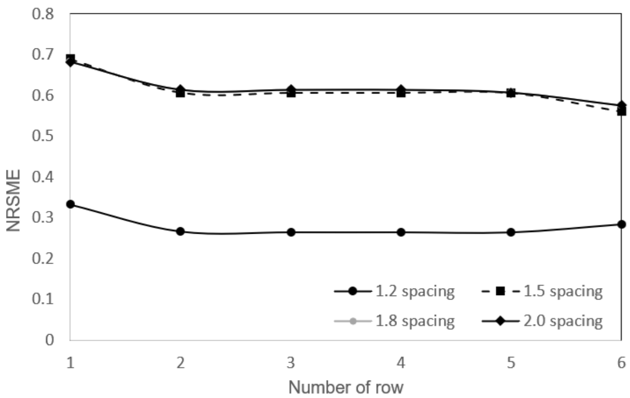

Fig. 9 - NRSME vs pitch distance to diameter

4. Conclusion

A 2-dimensional numerical study explored crossflow tubular heat exchangers' thermal and flow characteristics in a staggered arrangement and determined their overall performance. The study examines the effects of the Reynolds Number, pitch distance-to-diameter ratio, and number of rows of equilateral triangular-arranged cylindrical tubes, with Reynolds numbers between 1000 and 100000, different pitch distances to diameter, and six rows of cylindrical tubes in total. The study uses the Computational Fluid Dynamics (CFD) method with ANSYS Fluent 2022 R1 software to make the prediction process quicker and to obtain sufficient results. The study concludes that the Reynolds Number affects the value of surface heat flux and that ANSYS Fluent is an appropriate method for evaluating the crossflow of a turbulent heat exchanger in a staggered formation, as it has been successfully validated and compared with experimental data. It also finds that the distributions of pressure and velocity are dependent and that the surface heat flux and pumping power per area determine the coefficient of performance (*COP*).

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. The assistance provided, including access to facilities, provision of materials, and analysis of findings, has been instrumental in completing this research paper.

References

[1] Grimison, E. D., “Correlation and utilization of new data on flow resistance and heat transfer for cross flow of gases over tube banks.” *Transactions of the ASME,* 59 (1937): 583–594.

[2] Abdel-Fattah, A., Abou-Taleb, F.S., and Moustafa, G.H., “Behavior of air jet impinging on curved surfaces.” *J. Aerosp. Eng.*, 27(5) (2014): 1 -11

[3] Mangrulkar C. K. Dhoble A. S., Chakrabarty S. G., and Wankhede U. S., “Experimental and CFD prediction of heat transfer and friction factor characteristics in cross flow tube bank with integral splitter plate.” *International Journal of Heat and Mass Transfer,* 104 (2017): 964-978.

[4] Marzuki, M.A.M., Nordin, N., and Rasidi, S.M., “Effect of Inner Wall Curvature on Loss Characteristic and Flow Rectification of Curved Diffuser.” *Journal of Complex Flow,* 4(1) (2022): 1-7

[5] Öztekin, E., Aydin, O., and Avci, M., “Heat transfer in a turbulent slot jet flow impinging on concave surfaces.” *Int. Commun. Heat Mass Transf.*, 44 (2013): 77–82.

[6] Chan, T.L., Leung, C.W., Jambunathan, K., Ashforth-Frost, S., Zhou, Y., and Liu, M.H., “Heat transfer characteristics of a slot jet impinging on a semi-circular convex surface.” *Int. J. Heat Mass Transf.,* 45(5) (2002): 993–1006.

[7] Nuruddin, U.N., and Seri, M.S., “Numerical study of the jet impingement cooling on smooth concave surfaces.” *Journal of Complex Flow,* 3(1) (2021): 26-31

[8] Nurin Kamilia Abdul Kadir and Suzairin Md Seri., “2-Dimensional Numerical Study of Cylindrical Tubes in Crossflows Arranged in Staggered Formation.” *Journal of Complex Flow,* 5(1) (2023): 16-20

[9] Seong-Yeon Yooa , Hwa-Kil Kwonb, Jin-Hyuk Kima, “A study on heat transfer characteristics for staggered tube banks in cross-flow.” *Journal of Mechanical Science and Technology*, 21 (2007): 505~512

[10] E. Baydar and Y. Ozmen, “An experimental and numerical investigation on a confined impinging air jet at high Reynolds numbers,” *Appl. Therm. Eng.*, vol. 25, no. 2–3, pp. 409–421, 2005

[11] Buyruk, E., 2002, “Numerical study of heat transfer characteristics on tandem cylinders, inline and staggered tube banks in cross-flow of air,” *Int. Communications in Heat and Mass Transfer,* 29 (2002): 355-366.

[12] Merker, G. P., Hanke, H., “Heat transfer and pressure drop on the shell-side of tube-banks having oval-shaped tubes,” *Int. J. of Heat and Mass Transfer,* 29 (1986): 1903-1909

[13] Aiba, S.S., Ota, T., and Tsuchida, H., “Heat transfer of tubes closely spaced in an in-line bank.” *Int. J. Heat Mass Transfer,* 23 (1980): 311-319