Journal of Emerging Technologies and Industrial Applications
Vol. 3 No. 2 (2024) pp. 1-5
e-ISSN: 2948-507X
© Malaysia Board of Technologists (MBOT)
Journal homepage: https://jetia.mbot.org.my/index.php/jetia/index

Convective Heat Transfer from Tube Banks of 8 Rows with Staggered Arrangements in Crossflow

Ahmad Fikri Adli Zol Amri[1], Suzairin Md Seri[2,a*]

[1]Faculty of Mechanical Engineering and Manufacturing,
Universiti Tun Hussein Onn Malaysia, Johor, 86400, MALAYSIA

[2]Flow Analysis, Simulation, and Turbulence Research Group (FASTREG),
Faculty of Mechanical and Manufacturing Engineering,
Universiti Tun Hussein Onn Malaysia, MALAYSIA
Email: [a]suzairin@uthm.edu.my,

Abstract: This paper is studying about the ideal geometry of a crossflow heat exchanger under different ratio of pitch distances, by doing simulations on Computational Fluid Dynamics, CFD to observe the fluid flow characteristics and thermal characteristics. The aim of this study is to find a relationship between the thermal performance of crossflow tubular heat exchangers in staggered configurations and the flow geometry. The performance of heat exchanger are measured by how much heat being transferred. To increase the rate of heat transfer of the heat exchanger, one can simply increase the surface area of the fluid contact which in this case the surface area of a cylindrical tube. Those characteristics are determined by applying a number of varying inputs which consist of different Reynolds number, velocity and temperature in the CFD domain. The pressure coefficient, Cp, Nusselt number, Nu, and the efficiency are the main output data that will be used to validate and correlate this study. the efficiency of the crossflow heat exchanger will decrease when the ratio of the pitch distance increased. Besides that, the higher the Reynolds number, the lower the efficiency of the crossflow heat exchanger.

Copyright © 2024 MBOT Publishing.
All right reserved.

Received 10 June 2024; Accepted 18 September 2024; Available online 28 December 2024

Keywords: Crossflow Heat Exchanger, Computational Fluid Dynamics, CFD

*Corresponding Author:

Suzairin Md Seri,
Faculty of Mechanical Engineering and Manufacturing,
Universiti Tun Hussein Onn Malaysia, Johor, 86400, MALAYSIA
Email:

1. Introduction

A heat exchanger is a device that transfers heat from one working fluid to another. Both cooling and heating operations employ heat exchangers. Nowadays, one of the most important study areas is the lowering of energy costs. The most important characteristics are enhancing the heat exchanger's performance to raise the heat capacity while lowering the pressure drop (Ahmed M. Nagib Elmekawy, 2020). They are widely utilized in sewage treatment, natural gas processing, power plants, chemical and petrochemical plants, air conditioning, refrigeration, and space heating.

A significant number of papers have already examined the thermal-hydraulic performance of gas flow across tube banks experimentally, analytically, and computationally. Its extensive use in everything from industrial operations to home heating and cooling demonstrates the interest. However, all those papers and publications are limited to their own ranges and scopes. The scholars create the study under a multiple design of heat exchanger with different scopes to provide a development to an existing heat exchanger that are used in industries. According to a study made by Daniel Bacellar, 2016, three major holes have not yet been filled despite the substantial number of publications. First off, there aren't many design variables in the studied design areas, and they have a small range. The majority of research take fixed tube rows and/or tube pitch ratios into account. Second, studies are often restricted to straightforward parametric analyses of fluid flow. Third, only the topological factors (pitch ratios) are taken into account in addition to Reynolds numbers in all studies that address the tube banks problem.

The aim of this study is to find a relationship between the thermal performance of crossflow tubular heat exchangers in staggered configurations and the flow geometry. The performance of heat exchanger are measured by how much heat being transferred. To increase the rate of heat transfer of the heat exchanger, one can simply increase the surface area of the fluid contact which in this case the surface area of a cylindrical tube. This can be proven by the formula of rate of heat transfer where the surface area is directly proportional with the rate of heat transfer. However, with the fixed size of the fluid duct, increasing the surface area of the cylindrical tube will restrict the fluid flow, which means the heat transfer performance are closely related with the flow characteristics of the fluid.

2. Methodology

For this study, the method that are going to used are in analytical study which is CFD analysis. 2D simulation will be created in a software called ANSYS Fluent, one of the industry's top fluid simulation programme, renowned for its superior accuracy and extensive physics modelling capabilities.

2.1 Flowchart

The preparation and methodology for this investigation are depicted in the flowchart below. Pre-processing, problem-solving, and post-processing are the three primary steps of this flowchart that need to be taken into account. This will serve as a roadmap for doing the simulation research correctly.

2.2 Governing Equations

For fluids with constant density, the fundamental laws of fluid mechanics state that mass must be preserved within a control volume. In order for the total mass entering the control volume to equal the total mass exiting the control volume plus the mass accumulating within the control volume, it must also not exceed it. Below is the derived equation of conservation of mass also known as the continuity equation.

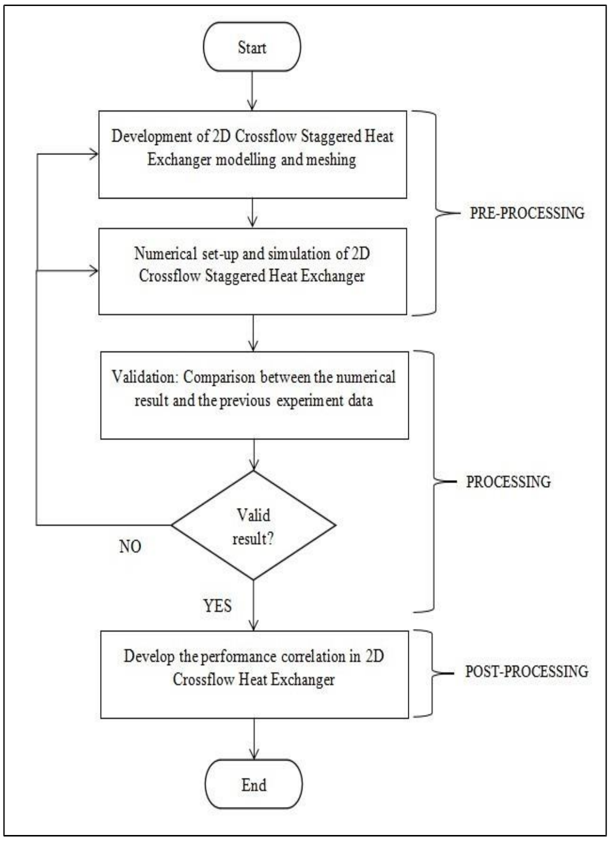

Fig. 1 - CFD Analysis Flowchart

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial x} + \frac{\partial(\rho w)}{\partial x} = 0$$

$$\frac{\partial \rho}{\partial t} + \vec{\nabla} \cdot (\rho \vec{V}) = 0 \qquad (1)$$

For fluids with constant density, the fundamental laws of fluid mechanics state that mass must be preserved within a control volume. In order for the total mass entering the control volume to equal the total mass exiting the control volume plus the mass accumulating within the control volume, it must also not exceed it. Below is the derived equation of conservation of mass also known as the continuity equation.

$$\rho \left[\frac{\partial \vec{V}}{\partial t} + (\vec{V} \cdot \nabla)\vec{V} \right] = -\nabla p + \mu \nabla^2 \vec{V} + \rho \vec{f}$$

$$\frac{\partial \vec{V}}{\partial t} + (\vec{V} \cdot \nabla)\vec{V} = -\frac{1}{\rho}\nabla p + \nu \nabla^2 \vec{V} + \vec{f} \qquad (2)$$

The conservation of energy equation must be taken into account in addition to the conservation of mass equation in order to investigate the temperature distribution in an incompressible fluid flow. The governing equation in this scenario is sometimes referred to as the heat conduction equation or the energy equation.

$$\rho c_p \left[\frac{\partial T}{\partial t} + (\vec{V} \cdot \nabla) T\right] = k\nabla^2 T + Q \qquad (3)$$

2.3 Geometry Model

This model is built on the scope that has been set in the beginning of the study. It has 8 rows of cylindrical tubes with staggered arrangements, diameter of 25 mm each tube, inlet distance 125 mm, outlet distance, 500 mm, and the distance between the cylindrical tubes are based on the pitch distance ratio which is 1.2, 1.5, 1.8, and 2.0. Figure below shows the geometry model for pitch ratio 1.2.

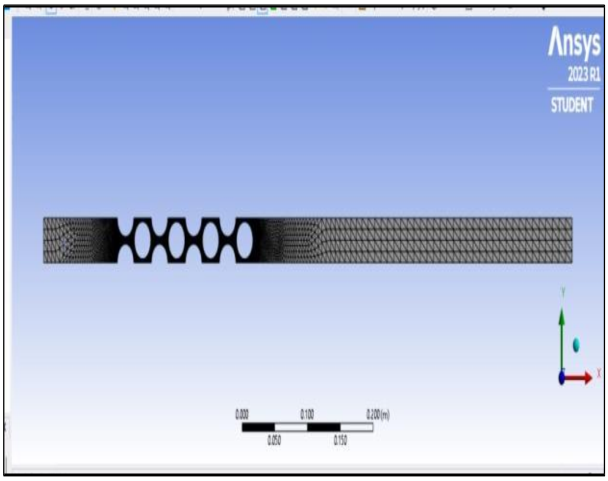

Fig. 2 - Design of the Geometry

2.4 Setup

For the boundary conditions, the pressure at the outlet are considered as pressure gauge, which means the value of pressure is equal to atmospheric pressure, Patm. As for the velocity inlet, Vinlet at, the value was determined by using the Reynolds number that we are going to study. Based on the experimental paper, the Reynolds number defined in the paper are defined at the minimum clearance for the flow. Therefore, we need to define the Reynolds number first before acquiring the velocity of the flow at the inlet.

$$Re = \frac{\rho U_t D}{\mu} \qquad (4)$$

$$U_t = \frac{Re\mu}{\rho D} \qquad (5)$$

$$U_t = \frac{U_\infty C_y}{C_y - D} \qquad (6)$$

$$\frac{Re\mu}{\rho D} = \frac{U_\infty C_y}{C_y - D}$$

$$U_\infty = \frac{Re\mu(C_y - D)}{\rho D C_y} \qquad (7)$$

U_t = Velocity at the minimum clearance, ms^{-1}
U_∞ = Free stream velocity/Inlet velocity, ms^{-1}
D = Diameter of cylinder, m
C_y = Transverse distance, m

Table 1 - Velocity Inlet for Different Reynolds Number

No.	Re	Ratio 1.2	Ratio 1.5	Ratio 1.8	Ratio 2.0
1	5000	0.4868	0.9736	1.2981	1.4604
2	10000	0.9736	1.9472	2.5963	2.9208
3	15000	1.4604	2.9208	3.8944	4.3812
4	20000	1.9472	3.8944	5.1926	5.8416
5	30000	2.9208	5.8416	7.7888	8.7624

3. Results and Discussion

This chapter will be reviewing the acquired data from the simulation performed by comparing and analyzing the differences existed with the experimental data and previous study reference. The results does not have to be exactly the same with the other results but instead, it should have some similarities such as the graph pattern or the relationship between the controlled and manipulated variables. The difference in the procedure and method of study should also be considered since it may be a factor that affects the validity of the results.

3.1 Results

To describe the pressure field under a various situation, engineers or researchers often use a dimensionless form of pressure instead which is pressure coefficient, Cp where the equation are stated below

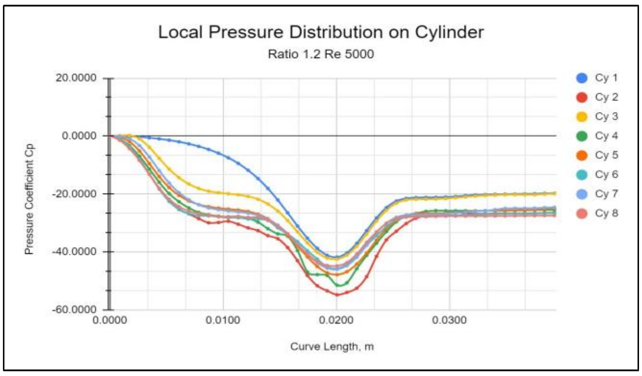

Fig. 3 - Local Pressure Distribution for Ratio 1.2

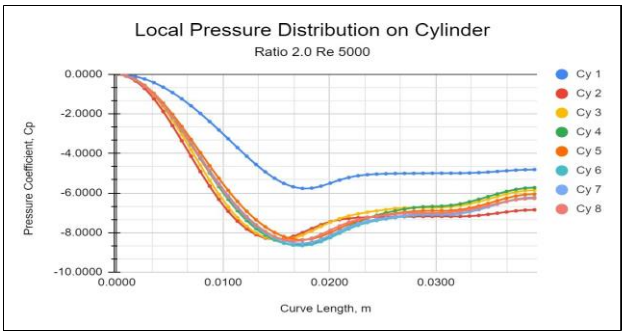

Fig. 4 - Local Pressure Distribution for Ratio 2.0

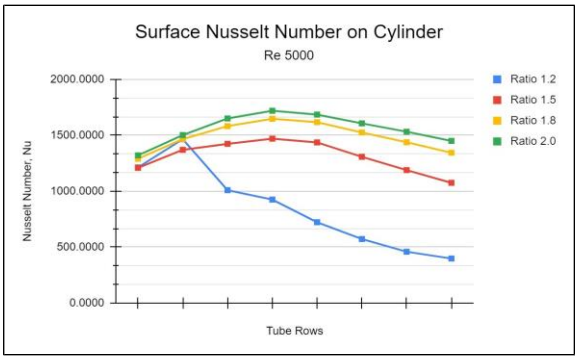

Fig. 5 - Graph of Nusselt Number for Re 5000

In further analysis, we also conduct an average Nusselt number studies acting on the cylinder for each ratio to study the thermal characteristics. The results shows the same as with the fluid flow characteristics, where ratio 1.2 has a very significance difference of the graph pattern compared to the other ratio. Meanwhile, ratio 1.5, 1.8, and 2.0 has a similar projectile pattern with only the difference of Nusselt number reading differs them three.

3.2 Discussions

In a question of which geometry ratios are the best, a study need be conducted to analyse the efficiency for each ratio under different Reynolds number. In order to do that, the formula for the efficiency of crossflow heat exchanger are derived as follow.

$$\frac{Pumping\ Power}{Area} \quad P_{inlet} - P_{outlet})V_{inlet} \qquad (8)$$

$$Heat\ Flux, Q = -k\frac{dT(x)}{dx} \qquad (9)$$

$$Efficiency, \eta = \frac{Average\ Heat\ Flux}{Power\ Pump/Area} \times 100 \qquad (10)$$

Pumping power over area can be calculated with the inlet pressure computed from the simulation software. Heat flux can be directly extracted from the simulation report. Thus a set of efficiency data for each ratio under 5 different Reynolds number can be carried out.

Table 2 - Efficiency of Heat exchanger for Each Ratio

No.	Re	Ratio 1.2	Ratio 1.5	Ratio 1.8	Ratio 2.0
1	5000	24377.8460	19738.0383	17855.4909	17313.1543
2	10000	5080.9710	4368.9723	3829.4870	3647.7210
3	15000	2127.0516	1001.3200	1561.6942	1495.7654
4	20000	1138.0649	1001.3200	848.2241	809.0225
5	30000	528.7036	435.8680	370.1687	351.2426

4. Conclusion

From the analysis of crossflow heat exchanger efficiency based on different ratios of pitch distance, we can say that the geometry ratio of 1.2 has the most efficient for all Reynolds number. Meanwhile, the last ratio which is ratio 2.0 are the least efficient among all the other geometries. In addition, higher Reynolds number was more likely to have lower efficiency.

The outcome of the study seems admissible because, the lower the velocity of the fluid, the more time for the heat to be transferred trough convection. Plus, the smaller ratio of the pitch distance will give much less area of clearance which eventually slow down the fluid flow and more fluid particles will come in contact directly with the tube cylinder.

To conclude, the efficiency of the crossflow heat exchanger will decrease when the ratio of the pitch distance increased. Besides that, the higher the Reynolds number, the lower the efficiency of the crossflow heat exchanger.

Acknowledgement

The authors also would like to thank the Ministry of Higher Education, Malaysia and Universiti Tun Hussein Onn Malaysia for supporting this research under Fundamental Research Grant Scheme (FRGS) and Internal Grant (Vot Q382), respectively.

References

[1] Bacellar, D., Aute, V., Huang, Z., & Radermacher, R. (2016). Airside friction and heat transfer characteristics for staggered tube bundle in crossflow configuration with diameters from 0.5 mm to 2.0 mm. International Journal of Heat and Mass Transfer, 98, 448–454. https://doi.org/10.1016/j.ijheatmasstransfer.2016.02.072

[2] Khan, W. A., Culham, J. R., & Yovanovich, M. M. (2006). Convection heat transfer from tube banks in crossflow: Analytical approach. International Journal of Heat and Mass Transfer, 49(25-26), 4831–4838. https://doi.org/10.1016/j.ijheatmasstransfer.2006.05.042

[3] Mangrulkar, C. K., Dhoble, A. S., Abraham, J. D., & Chamoli, S. (2020). Experimental and numerical investigations for effect of longitudinal splitter plate configuration for thermal- hydraulic performance of staggered tube bank. International Journal of Heat and Mass Transfer, 161, 120280. https://doi.org/10.1016/j.ijheatmasstransfer.2020.120280

[4] Mangrulkar, C. K., Dhoble, A. S., Chakrabarty, S. G., & Wankhede, U. S. (2017). 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, 964–978. https://doi.org/10.1016/j.ijheatmasstransfer.2016.09.013

[5] Mangrulkar, C. K., Dhoble, A. S., Deshmukh, A. R., & Mandavgane, S. A. (2017). Numerical investigation of heat transfer and friction factor characteristics from in-line cam shaped tube bank in crossflow. Applied Thermal Engineering, 110, 521–538. https://doi.org/10.1016/j.applthermaleng.2016.08.174

[6] Nagib Elmekawy, A., Ibrahim, A., Al-Ali, S., Shahin, A., & Al, S. (2020). Two-Dimensional CFD Study on the Enhancement of Tube Bank Configuration Heat Exchanger Performance enhancement for tube bank staggered configuration heat exchanger -CFD Study View project PhD Thesis on energy harvesting using piezoelectric materials View project two- dimensional cfd study on the enhancement of tube bank configuration heat exchanger. Two-Dimensional CFD Study on the Enhancement of Tube Bank Configuration Heat Exchanger. https://doi.org/10.1115/FEDSM2020-20416

[7] Shah, K., Patel, S., Priyank, I., Sardar, P., Patel, V., Kush, I., Sardar, M., Chetan, I., Yadav, O., & Vallabhbhai, S. (2013). Cfd Analysis Of Heat Exchanger Over A Staggered Tube Bank For Different Angle Arrangement Of Tube Bundles.

[8] Wang, Y., Gu, X., Jin, Z., & Wang, K. (2016). Characteristics of heat transfer for tube banks in crossflow and its relation with that in shell-and-tube heat exchangers. International Journal of Heat and Mass Transfer, 93, 584–594. https://doi.org/10.1016/j.ijheatmasstransfer.2015.10.018

[9] Yilmaz, A., Erdinç, M. T., & Yilmaz, T. (2017). Optimization of Crossflow Staggered Tube Banks for Prescribed Pressure Loss and Effectiveness. Journal of Thermophysics and Heat Transfer, 31(4), 878–888. https://doi.org/10.2514/1.t5033

[10] Yoo, S.-Y., Kwon, H.-K., & Kim, J.-H. (2007). A study on heat transfer characteristics for staggered tube banks in cross-flow. Journal of Mechanical Science and Technology, 21(3), 505–512. https://doi.org/10.1007/bf02916312