IJCRT.ORG ISSN : 2320-2882

INTERNATIONAL JOURNAL OF CREATIVE RESEARCH THOUGHTS (IJCRT)
An International Open Access, Peer-reviewed, Refereed Journal

A Numerical Analysis on Heat Transfer Enhancement over Leading Edge Wing Tube and Optimization of Inline Pitch

[1]Niravkumar R Bhavsar, [2]Dr Unnati A Joshi, [3]Dr P V Ramana
[1]Research Scholar, [2]Professor, [3]Professor
[1]Mechanical Engineering Department,
[1]Parul University, Gujarat, India.
[2]Mechanical Engineering Department,
[2]Parul University, Gujarat, India.
[3]Mechanical Engineering Department,
[3]Sardar Vallabhbhai Patel Institute of Technology, Vasad, Gujarat, India.

Abstract: The goal of the current numerical analysis is to compare the effectiveness of the unique leading edge wing shape tube cross section to both circular and trailing edge wing shape tube cross sections with equivalent diameter and similar hydrodynamic and thermal boundary conditions. The entire research is numerical analysis using computational fluid dynamics ANSYS Fluent v16. Various inline pitch to tube diameter ratios (p/d) of 1.2 to 1.8 are tested and evaluated for Re from 3500 to 19,000. Additionally, determine the ideal inline tube pitch due to the increased heat transfer rate and decreased pressure drop with increasing inline pitch distance.

***Index Terms* - Fluid flow over noncircular tube, simulation of pressure drop and simulation of heat transfer rate, leading edge wing shape tube.**

I. Introduction

Wide-ranging research opportunities are provided by the improvement of heat transmission in heat exchanges as fluid flows across tubes. The condenser in air conditioning and refrigeration systems is widely used in many home and industrial applications. The radiator is a well-known heat exchanger used in motor vehicles and locomotive engines. The current study takes into account both of these types of heat exchangers; operate similarly, with hot fluid-filled tubes being thrust into the ambient air. By controlling fluid momentum transfer (pressure loss), minimizing wake formation, increasing contact between the tube surface and a heating or cooling fluid flowing inside and over the tube, altering the fluid flow direction, enhancing the fluid flow rate, Jet impingement, inserting additive, using alternative fluids with different physical properties, modification of the heat exchange surface - increased wall roughness, or altering the geometry, it is apt possible to rise heat transfer rate. The goal of the current study is to suggest a range of improved heat transfer with less pressure loss when fluid flows over a leading edge wing shape as compared to the conventional tube geometries. The research articles presented by Jayavel [1], Najla El Gharbi [13] frames the base in the current investigation methodology and its approach, Najla El Gharbi [13], Niravkumar et al. [19] have all influenced the computational approach used here. The use of computational technique ensures time saving and omits very high expenditures incurred in setting up physical experimental set up and performing experiments just modeled to recognize fluid flow behavior and energy exchange.

The two tubes are arranged in tandem in line for the purpose of the current numerical analysis. The circular tube shape is the subject of the numerical study initially, followed by the trailing edge wing shape. For the purpose of validating the current work, their results are compared with previous experimental results and numerical results mentioned in previous research articles. The leading edge wing shape was then materialized for numerical analysis, and it was compared with the circular tube and the trailing edge wing shape tube. Through this study, the authors have seen how leading edge wing shape tubes in shell and tube heat exchangers can be used to achieve improved heat transfer rate and we can predict their use in future heat exchangers.

While reading through many of the earlier research articles, we discovered that the fluid flowing over an elliptical tube offers a higher rate of heat transfer with a smaller pressure drop than the fluid flowing over a circular tube shape; in addition, several studies found that the trailing edge wing shape (also referred to as the cam shape by some authors) offers an even higher rate of heat transfer at a smaller pressure loss than the elliptical tube shape.

II. PROBLEM DEFINITION

Leading edge wing shape tubes have largely been focused, placed in a 3D, incompressible fluid flow domain, a cross flow tube bundle heat exchanger is considered, the tubes are lying inline in the longitudinal cross section, as depicted in Fig. 1. Only a small portion of the domain model forms the tube bank's symmetric boundary condition. A zero angle of attack for wing shape tube that exerts the air velocity.

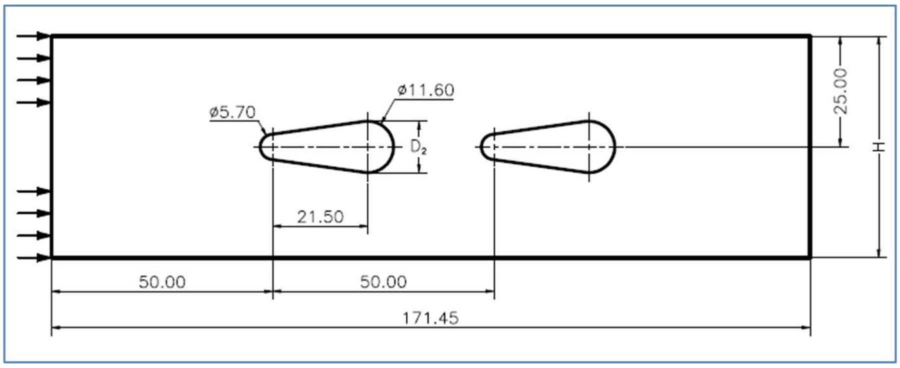

Figure 1. A computational domain of Inline arranged Leading edge wing shape. Dimensions in mm.

The length of the tube in z direction is 30 mm. Later, the numerical results were obtained for the different Inline tube pitch distance and P/D ratio and numerical results were analyzed. Different inline tube pitch distances are shown in Table 1, for those variables the numerical performances are analyzed.

Table 1. Different Inline tubes pitch distance, P/D ratio.

Inline Pitch mm	40	45	50	55	60	65	70
Pitch to Diameter Ratio P/D	1.3	1.5	1.7	1.8	2.0	2.16	2.32

III. CFD NUMERICAL METHOD

3.1 The CFD Numerical Method

As seen in Fig. 3, the boundary conditions at inlet are uniform steady state flow, solely in the x-direction and uniform fluid temperature 300 K. The outer wall surface of the cylinder is set as Tw = 400 K constant temperature and no-slip boundary condition. As a result of a minimal induced buoyancy force, the temperature difference between Tw and T is not very great. As a result, it makes sense to solely take into account forced convection in the formulation used here; the thermophysical characteristics of the flowing fluid are presumptively independent of temperature. The energy equation assumes that the flow is incompressible at low velocities in range of 1 m/s to 5 m/s; therefore viscous dissipation is not taken into account. At symmetric boundaries, a no slip boundary condition is set. At the outlet of the fluid domain, the outflow boundary condition is stated. Using computational fluid dynamics tool, continuity, momentum, and energy equations for incompressible fluids are solved to establish the fluid flow and boundary conditions governing the flow and temperature distribution in the fluid and over the tube surfaces. Air is the fluid on the shell side, and the numerical analysis can vary its velocity. Table 2 displays the material's thermophysical characteristics with regard to air and tubes. The range of Re in this study is 3,500 to 19,000 [8].

Table 2 The Thermo-physical Properties of the fluid flowing over tube and the tube material.

Property	Air-flowing over tube	Tube wall
Density (ρ_a)	1.987 kg/m^3	2700 kg/m^3
Heat Capacity (C_{pa})	1005.91 J/kg K	879 J/kg K
Thermal Conductivity (ka)	2.5849 W/m K	229 W/m K
Temperature (Ta)	300 K	400 K
Viscosity (µa)	1.8275x10-5 m^2/s	-

3.2 Mesh Generation - Discretization, Greed Dependency Test and Data Reduction

Numerical investigations with various mesh densities are carried out to assess the dependence of the numerical results on the grid density. While the number of grids ranges from big to medium grid size, the computational results of the temperature gradient between inlet and outlet are seen with, as little as 0.02% variance.

Table 3 Leading Edge Wing Shape tube: Different Mesh effect on change in Temperature Gradient, change in Pressure Gradient.

Mesh size	Number of Elements	Outlet Temperature (K)	% rise in Temperature Gradient	Inlet Pressure (Pa)	Outlet Pressure (Pa)	% Rise in Pressure drop
Coarse	159490	305.224	0.000	101326.6	101324.08	0
Medium	229125	305.391	0.056	101326.8	101325.03	0.0009
Fine	286288	305.395	0.001	101327.2	101325.24	0.0002

IV. DATA REDUCTION METHOD

4.1 Heat Transfer computation.

Numerical Experimentation data was collected after the steady state is reached. The mean air velocity was calculated by eq. (1). For analysis, Reynolds number for air has been kept varied from 3,500 to 24,000.

$$V_{ai} = \sqrt{2g\left(\frac{\rho_w}{\rho_{a\,f}}\right)\Delta h_{dyn}} \qquad (1)$$

The air side, mean Heat transfer rate is calculated by equation:

$$Q_a = m_a \times C_p\,(Ta_{out} - Ta_{in}) \qquad (2)$$

Heat transfer through condensation and radiation is neglected. Therefore, the overall heat transfer rate:

$$Q = h_a \times A_t \times LMTD \qquad (3)$$

Where, A_t is the tube surface area and LMTD- logarithmic mean temperature difference is calculated by the equation:

$$LMTD = \frac{T_a - T_{out}}{ln\frac{(T_a - T_t)}{(T_{out} - T_t)}} \qquad (4)$$

Air side average heat transfer coefficient is calculated as:

$$h_a = \frac{Q}{A_t\,LMTD} \qquad (5)$$

The local Nusselt Number Nu is calculated by the equation:

$$Nu_a = \frac{h_a D_{eq}}{k_a} \qquad (6)$$

Where, D_{eq} is outer equivalent diameter of the tube.

$$\rho\frac{\partial}{\partial x_j}(u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial u_1}{\partial x_1}\right)\right] + \frac{\partial}{\partial x_j}\left(-\rho\overline{u_i u_j}\right) \qquad (7)$$

$$-\rho\overline{u_i u_j} = \mu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \left(\rho k + \mu_t\frac{\partial u_k}{\partial x_k}\right)\delta_{ij} \qquad (8)$$

4.2 Pressure drop computation

The static pressure difference between the inlet and outlet of the computational domain are calculated by the equation:

$$\triangle p = p_{in} - p_{out} \qquad (9)$$

Having calculated this difference, the influence of the geometry parameters of the inlet air properties on the pressure loss are investigated using Euler number equation as given here, v is the maximum velocity of the fluid,

$$Eu = \Delta p/\rho_a v^2 \qquad (10)$$

V. VALIDATION

The current numerical analysis is carried out on a tube with a circular cross-section, and the results are compared to the Zukauskas's [4,5,10] experimental work explains a strong support for the Zukauskas experimental work done, as depicted in the same image. Numerical optimization of heat exchangers with circular and non-circular shapes is performed in the current numerical analysis on circular shape tubes and trailing edge wing shape tubes, which are compared to numerical investigations carried out by Najla El Gharbi.

VI. RESULTS AND DISCUSSION

The results are shown in Fig. 2, where the leading edge wing shape achieves an outlet temperature that is 1% to 2% higher than the trailing edge wing shape. According to the outlet temperature plot. In order to transmit heat from the tube wall to the cooling medium. The downstream side of the tube wall's upward slope makes it more intimate to be in contact with the increased mass flux of the cooling fluid [16].

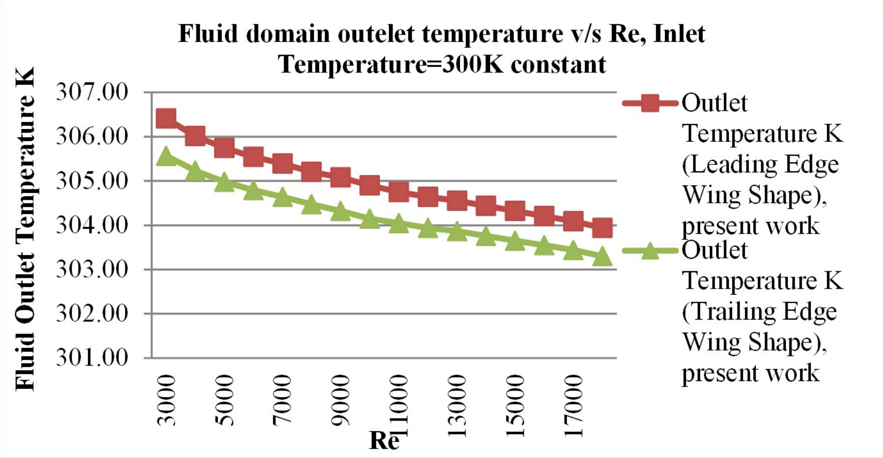

Figure 2 Outlet Temperature K for Leading Edge, Trailing Edge wing tubes.

Fig.3 to Fig.6 uses coloured contours to demonstrate the fluid flow separation phenomena, stream line function, velocity contour, temperature contour and pressure contours for visual perception and analysis purpose [17,18]

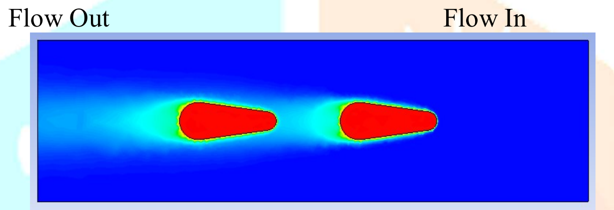

Figure 3 Temperature contour Leading Edge Wing Shape

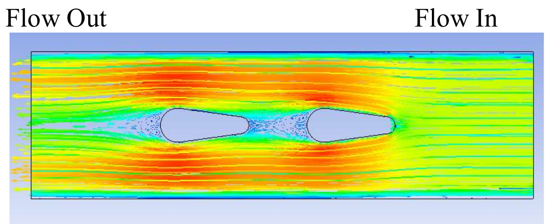

Figure 4 Stream line function Leading Edge Wing Shape

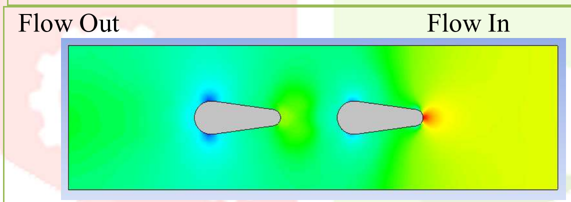

Figure 5 Pressure contour Leading Edge Wing Shape

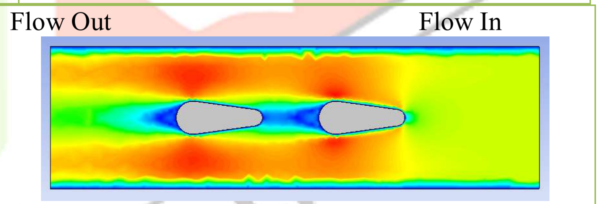

Figure 6 Velocity contour Leading Edge Wing Shape

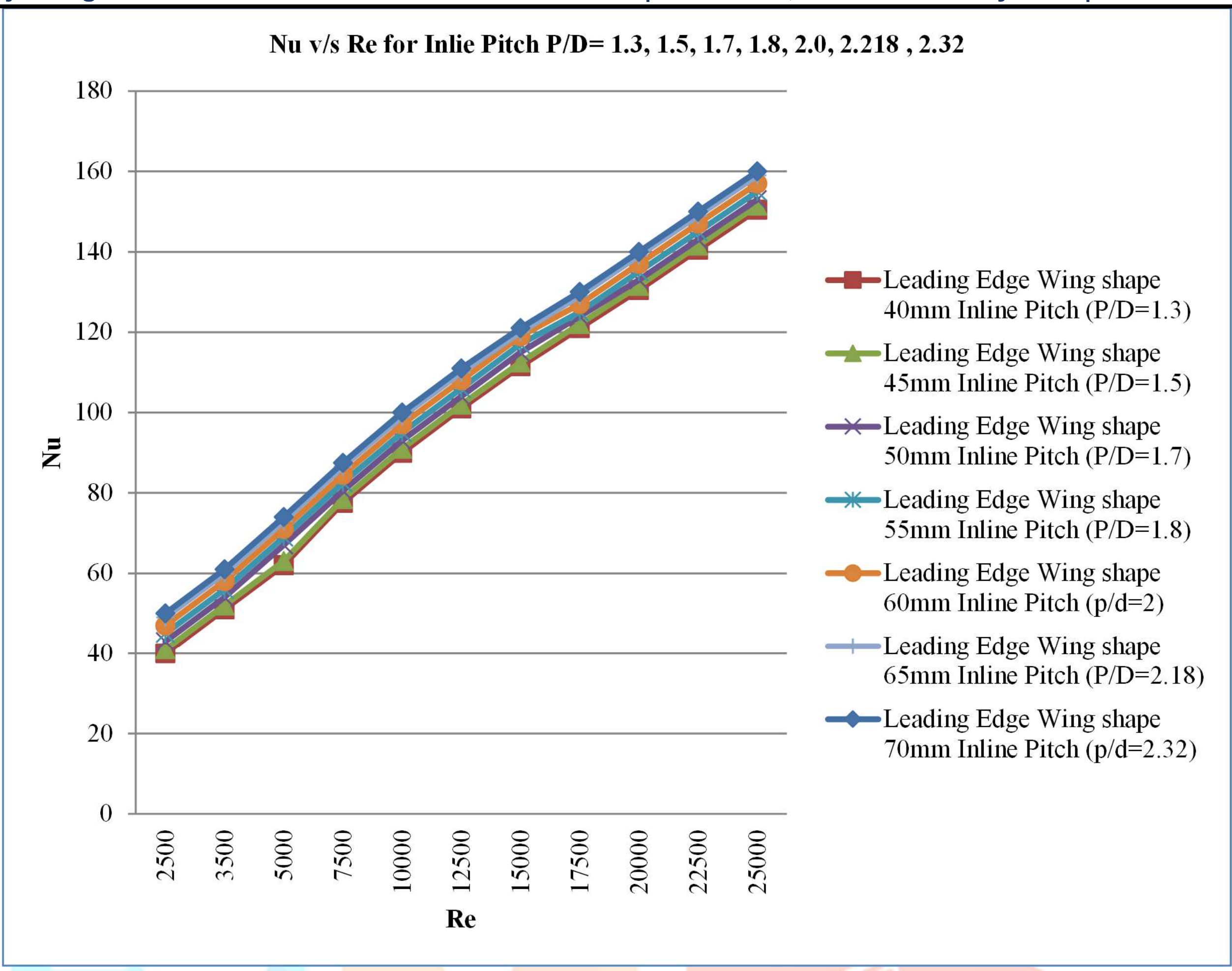

Figure 7. Nu v/s Re for Inlie Pitch

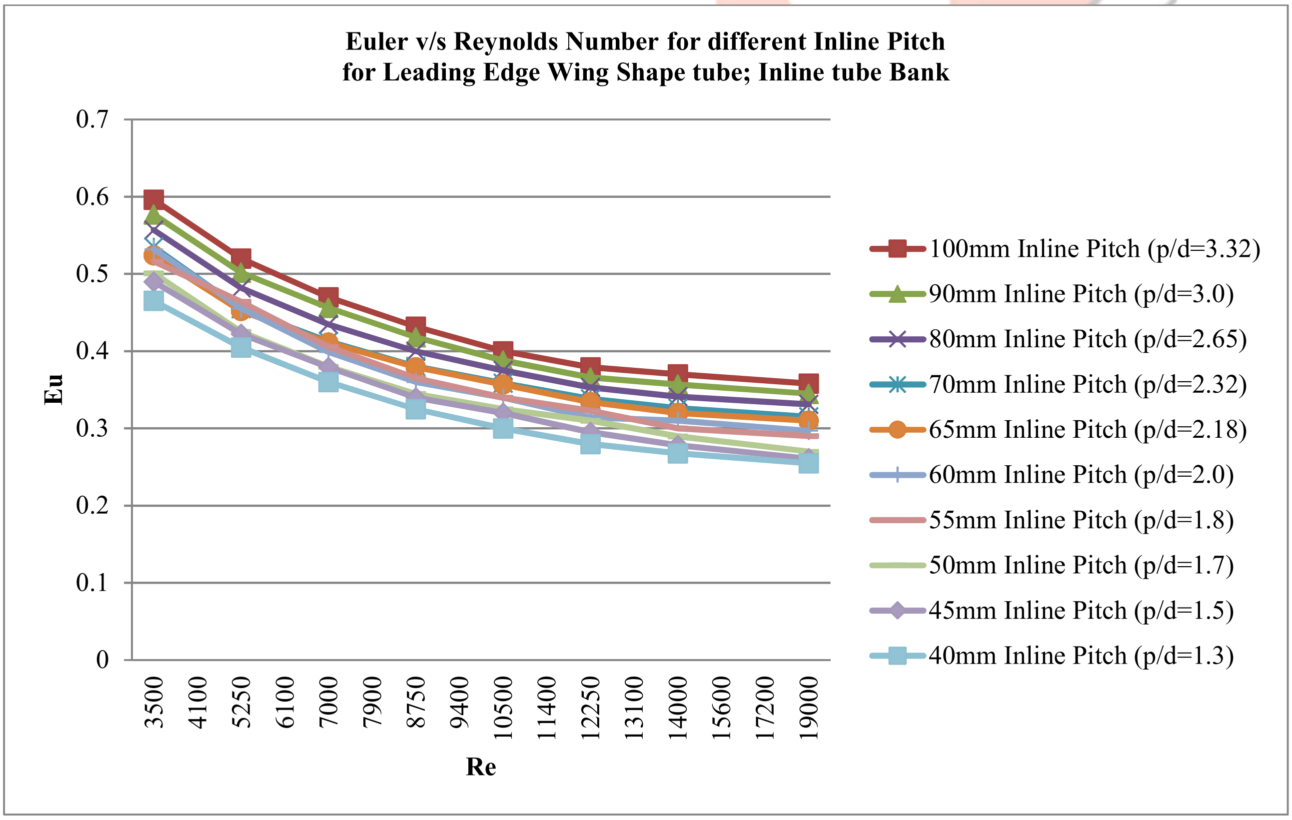

Figure 8. Euler v/s Reynolds Number for different Inline Pitch.

VII RESULT SUMMARY

The following result summary are made in light of the findings of this study and the examination of the impact of the operational parameters:

The tube pitch (the distance between the centres of adjacent tubes) in a shell and tube heat exchanger significantly affects **heat transfer performance, pressure drop, and overall efficiency**. Here's how different inline pitch distances impact these parameters.

The impact of pitch distance on Heat Transfer Rate:

Smaller Pitch (Pt/do = 1.25)

- Increases tube density, leading to a larger **heat transfer surface area** per unit volume.
- Enhances **convection heat transfer** due to the closer proximity of tubes, which can increase turbulence.
- However, if the pitch is too small, it may cause **flow stagnation**, reducing heat transfer effectiveness.

Larger Pitch (Pt/do > 1.5)

- Allows for better shell-side fluid flow distribution, reducing the chances of stagnant zones.
- However, a lower tube density reduces the overall **heat transfer area per unit volume**, which may lower efficiency.

The impact of pitch distance on Pressure Drop:

Smaller Pitch

- Results in **higher pressure drop** on the shell side due to **restricted flow paths** between tubes.
- Increased pressure drop means **higher pumping power requirements** and greater operational costs.

Larger Pitch

- Reduces **pressure drop** since there is more space for fluid movement, leading to smoother flow patterns.
- A lower pressure drop reduces energy consumption but may lead to **less turbulence**, which can lower the heat transfer coefficient.

The impact of pitch distance on Flow Pattern and Fouling

With a smaller tube pitch increases **turbulence** and enhances heat transfer but can make **cleaning difficult**.

With a larger tube pitch allows for **easier mechanical cleaning**, which is beneficial in fouling applications.

Tube Pitch	Heat Transfer Rate	Pressure Drop	Maintenance
Smaller Pitch (Compact Design)	Higher	Higher	More difficult
Larger Pitch (Spacious Design)	Lower	Lower	Easier

VIII. CONCLUSION

In this work, a numerical study on a three-dimensional fluid domain is used to investigate the heat transfer rate and pressure drop characteristics of a cooling medium air in cross flow over an inline tube bank arrangement. If heat transfer enhancement is the priority, a **smaller pitch with increased turbulence** is preferred. If **pressure drop and maintenance are concerns**, a **larger pitch with better flow distribution** is recommended. The observations come to the conclusion that the optimal tube pitch depends on the trade-off between heat transfer efficiency and pressure drop, in this case it is 55mm.

IX. ACKNOWLEDGMENT

I sincerely express my gratitude to **Dr. Unnati A. Joshi** and **Dr. P.V. Ramana** for their invaluable guidance, encouragement, and insightful support throughout this work. Their expertise and continuous motivation have been instrumental in the successful completion of this work.

REFERENCES

[1] S. Jayavel, Shaligram Tiwari, Numerical study of heat transfer and pressure drop for flow past inline and staggered tube bundles, Int. J. Numer. Methods, Heat Fluid Flow 19 (8) (2009) 931–949.

[2] T. Ota, H. Nishiyama, Y. Taoka, Heat transfer and flow around an elliptic cylinder, Int. J. Heat Mass Transf. 27 (1984) 1771–1779.

[3] E.S. Sayed, O.M. Ahmed, A.O.M. Mesalhy, T.M. Khass, A.H. Hassan, Parametric study of air cooling process via water cooled bundle of wing shaped tubes, EIJST 15 (3).

[4] A. Zukauskas, Heat transfer from tubes in cross-flow, Adv. Heat Transf. 8 (1972) 93–160.

[5] A. Zukauskas, R. Ulinskas, Heat Transfer in Tube Banks in Crossflow, Hemisphere, Washington, D.C., 1988.

[6] J.P. Holman, Heat Transfer, 4th ed. McGraw-Hill, Inc., United States of America, 1976.

[7] V. Yakhot, L.M. Smith, The renormalization group, the ε-expansion and derivation of turbulence models, J. Sci. Comput. 7 (1) (1992) 35–61.

[8] Guide, ANSYS Fluent User's, Ansys, Inc. Release 14, 2011.

[9] Cengel,Y. A. and Ghajar, A. J., Heat and Mass Transfer, 4th ed., Tata McGraw Hill.

[10] Zukauskas, A., Ulinskas, R.V., [1985] ”Efficiency parameters of heat transfer in tube banks”, Heat Transfer Engineering, Vol.6, No.1, PP.19-25.

[11] Holman, J.P., [2001] ”Experimental methods for engineers”, seventh ed. McGraw-Hill Int., New York, USA.

[12] A. Lankadasu, S. Vengadesan, Onset of vortex shedding in planar shear flow past a square cylinder, Int. J. Heat Fluid Flow 29 (4) (2008) 1054–1059.

[13] Najla El Gharbi Abdelhamid Kheiric, Momammed El Ganaoui, Ryan Blanchard, “Numerical optimization of heat exchangers with circular and non-circular shapes”, Case Studies in Thermal Engineering 6 (2015) 194–203, Elsvier

[14] Snehal A. Powar, Ashish R. Wankhade, Neelam Gohel, “ Review on Thermal performance of cross flow heat exchanger using non-circular shape of tubes”, IRD India, ISSN: 2319-3182, Volume-4, 2015 print.

[15] Arash Mirabdolah Lavasani, Hamidreza Bayat, Taher Maarefdoost, “Experimental study of convective heat transfer from in-line cam shaped tube bank in crossflow”, Applied Thermal

[16] A.M. Lavasani, H. Bayat, T. Maarefdoost, Experimental study of convective heat transfer from in-line cam shaped tube bank in crossflow, Appl. Therm. Eng. 65 (1) (2014) 85–93.

[17] C.K. Mangrulkar, A.S. Dhoble, A.R. Deshmukh, S.A. Mandavgane, Numerical investigation of heat transfer and friction factor characteristics from in-line cam shaped tube bank in crossflow, Appl. Therm. Eng. 110 (2017) 521–538.

[18] Rawad Deeb, D.V.Sidenkov, Numerical Simulation of the flow past the Three-Raw staggered drop-shaped tubes bundle at the different angle of attack, (2022) DOI: 10.1109/Inforino53888.2022.9782958.

[19] Niravkumar R. Bhavsar, Unnati A. Joshi, P.V.Ramana, CFD Analysis of Heat Transfer Enhancement of Shell Side Fluid Flow Over Inline, Non-circular Leading-Edge Wing Shape Tube, Transactions of the Indian National Academy of Engineering (2022), https://doi.org/10.1007/s41403-022-00375-7.