IJCRT.ORG ISSN : 2320-2882

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

# Heat Transfer Enhancement With Non-Circular Leading Edge Wing Tube In Heat Exchanger, CFD Approach

**[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:*** This numerical analysis has been performed to compare the effectiveness of the unique leading edge wing shape tube cross section with i. a circular and ii. a trailing edge wing shape tube with equivalent hydraulic diameter and has been tested in similar hydrodynamic and thermal boundary conditions. The 3D models geometry using finite volume discretization, tested and evaluated for 3500≥ Re ≥19,000. Later, it is identified, the leading edge wing shape tube cross section results in to enhanced heat transfer rate, enhanced Nusselt number Nu, and concomitant decreased Euler number Eu.

***Key Words* -** Fluid flow over non-circular 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. [19,20] 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. Fluid flow behavior was studied in detail and understood the fluid flow behavior over various tube surfaces, [21].

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 (a, b, c). Only a small portion of the domain model forms the tube bank's symmetric boundary condition with periodic boundary condition at inlet and outlet. A zero angle of attack for wing shape tube that exerts the air velocity [15,16, 19].

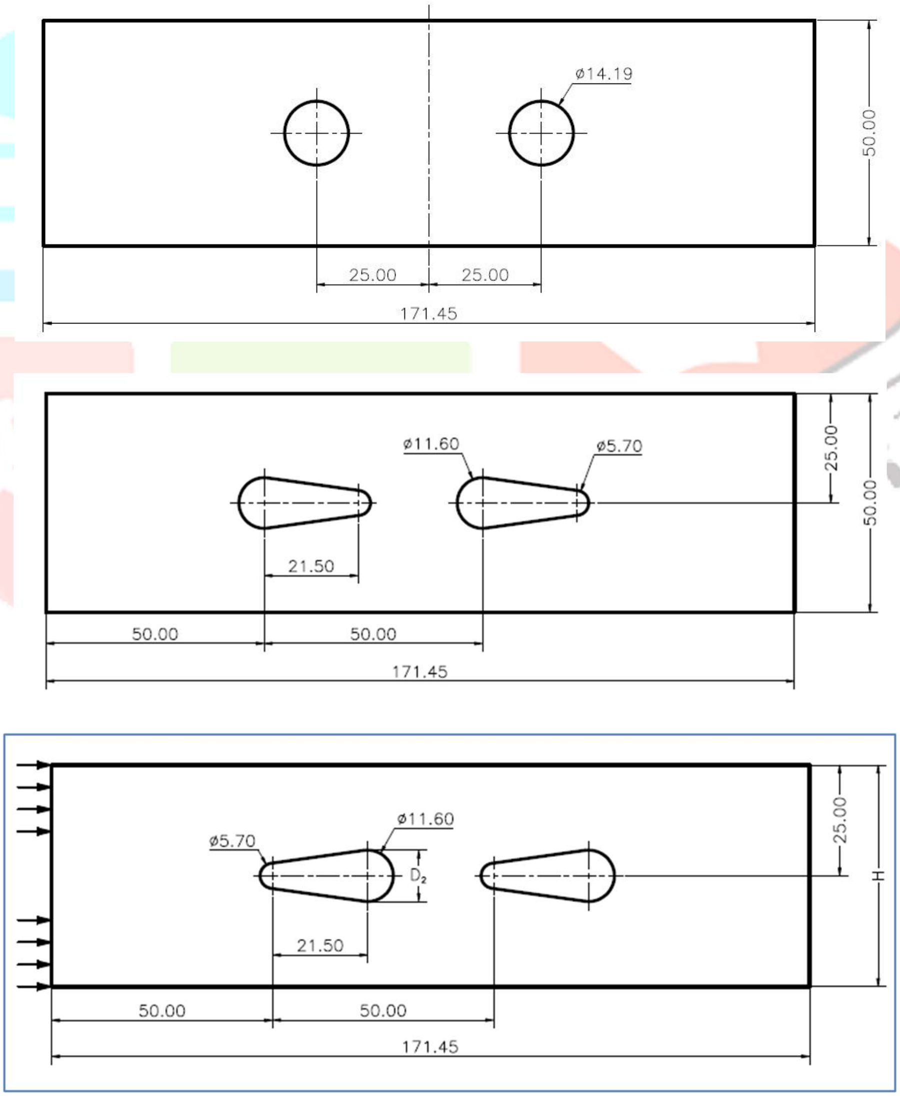


**Figure 1. A computational domain of Inline arranged (a) Circular tube (b) Trailing edge wing shape (c) Leading edge wing shape. All dimensions are in mm. Dimensions in mm.**

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 tube shapes arranged in Inline.

| | Circular Tube | Trailing Edge Wing Shape | Leading Edge Wing Shape (Present work) |
|---|---|---|---|
| Inline Pitch mm | 50 | 50 | 50 |
| Pitch to Diameter Ratio P/D | 1.7 | 1.7 | 1.7 |

## 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 ($\rho_a$) | 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 ($\mu$a) | 1.8275x10-5 m$^2$/s | - |

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 Circular 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 | 53208 | 303.866 | 0.000 | 101324.60 | 101322.97 | 0 |
| Medium | 76101 | 303.934 | 0.022 | 101324.60 | 101322.78 | 0.0002 |
| Fine | 143464 | 303.993 | 0.019 | 101324.60 | 101322.64 | 0.0001 |

# 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.

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

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. Fig. 2, Fig. 3 show the Nu rises as rise in Re and Eu drop with rising Re. A raised Nu is a result of increased turbulence activity of the fluid, improved convective heat transfer as well. The numerical result of Nu acquired from the current study on a trailing edge wing shape tube provides a very good conformity with the previous referred work.

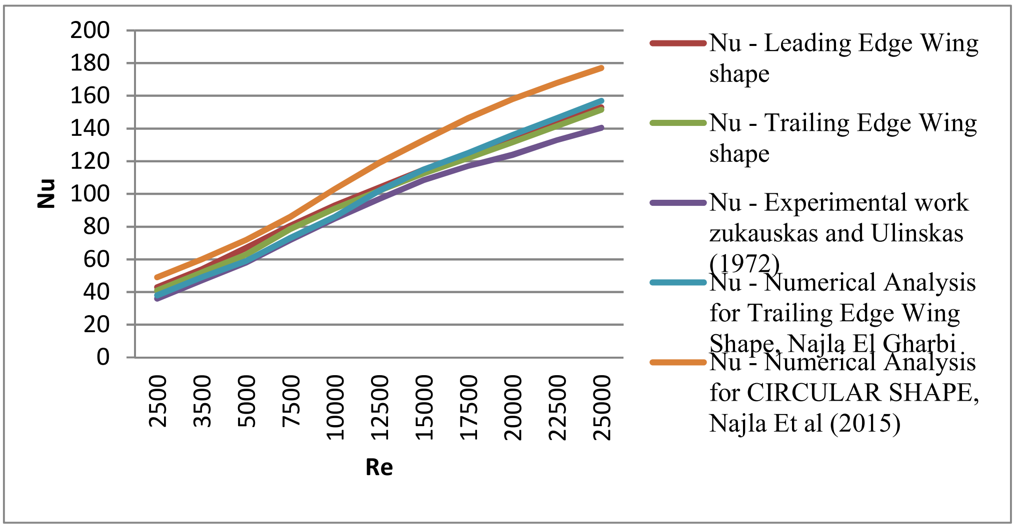


**Figure 2. Validation of the present work on circular tube shape with past research work.**

## VI. RESULTS AND DISCUSSION

The authors' interest in the current study of fluid flow over leading edge wing shape tubes with identical hydraulic diameter of circular tubes and trailing edge wing shape tubes with similar boundary conditions was sparked by the investigation's encouraging and favorable results. As shown in Fig. 3, where the leading edge wing shape achieves an outlet temperature that is 1.5% to 2.5% higher than the trailing edge wing shape. According to the outlet temperature plot, Fig. 5 temperature contour is displayed. In order to transmit heat from the tube wall to the cooling medium, the leading edge tube profile must expose the most of the tube profile among all other shapes, as shown in Fig. 5. 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. The idea of investigating novel ways to use the noncircular wing shape tube geometry that might result in a higher heat transfer rate and a lower pressure drop is based on the physical interaction between the tube wall and the fluid flow on the shell side. Fig. 4 shows that the leading edge wing shape tube offers maximum heat transfer rate as compared to others. Fig. 6, 7, 8, coloured contours to illustrate the phenomenon of fluid flow separation. Comparing all Fig. 6,7,8 reveals that the leading edge wing configuration facilitates the most interaction between fluid particles and the maximum surface area of the tube wall. The fundamental physics of an increased heat transfer rate were established by optimal contact between the tube wall and the fluid. Eu is significantly greater for the circular tube design compared to non-circular tube designs. This occurs because the separation point is significantly earlier than it would be on an ellipse or wing.

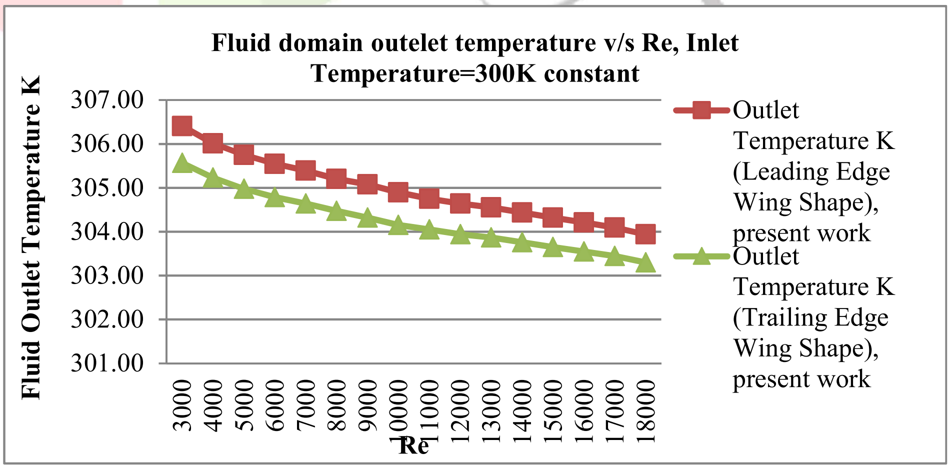


**Figure 3 Outlet Temperature K for Leading Edge, Trailing Edge wing tubes.**

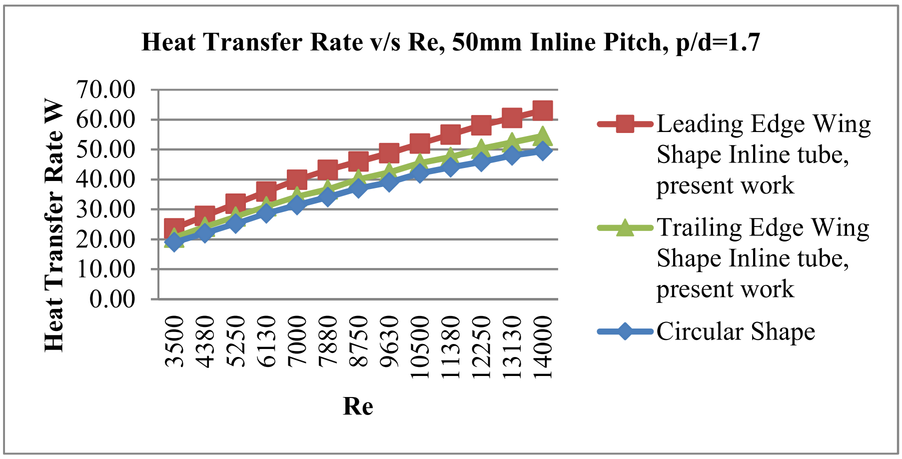


**Figure 4 Heat Transfer rate for Leading Edge, Trailing Edge wing and Circular tubes.**

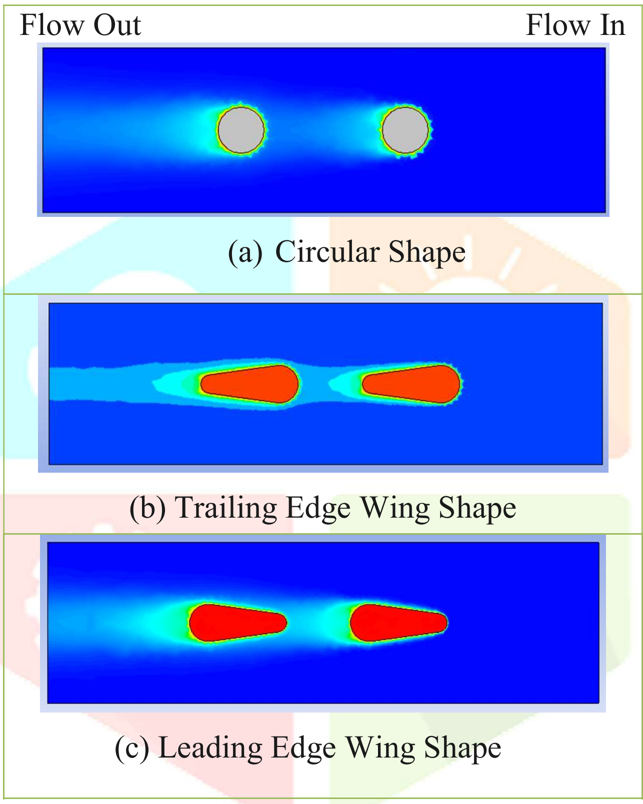


**Figure 5. Temperature contour**

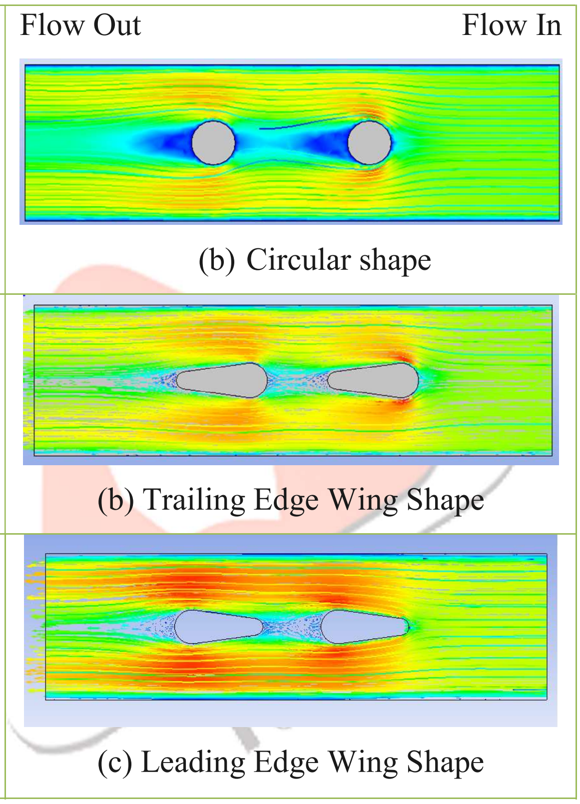


**Figure 6. Stream line function**

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]

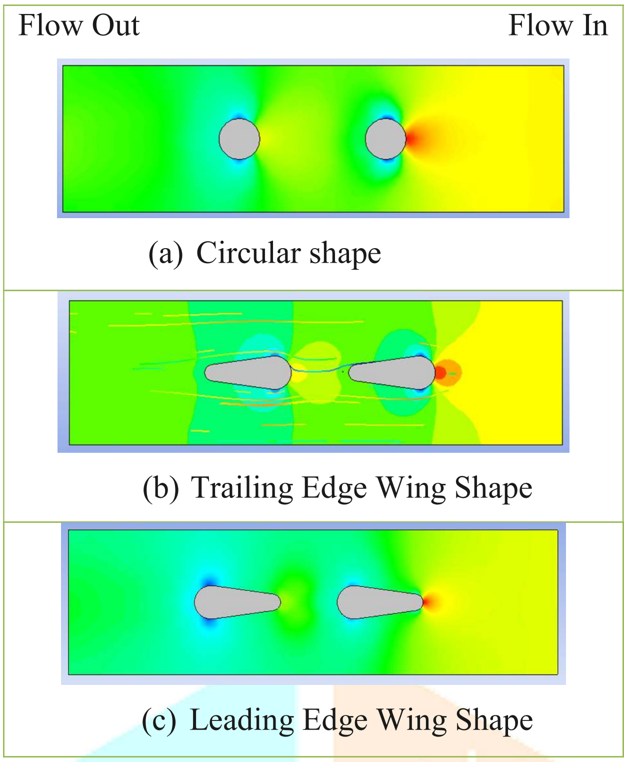


**Figure 7. Pressure contour - fluid flow over tube**

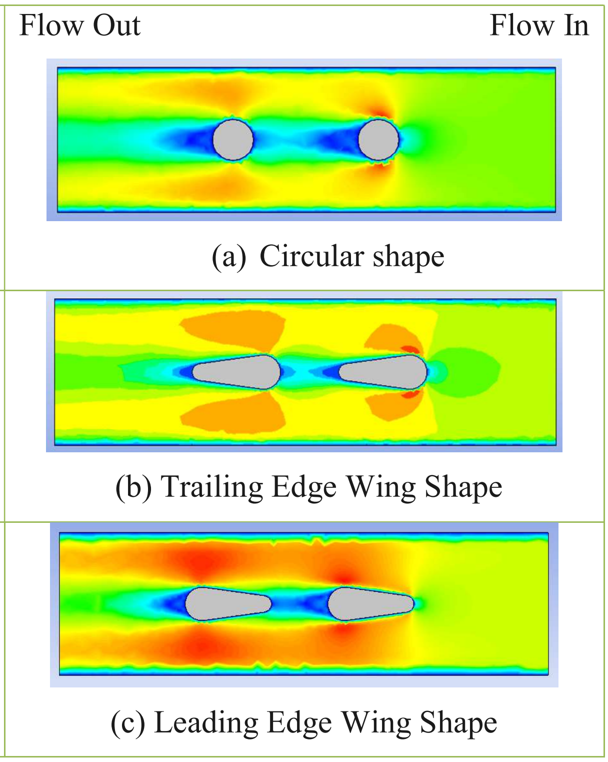


**Figure 8. Velocity contour fluid flow over tube**

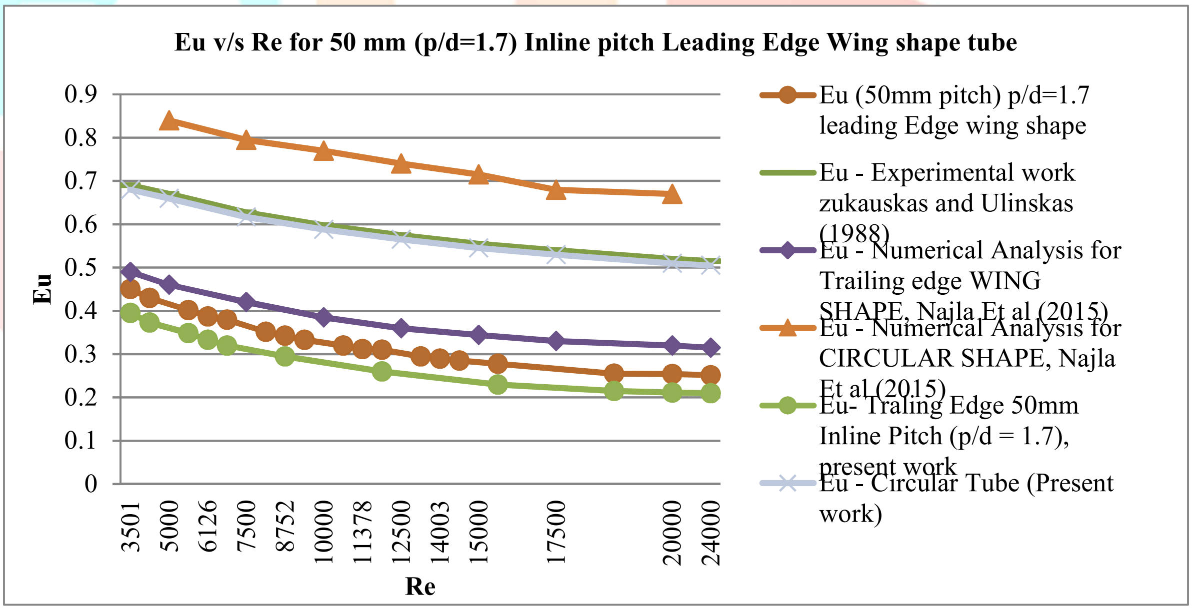


**Figure 9. Eu v/s Re for 50 mm (p/d=1.7) Inline pitch Leading Edge Wing shape tube**

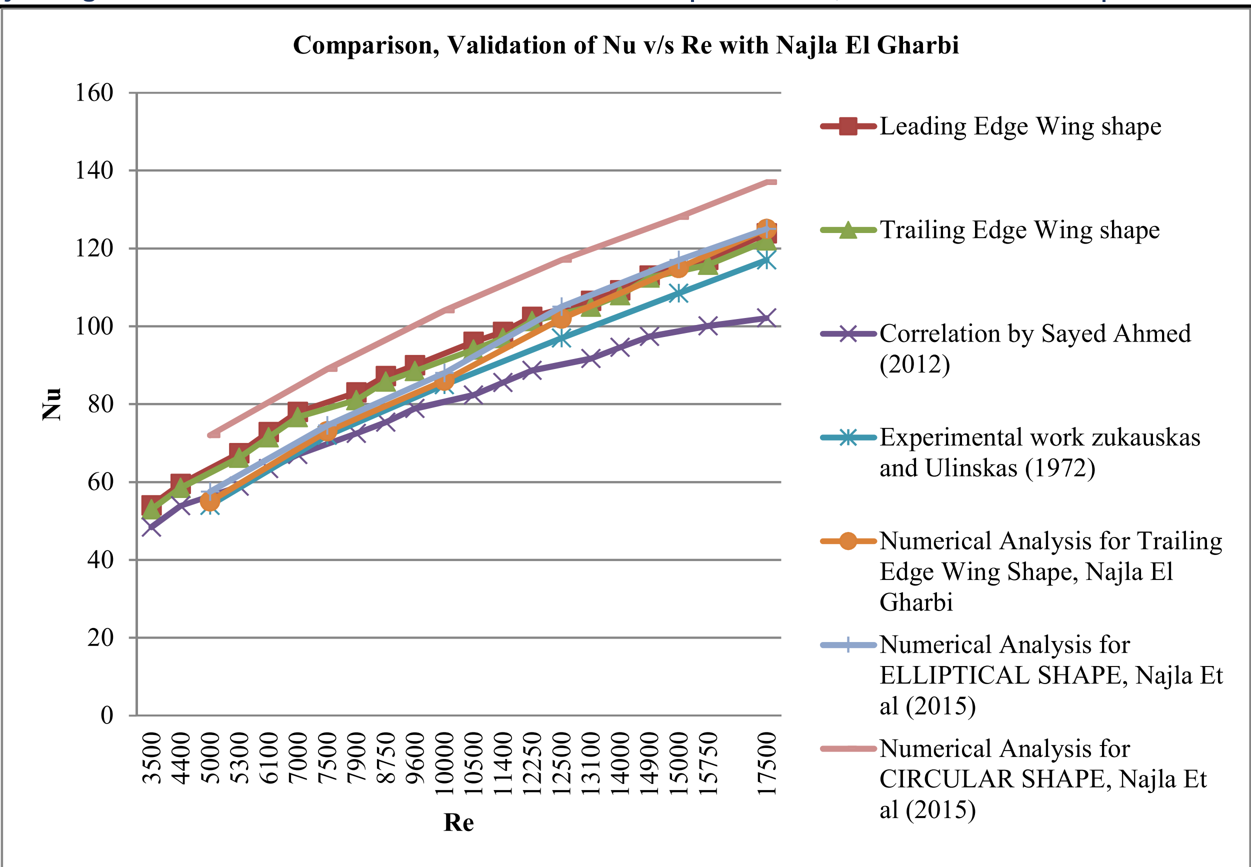


**Figure 10. Comparison, Validation of Nu v/s Re with Najla El Gharbi.**

## VII RESULT SUMMARY

The numerical investigation focused on evaluating the heat transfer efficiency and pressure drop characteristics of three distinct tube geometries: circular, trailing edge wing-shaped, and leading edge wing-shaped tubes. The analysis employed Computational Fluid Dynamics (CFD) simulations to determine the effects of different tube configurations on heat exchanger performance under varying Reynolds numbers (Re) and inline pitch distances (P/D ratios).

### 1.Heat Transfer Enhancement

The leading edge wing tube demonstrated the highest heat transfer rate, surpassing the circular and trailing edge wing tubes. Heat transfer rate was observed to improve by 12-15% in the leading edge wing shape compared to the trailing edge wing shape. The superior performance of the leading edge wing shape is attributed to its optimized geometry, which increases interaction between fluid particles and the tube surface, promoting better heat exchange. The Nusselt number (Nu) showed an increasing trend with Reynolds number across all tube geometries, with the leading edge wing shape consistently achieving the highest values. The study also analyzes the turbulent intensity and its impact on the heat transfer coefficient. The results show that the turbulent intensity increases with Re, leading to enhanced heat transfer due to improved fluid mixing and radial heat transport. The leading edge wing shaped tube exhibits higher turbulent intensity compared to the other geometries, which contributes to its superior heat transfer performance. As shown in Fig. 4 and Fig. 10, the heat transfer coefficient (h) is found to increase with Re, as the increased turbulence enhances the convective heat transfer process. This is particularly evident in the leading edge wing-shaped tube, where the slope of the tube geometry promotes better fluid flow and heat transfer.

### 2. Pressure Drop Characteristics

As shown in Fig. 9 leading edge wing shape exhibited a 3-7% reduction in pressure drop compared to other geometries. The Euler number (Eu) plots indicated that the leading edge wing shape minimized flow separation wake formation, lead to reduced drag forces and improved energy efficiency. Pressure contours Fig. 7and velocity Fig. 8 demonstrated that the leading edge wing shape maintained a smoother pressure gradient and velocity distribution, unlike the abrupt variations seen in circular and trailing edge wing shapes.

### 3. Comparative Analysis of Tube Configurations

Temperature contours Fig. 5 confirmed that the leading edge wing shape maintained a more uniform temperature distribution, improving thermal efficiency. Streamline analysis indicated that the leading edge wing shape reduced recirculation zones, leading to better convective heat transfer. The wake formation behind the leading edge wing shape was notably smaller, reducing energy losses and pressure drop.

## VIII. CONCLUSION

This research focuses on evaluating the thermal and fluid dynamic performance of three distinct tube geometries: circular, trailing edge wing shaped, and compared with present study with leading edge wing shaped tubes. Through advanced Computational Fluid Dynamics (CFD) simulations, the study explores the effect on heat transfer rates and pressure drop characteristics. The findings reveal significant improvements in heat transfer efficiency and energy savings when using leading-edge wing-shaped tubes, making them a promising alternative to traditional tube designs in heat exchanger applications. The leading-edge wing-shaped tube displayed a 12-15% improvement in heat transfer rate compared to the trailing edge wing shaped tube and considerably outperformed the circular tube.

The leading edge wing shape maximizes the surface area for heat exchange and promotes optimal contact between the fluid and the tube wall. According to the findings of the Nusselt number (Nu) investigation, the leading edge wing shaped tube consistently attained higher values across the whole range of Reynolds numbers, which ranged from 3,500 to 19,000, showing that it exhibited superior convective heat transfer. The leading edge wing shaped tube demonstrated a reduction in pressure drop that ranged from 3 to 7 percent when compared to the other tube designs. A decrease in drag forces and energy losses can be attributed to the efficient creation of wakes and the minimizing flow separation, both of which contribute to this reduction. The decreased pressure drop features of the leading edge wing shaped tube were further validated by Euler number plots, which made the tube more energy efficient for applications involving heat exchanger temperature, pressure, and velocity contours illustrated that the leading edge wing shaped tube maintained a more uniform temperature distribution, smoother pressure gradient, and improved velocity profile compared to the circular and trailing-edge wing-shaped tubes. Streamline study demonstrated that the leading-edge wing shape reduced the number of recirculation zones and flow separations, which resulted in improved fluid flow distribution and greater convective heat transfer.

## 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.

**[20]** Niravkumar R. Bhavsar et. al., A Numerical Analysis on Heat Transfer Enhancement over Leading Edge Wing Tube and Optimization of Inline Pitch, International Journal of Creative Research Thoughts (2025), ISSN:2320-2882, pg. g975-g981, DOI: http://doi.one/10.1729/Journal.43818.

**[21]** Fares Djeffal et. al., Numerical Investigation of Thermal-Flow Characteristics in Heat Exchanger with Various Tube Shapes, MDPI Appl. Sci. 2021, 11, 9477. https://doi.org/10.3390/app11209477.