Frontiers in Heat and Mass Transfer (FHMT), 19, 6 (2022)
DOI: 10.5098/hmt.19.6

Global Digital Central
ISSN: 2151-8629

Frontiers in Heat and Mass Transfer

Available at www.ThermalFluidsCentral.org

CFD SIMULATION IN THERMAL-HYDRAULIC ANALYSIS OF AIRFLOW ON DIFFERENT ATTACK ANGLES OF ROW FLAT TUBE

Farhan Lafta Rashid [a], Sarmad Kamal Fakhrulddin[b], Muhammad Asmail Eleiwi [c], Ahmed Kadhim Hussein [d,e], Tahseen Ahmad Tahseen[f], ObaiYounis[g,h,*], Mohammed Ibrahim Ahmed [i]

[a]*Petroleum Eng. Dept., College of Eng., University of Kerbala, Karbala, Iraq*
[b]*Petroleum Eng. Dept., College of Eng., University of Kirkuk, Kirkuk, Iraq*
[c] *Mech. Eng. Dept., College of Eng., Tikrit University, Tikrit, Iraq*
[d]*Mechanical Engineering Department, College of Engineering, University of Babylon, Babylon City, Iraq*
[e] *College of Engineering, University of Warith Al-Anbiyaa, Karbala, Iraq*
[f] *Mech. Eng. Dept., College of Eng., University of Kirkuk, Kirkuk, Iraq*
[g]*Mechanical Engineering Department, College of Engineering in Wadi Addwasir, Prince Sattam Bin Abdulaziz University, Al-Kharj, 11942, KSA*
[h]*Mechanical Engineering Department, Faculty of Engineering, University of Khartoum, Khartoum, 11111, Sudan*
[i]*Department of Mathematics, College of Education for Pure Sciences, Tikrit University, Iraq*

ABSTRACT

The current study numerically analyzes the heat transfer enhancement and laminar fluid flow characteristics of four flat tubes with varying attack front airflow. The heat transfer characteristics of flat tubes are investigated in terms of Reynolds number, heat fluxes, and inclination angle. Four Reynolds numbers are studied (100, 200, 300, and 400), and three heat fluxes on the surface of the tubes are (1000, 2500, 3800 W/m^2), the inclination angle of the four flat tubes are (30^o, 45^o, 90^o). ANSYS Fluent software v.18 discretizes and solves the governing equations using the finite volume approach across a specified control volume. According to the available data, using flat tubes increases the heat transfer in the channel. When tubes were used, the reattachment distance was reduced. When the Reynolds number is changed from 100 to 300, the heat transfer enhancement rises from 40.8 % to 55.79 % for the increase in the heat flux of 24.99 %.

Keywords: *Flat tube, Attack front air angle, Forced convection, One-row tube, CFD simulations.*

1. INTRODUCTION

Heat transfer and fluid flow in tube rows are idealized versions of several processes of industrially significant. Bundles of the tube are often used in the type of cross-flow heat exchanger, although their design stays dependent on empirical pressure drop and heat transfer correlations. Cross-flow tube banks heat exchangers are used in a variety of thermal and chemical engineering processes (Tahseenet al., 2013). Researchers are increasingly interested in improving the performance of small heat exchangers, which are used in a variety of applications such as refrigeration, power production, vehicle radiators, HVAC systems, and so on. Vortex generators (VGs) offer improved heat exchanger performance by producing secondary flow (Zeeshanet al., 2019). The need for developed instruments of consolidated heat exchanger with a performance of high thermos-hydraulic in implementations like heating, combined cooling, ventilation, refrigeration, air-conditioning, and others is driving up demand in the thermal industries. This has resulted in a variety of fin-and-tube heat exchanger (FTHE) designs (Lotfi and Sunden, 2020). FTHEs with tubes of elliptical geometry are used in small equipment of thermal recovery for energy saving in the field of thermal engineering implementations. Heat exchangers have been subjected to extensive study in order to increase their efficiency. Many types of heat exchangers are vastly utilized in sectors such as space spacecraft, transportation, and electronic equipment (Djeffalet al., 2021); thus, these efforts are necessary. Increased heat exchanger efficiency would result in cost, space, and material savings. The heat transfer between a fluid flow and heat exchanger must be determined prior to constructing a heat exchanger with operating performance and optimal parameters (Appa and Gulhane, 2014). Various mathematical models, empirical observations, numerical simulations have been used to investigate the features of heat exchangers (Toolthaisong and Kasayapanand, 2013).

Sahim and Puspitasari (2017) conducted a numerical investigation the forced convection heat transfer performance form one row of elliptical tube with the aspect ratio 0.6 and 0.8. the effect of vertical distance between the tubes on heat transfer were stuided. The results show that the coefficient of drag changes with the transversal distance between the tubes. Besides,The performance of heat transfer for lower value of transversal distance has small values compared to higher distance. Corresponding, the heat transfer ability is depending on the elliptic tube aspect ratio.

Toolthaisong and Kasayapanand (2013) conducted an experimental study of the airside pressure drop and heat transfer characteristics for cross-flow heat exchangers with staggered superficies tube configurations at a steady state. Their research focuses on the impact of attack angles on the airside. The tube aspect ratios influence was also investigated. A heat exchanger with four aspect ratios ranged (0.18, 0.39, 0.66, and 1) and angles of attack ranged (0^o, 30^o, 60^o, 90^o, 120^o, and 150^o). El-Said (2020) investigated the impact

* Corresponding author. Email: oubeytaha@hotmail.com

of air mass flow rate, whole diameter, the inclination angle of the baffle on solar air heater thermal efficiency, as well as the temperature and the pressure differential between the input and output. The air heater's optimum thermal efficiency is around 77 % when the air mass flow rate is (0.03 kg/s), the diameter of the hole is (3 mm), and the baffle angle is (7°). The numerical results were critical for the use of this enhanced solar air heating, demonstrating that the air channel's curved cribriform repulsions are a critical element in boosting collector performance. Tahseenet al.,(2014) predicted the pressure drop and heat transfer for in-line flat-tube arrangement in a cross-flow case. The Reynolds number can range between 10 and 320. The heat transmission and pressure drop findings were given for a tube shape with transverse and longitudinal pitch. The anticipated dimensionless pressure and average Nusselt number estimates are in good accord with earlier research. With a mean relative error of 2.97 % for ANFIS model results, pressure drops less than 2.97 %, less than 1.9 % for the average Nusselt number. Eleiwiet al., (2020a) investigated the heat transfer coefficient for airflow through four-flat tubes with varied intake attack angles. The Re numbers ranged from 1668 to 3782. Heat flux ranged (13.2, 38.5, and 99.8 W/m^2). The intake airflow angles were ranged (30°, 45°, and 90°), which are utilized as samples. The results showed that both the Nu and Re values grow for all heat flow scenarios, but these numbers drop as the air attack increases. Eleiwiet al., (2020b) used an artificial neural network to prognosticate the heat transfer coefficient for four-flat -tubes in cross-flow of air (ANNs). The heat transfer coefficient for three air intake angles (90°, 45°, and 30°) was tested independently with inlet air velocity of 0.2, 0.5, 0.6, 0.8, and 1.2 m/s in an experimental rig with sloping the air entering flow orientation. Heat fluxes were from 13.2, 38.5, and 99.8 W/m^2. The estimated coefficient of heat transfer matches the experimental data well.

The 2D dimension numerical simulation of fluid flow over two tubes with horizontal heat exchange investigated by Wang et al., 2021. The effect of reed length and the cylinder spacing between tubes on heat transfer and fluid flow were stuided. the result show the heat transfer in the downstream of tubes increases by up to 14% at the the reed tube compared to the clean tube in the tube spacing is 1.5. If the tube spacing is 2 the heat transfer at the downstream tube gardually decreases.

Computational fluid dynamics was used by Abdulkarimet al., (2021) to study a unique laminar back-facing step flow heat transfer situation. Laminar flow encounters three circular grooves on the bottom surface of a back-facing step. The grooves have a constant surface temperature. A separation bubble appears after taking a backward step, followed by a reattachment point. Between 10 and 250, the Reynolds numbers varied. Because the flow structure varies substantially, Reynolds number has a considerable influence on heat flux and, as a result, Nusselt number.

According to the literature survey, despite the numerous research on airflow through a row flat tube with various attack angles, no prior results on the problem of improving heat transfer in this system for the laminar fluid flow range are accessible to the authors' knowledge. As a result, this research aims to see how air attack angles affect heat transmission in row flat tubes. The influence of several relevant factors like Reynolds number and heat flux will be investigated.

2. PROBLEM DESCRIPTION AND MATHEMATICAL MODEL

The text of the paper (except the abstract) must be formatted in two-This research looks into four isothermal heated vertical flat tubes in a line. The used flat tubes have two common diameters involving the longitudinal $D_L$is 18.46 mm and transversal D_T is 10.29 mm,the constant heat flux in the tube surface″, inlet velocity $u_{inlet}$and the constant free airstream of temperature T_{inlet}. The S_T indicatesthe transverse pitch to a small diameter rateand equal 2D$_T$. The flat tube is long enough that the tube's end effect may be ignored. As a result, the flow field may be considered to be 2-D. Figure 1 depicts the flow arrangements and computed scopes in a flow across a row of flat tubes.

The following assumptions were taken when constructing the model: (1) airflow physical parameters are uniform; (2) flow of air is laminar and incompressible; and (3) steady-state condition for heat transfer and fluid flow. The following are the governing equations for continuity, momentum, andenergy: The conservation of mass may be represented in the following equation (Alnakeebet al., 2021):

$$\frac{\partial y}{\partial x} + \frac{\partial u}{\partial y} = 0 \tag{1}$$

The momentum equation of the velocity components v and u, respectively, is as follows (Mohamedet al., 2021) (Rahman et al., 2022):

$$\rho\left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y}\right) = -\frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) \tag{2}$$

$$\rho\left(u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y}\right) = -\frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) \tag{3}$$

The energy equation is given by:

$$\rho C_p\left(u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y}\right) = k\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right) \tag{4}$$

The tube surface was subjected to a continuous heat flow (see Figure 1), with values of (1000, 2500, and 3800 W/m^2), while the channel's other walls were considered to be insulated with no gliding situation. Depending on the regular channel inlet dimension intake velocity, the Reynolds number values corresponding to inlet velocities ranged (100, 200, 300, and 400) as shown in Table 1. The Prandtl number value of working fluid (air) is (Pr = 0.71).

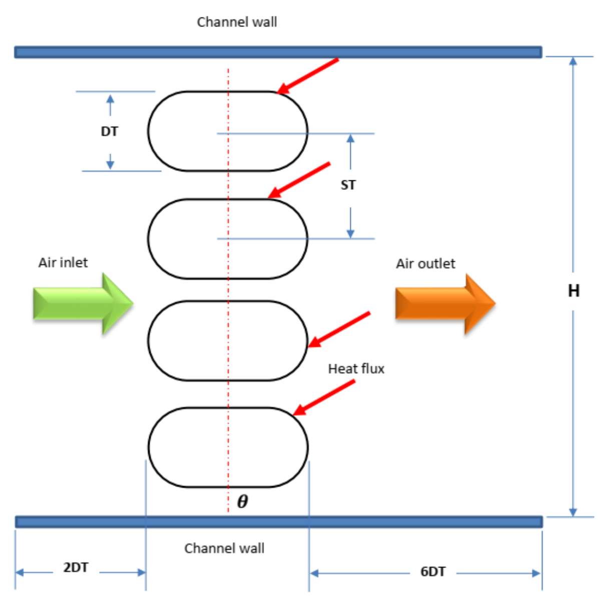


Fig. 1 Schematic diagram of the physical and solution domain at air attack 90°.

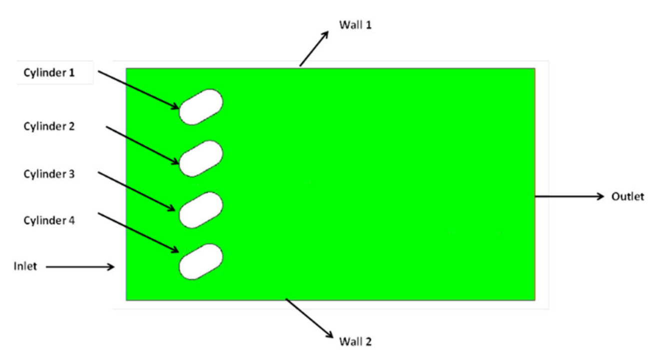


Fig. 2 Schematic diagram of the case study with boundary conditions.

Table 1 Boundary conditions

Location	*Parameter*	*Unit*	*Value*
Outlet	Gauge pressure	Pa	0
Inlet	Velocity	m/s	0.0174, 0.034, 0.052,0.069
Wall	Insulation		$q = 0$
Tube surface	Heat flux	W/m^2	1000, 2500, 3800

The Reynolds number is calculated as follows using the inlet channel height (H)(Al-Zughaibiet al., 2021) (Eleiwiet al., 2022):

$$Re = \frac{u_{inlet} \times H}{\upsilon} \tag{5}$$

The boundary conditions for the current issue are as follows:

- Heat flux changes the temperature at the tube's surface.

At channel inlet: $T_{inlet} = 300\ K, u$= unidirectional, and $v = 0$.

- In the x-direction, the fully developed flow assumption is employed at the channel exit(Hussein et al., 2021) (Rashid et al.,2022):

$$\frac{\partial u}{\partial x} = \frac{\partial v}{\partial x} = \frac{\partial T}{\partial x} = 0 \tag{6}$$

- Insulated wall with no-gliding conditions is presumed on the outflow conduit walls (barring the step downstream) and all solid superficies: v = 0, u = 0, and$\frac{\partial T}{\partial n} = 0$, defining n as normal orientation.

The Nu_x (local Nusselt number) can be calculated using the following equation:

$$Nu_x = -\left(\frac{\partial T}{\partial n}\right) \tag{7}$$

where (k) symbolizes to the working fluid thermal conductivity (air) and (h_X) represents the local heat transfer coefficient. The total Nusselt number is generated by integrating the local Nusselt number throughout the tube surface:

$$Nu = \frac{h \times H}{k} \tag{8}$$

where (H) is the channel height.

2.1 Grid generation

The computational grid layout is used to solve the issue, which is shown in Figure 3. The grid is created with the ANSYS FLUENT program and is non-uniform near the flat tube before becoming uniform elsewhere. In order to grab the strong slope in the hydrodynamic and thermal boundary layer, the mesh is finer along the flat tube walls. There are 326725 items in the computational domain.

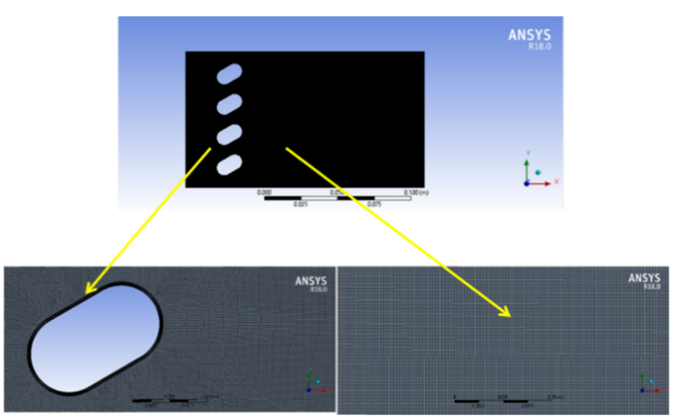


Fig. 3 Image of grid systems created by ANSYS FLUENT software.

2.2 Grid refinement test

ANSYS FLUENT software is used to solve Eqns. (1) to (4) numerically. The commercial program used FV (finite volume approach) for Cartesian velocity components based on a non-orthogonal coordinate system (FerzigerandPerić, 1999) and the SIMPLE algorithm (Patankar, 1980) to solve the continuity, momentum, and energy equations. The convergence was declared when the normalized maximum root mean square (RMS) value was fewer than 10-4. Table 2 shows that the number of cells in the computational domain has been adjusted from (135476) to (337864). As a result, the best number of cells for minimizing error and maximizing CPU resource use is 305930.

Table 2 Results of the different grids tested and analysis of proportional error with different cells number at $\alpha = 90^o$ and Re = 100.

Number of cells	*Overall Nu*	$\left\lvert\frac{Nu^{i+1} - Nu^i}{Nu^i}\right\rvert$ %
135476	127.324	-
204211	130.452	2.457
254467	132.67	1.901
305930	133.152	0.166
337864	133.424	0.069

2.3 Model validation

It is vital to verify the results acquired from the numerical technique before employing them. The heat transfer in the heat exchanger is checked numerically using the Nusselt number, and the two are compared using the Zukauskas correlation (Holman 2010). Figure 4 depicts the circular tube's numerical data, which are incredibly assented with the projected findings from the suggested relations, with an average value of less than 5.594 %.

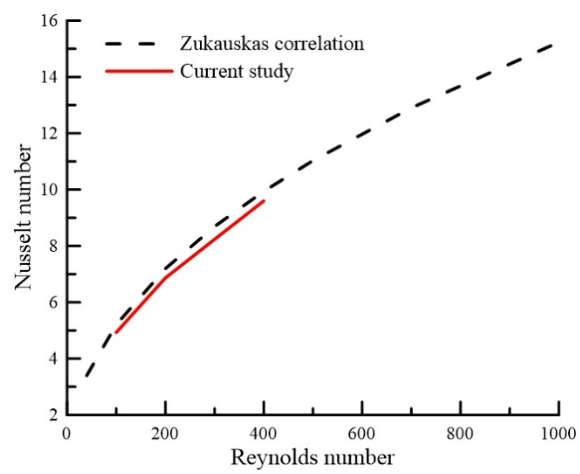


Fig. 4 Nusselt number verification for the circular tube.

3. RESULTS AND DISCUSSION

This research aims to determine how row flat tubes affect fluid flow and heat transfer characteristics. Attack air angles, heat flux on the tube surface, and Reynolds number are the primary characteristics that determine fluid flow and heat transfer in the domain. In the present study, the Reynold numbers are (100, 200, 300, and 400), the heat fluxes are (1000, 2500, and 3800 W/m^2). The air attack angles are (30^o, 45^o, and 90^o).

Frontiers in Heat and Mass Transfer (FHMT), 19, 6 (2022)
DOI: 10.5098/hmt.19.6
Global Digital Central
ISSN: 2151-8629

Figures 5-8 show the contours of velocity, pressure, and temperature distribution, respectively, around the tubes for different Reynolds numbers (100, 200, 300, and 400), with an inclination angle of 30°. The presence of flat tubes reduces the reattachment length of the flow separation backward the step because the flow separates in the region of the tubes, as seen in the images. Near the tubes, the flow increases, and a tiny recirculation zone is created backward. The recirculation zone changes the thermal boundary layer away from the step. The impact of the rising Reynolds number is obvious: the thermal boundary layer will be thinner with higher Reynolds numbers.

The inclination angle of this flat-tube was changed into 45° and 90°; selected results are shown in Figures from 9 to 14 for the velocity contour, pressure contour, and temperature contour, respectively, near the four flat tubes for new inclination angles.

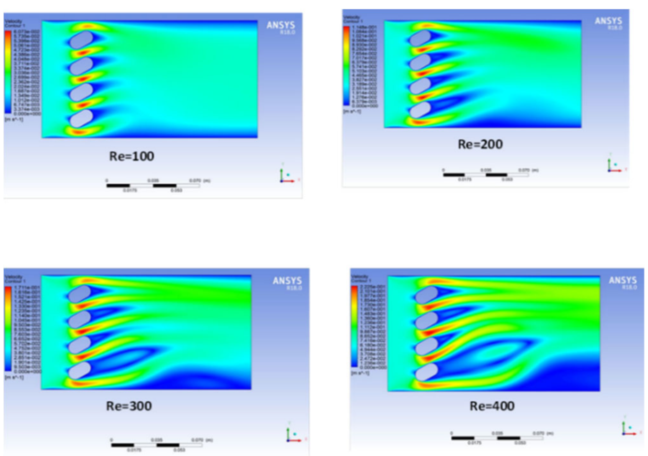


Fig. 5 Velocity contour for different Reynolds numbers with $\alpha = 30^{\circ}$

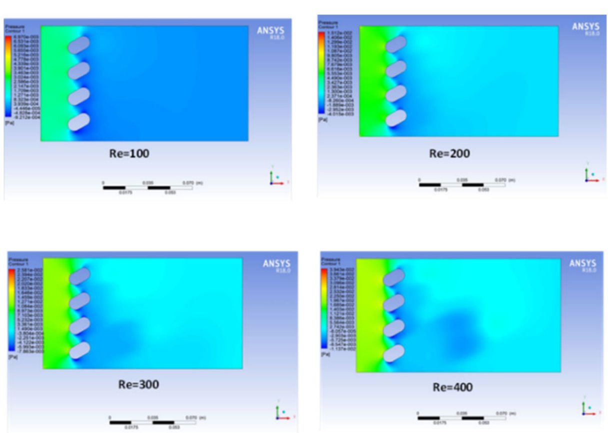


Fig. 6 Pressure contour for different Reynolds numbers with $\alpha = 30^{\circ}$

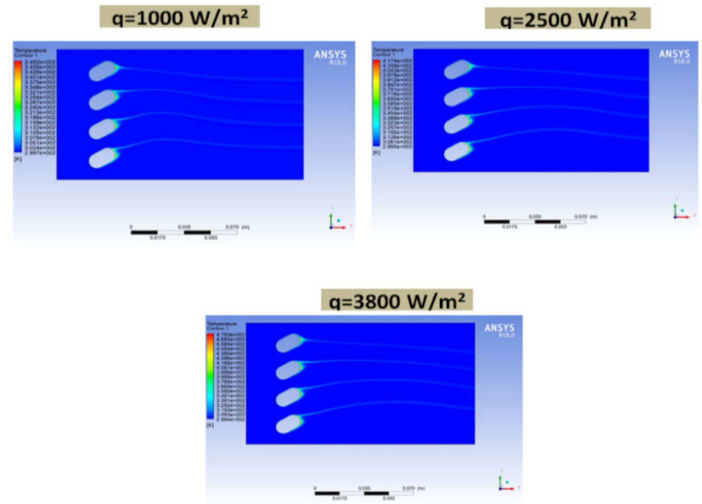


Fig. 7 Temperature contour for different heat flux and Re = 100 with $\alpha = 30^{\circ}$

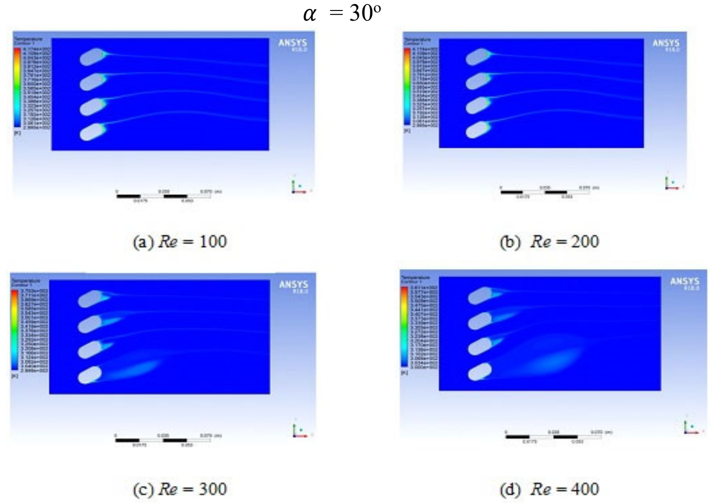


Fig. 8 Temperature contour for different Re and $q'' = 2500\ \mathrm{W/m^2}$ at $\alpha = 30^{\circ}$

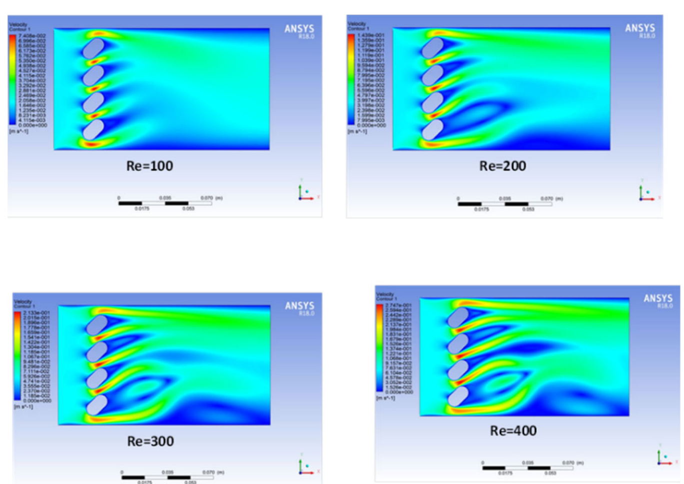


Fig. 9 Velocity contour for different Re number with $\alpha = 45^{\circ}$

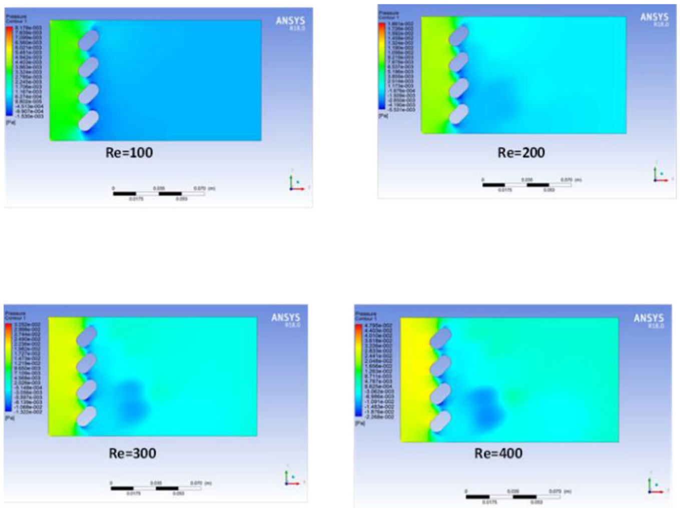


Fig. 10 Pressure contour for different Re number and $\alpha = 45^o$

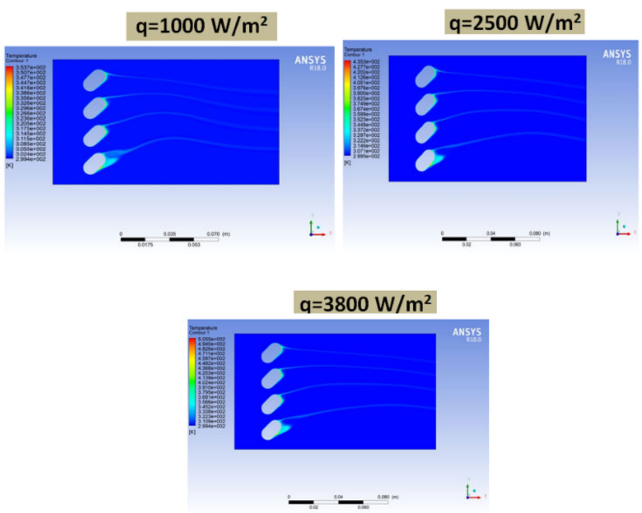


Fig. 11 Temperature contour for different heat flux and Re = 100 with $\alpha = 45^o$

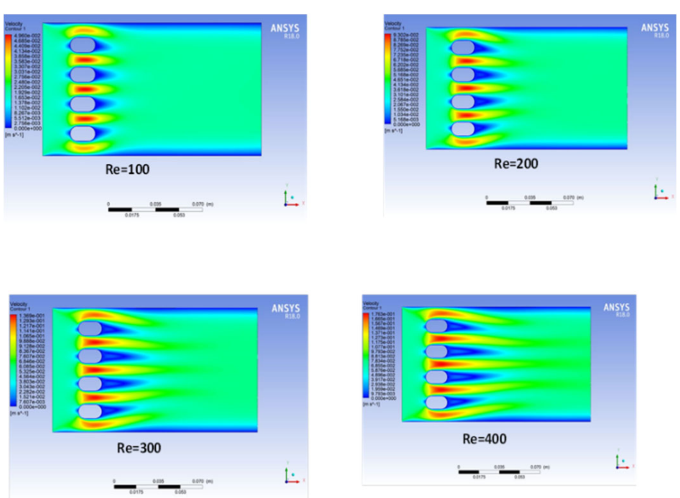


Fig. 12 Velocity contour for different Re number w ith $\alpha = 90^o$

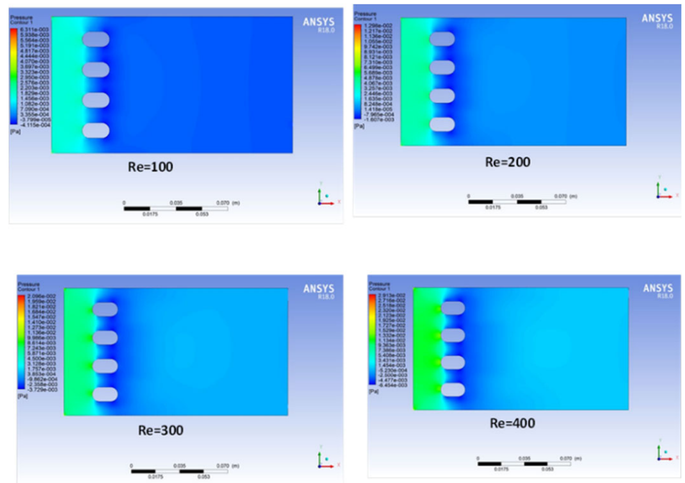


Fig. 13 Pressure contour for different Re number and $\alpha = 90^o$

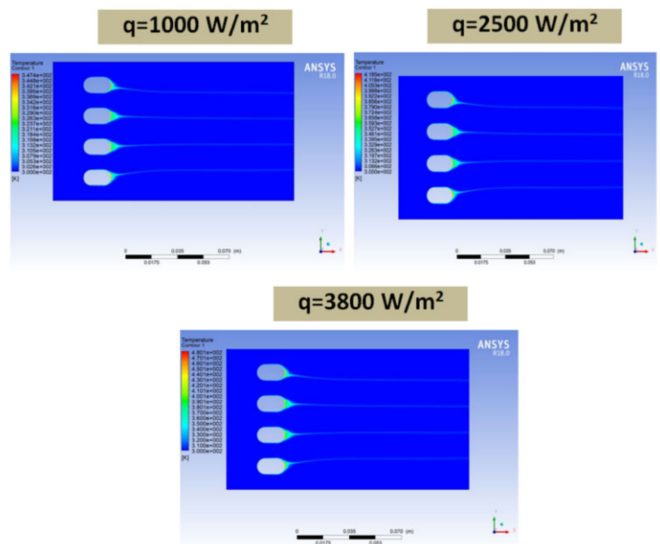


Fig. 14 Temperature contour for different heat flux and Re = 100 with $\alpha = 90^o$

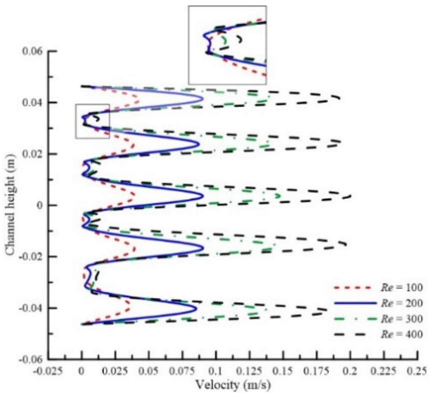
Increased inclination angle affects the recirculating zones following the flat tubes and the upper and lower walls behind the tubes. Changing the inclination angle will have a little impact on the pressure. Increasing the inclination angle will impact the temperature, particularly at minimal Reynolds numbers. The thickness of the thermal boundary layer will increase when the inclination angle is increased.

Figure 15 shows velocity profiles for inclination angles of 30^o, 45^o, and 90^o, as well as numerous Reynolds values. The lower corner of the step (x = 0 mm) is represented by the zero value in the y-axis. Owing to the existence of the flat tubes, the flow zone below and above them will be affected. At (x = 100 mm), the flow in the negative (y) has a lower velocity magnitude than the flow in the positive (y) because of the influence of step edge and inlet velocity. Between and after the tubes, the flow zone starts to behave as a spout. The tubes visibly influence the lower fluid flow region among the tubes, where the flow velocity increases. The velocity profile in the flow zone following the tubes changed to three jets, whereas the air velocity in the recirculating region behind each tube remained low.

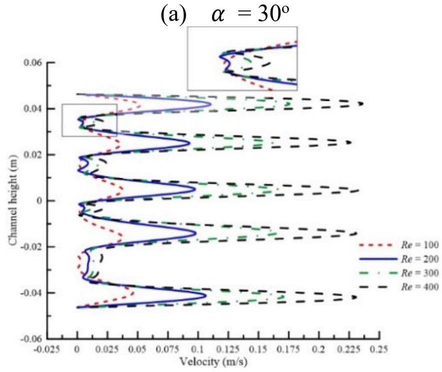
For all heat fluxes with a 30^o inclination angle, Figure 16 demonstrates that the channel's heat transfer and Nusselt number

DOI: 10.5098/hmt.19.6
ISSN: 2151-8629

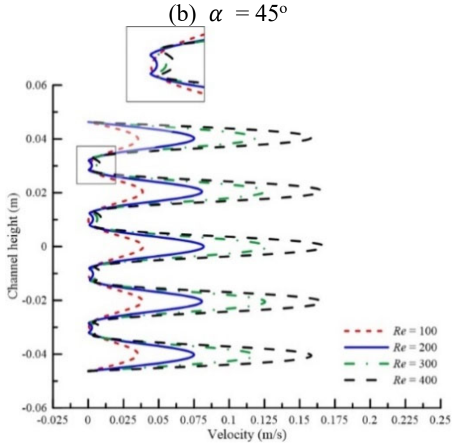
increase as the Reynolds number increases. At higher Re, the effect of increasing the heat flow on heat transfer is visible. When the heat flow is changed from 1000 to 2500 W/m^2, the heat transfer rises by 40.8 % at (Re = 100) and by 55.79 % at (Re = 300). Figure 17 shows the effect of heat flux on Nu with different Re and α. It was observed that the Nu increases by increasing heat flux and Re due to the thermal layer adjacent to the tubes being pushed forward as the Re number increases.



(a) α = 30°



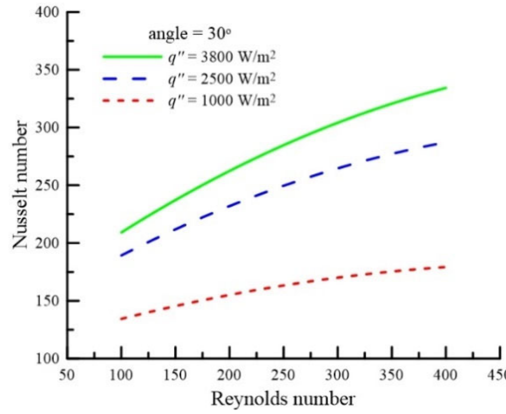
(b) α = 45°



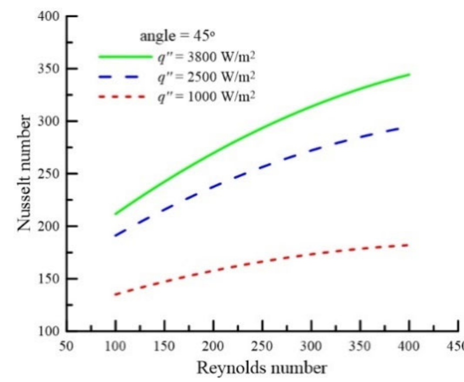
(c) α = 90°

Fig. 15 Velocity profile for different angles and Reynolds number.

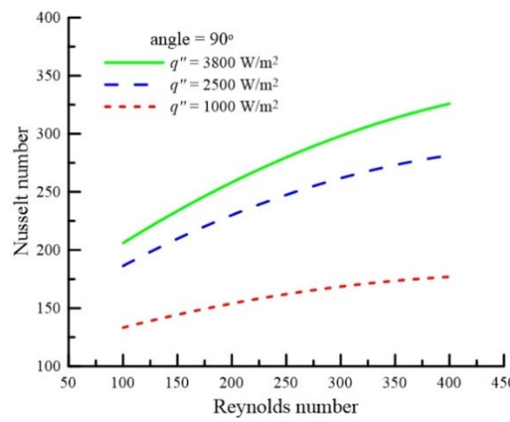
Figure 18 shows the relation between Re and Nu with different heat fluxes on tubes surface with different α. Generally, it was observed that the Nu for the α = 45° highest among the four air inlet angles with the same Re. The α = 90° is the lowest. The reason is the distance between tubes at α = 90° was in the same direction of flow, while on the α changes to 30° and 45° led to the occurrence of vortices behind the tubes, and they were lower value at α = 45°, which helped to increase the temperature difference between the working fluid and tubes surface. This behavior was also observed by Tang et al.(2019) and Eleiwiet al.(2020a).



(a) $\alpha = 30^o$

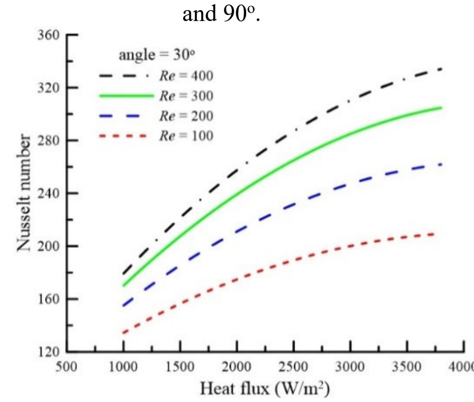


(b) $\alpha = 45^o$

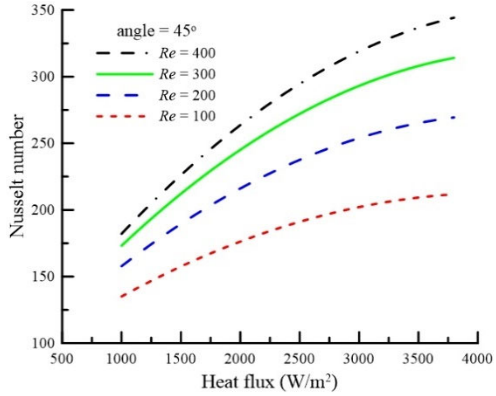


(c) $\alpha = 90^o$

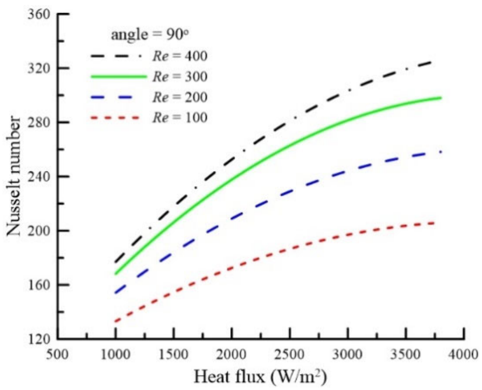
Fig. 16 Nusselt number variation with Re number for α = 30°, 45°, and 90°.



(a) $\alpha = 30^o$

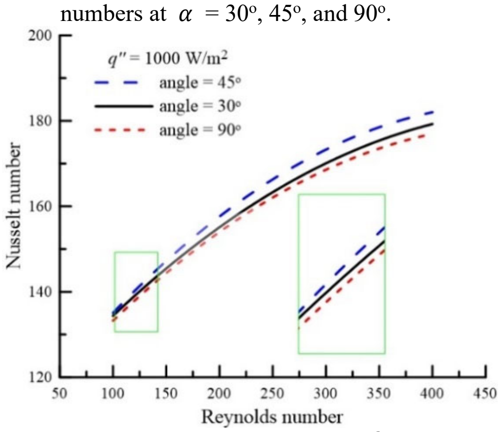


(b) $\alpha = 45^o$

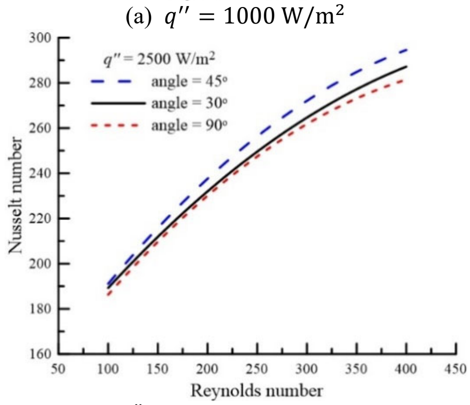


(c) $\alpha = 90^o$

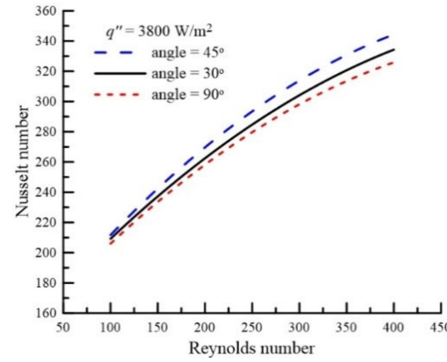
Fig. 17 Effect of the heat flux on the Nu number with different Re numbers at α = 30°, 45°, and 90°.



(a) $q'' = 1000\ \mathrm{W/m^2}$



(b) $q'' = 2500\ \mathrm{W/m^2}$



(c) $q'' = 3800\ \mathrm{W/m^2}$

Fig. 18 Variation of Nu number with Re number for different inclination angles

4. CONCLUSION

Numerical research of the laminar airflow characteristics and heat transfer augmentation of four flat tubes with diferent attack air angle is carried out in this paper. The effects of Reynolds number (Re = 100, 200, 300, 400), heat flux (q" = 1000, 2500, 3800 W/m^2), and row flat-tube inclination angle (30°, 45°, 90°) on heat transfer characteristics are investigated. Form the result it can be conclusion, the presence of flat tubes reduces the reattachment distance of the flow.The flat tube strongly affects the velocity field; it seems to be a similar jet profile after them.Thehigh velocity in the flow passage considerablyincreases with an decrease in attack angle for a fixed Reynolds number.Enhancing the heat transfer by (40.8%) at (Re = 100) for heat flux increment by 24.99%, while the heat transfer enhancement by (55.79 %) at (Re = 300). The heat transfer coefficient has a maximum value at an inclination angle of 45° and a minimum at 90°.

ACKNOWLEDGEMENTS

This publication was supported by the Deanship of Scientific Research at Prince Sattam bin Abdulaziz University, Alkharj, Saudi Arabia.

NOMENCLATURE

C_p	specific heat capacity (kJ/kg.K)
D_T	inside tube diameter (m)
D_L	longitudinal tube diameter (m)
k	thermal conductivity (W/m.K)
p	pressure (Pa)
Nu	Nusseltnumber
q''	heat flux (W/m^2)
Re	Reynolds number
Pr	Prandtl number
h	heat transfer coefficient (W/m^2.K)
H	channel height (m)
s_T	transversal pitch
T	temperature (°C)
v, u	components of velocity (m/s)
x,y	cartesian coordinates (m)

Greek symbols

μ	dynamic viscosity of air (kg/m.s)
ρ	the density of air (kg/m^3)
α	attack angle of flat tubes (degree)
υ	kinematic viscosity of air (m^2/s)

REFERENCES

Abdulkarim, A. H.,Eleiwi, M. A.,Tahseen, T. A.,and Canli, E., 2021, "Numerical forced convection heat transfer of nanofluids over back facing step and through heated circular grooves",Mathematical Modelling of Engineering Problems, 8(4),597-610. https://doi.org/10.18280/mmep.080413

Alnakeeb,M. A.,Saad,M. A. andHassab, M. A. 2021. "Numerical investigation of thermal and hydraulic performance of fin and flat tube heat exchanger with various aspect ratios",Alexandria Engineering Journal, 60, 4255-4265. https://doi.org/10.1016/j.aej.2021.03.036

Al-Zughaibi, A., Hussein, E.Q., Rashid, L.R., 2021, " Numerical Investigations of Fluid Structural Interaction for Aircraft Wing Flap Structure Using CFD Technique",Journal of Mechanical Engineering Research and Developments, 44, 138-150.

Appa, A.,and Gulhane, N.P., 2014, "CFD analysis of fin tube heat exchanger using rectangular winglet vortex generator",International Journal of Emerging Technologies in Computational and Applied Sciences, 8(5),450-457. https://doi.org/10.1007/s12206-012-0702-2

Djeffal, F., Bordja, L.,Rebhi, R.,Inc, M., Ahmad, H.,Tahrour, F., Ameur, H.,Menni Y., Lorenzini, G., Elagan, S. K., and Jawa, T. M., 2021, "Numerical investigation of thermal-flow characteristics in heat exchanger with various tube shapes",Applied Sciences,11,9477. https://doi.org/10.3390/app11209477

Eleiwi, M. A., Zainal, O. A.,Tahseen, T. A., and Mustafa, A. W., 2020a, "Effect of front air attack angleson heat transfer coefficient of the cross-flow of four flat-tube",Heat Transfer,1-17. https://doi.org/10.1002/htj.21897

Eleiwi, M. A.,Tahseen, T. A.,and Ghareeb, A. H., 2020b, "Intelligent control based estimation of heat transfer coefficient from four flat tubes with different attack air angles", Journal of Advanced Research in Fluid Mechanics and Thermal Sciences, 72 (2), 65-78. https://doi.org/10.37934/arfmts.72.2.6578

Eleiwi, M. A., Tahseen, T.A., and Hameed, A. F., 2020c. "Numerical study of fluid flow and heat transfer in a backward facing step with three adiabatic circular cylinder", Journal of Advanced Research in Fluid Mechanics and Thermal Sciences,72(1), 80-93. https://doi.org/10.37934/arfmts.72.1.8093

Eleiwi, M.E., Rashid, L.R., Khalaf, A.F., Tuama, S.A., 2022, "Numerical Investigation of Conjugate Heat Transfer Between Spherical Solid Body and Fluid",Mathematical Modelling of Engineering Problems, 9, 491-497. https://doi.org/10.18280/mmep.090227

El-Said,E. M.S.,2020, "Numerical investigations of fluid flow and heat transfer characteristics in solar air collector with curved perforated baffles",Engineering Reports, 2, e12142, 1-15. https://doi.org/10.1002/eng2.12142

Ferziger,J.H.,and Perić, M.,1999, Computational methods for fluid dynamics, third ed., Springer-Verlag Berlin Heidelberg USA.

Hussein, E.Q., Azziz, H.N., Rashid, F.L., 2021, "Aerodynamic Study of Slotted Flap for NACA 24012 Airfoil by Dynamic Mesh Techniques and Visualization Flow",Journal of Thermal Engineering, **7**,230-239.
J. P., Holman, 2010. Heat transfer, Tenth ed. McGraw-Hill Companies, Inc.New York, USA:

Lotfi, B., and Sunden, B., 2020, "Thermo-hydraulic performance enhancement of finned elliptical tube heat exchangers by utilizing innovative dimple turbulators",Heat Transfer Engineering, 41(13),1117–1142. https://doi.org/10.1080/01457632.2019.1611132

Mohamed A. A., Mahmoud A. S., and Mohamed A. H. 2021, "Numerical investigation of the thermal and hydraulic performance of fin and flat tube heat exchanger with various aspect ratios",Alexandria Engineering Journal, 60, 4255-4265. https://doi.org/10.1016/j.aej.2021.03.036

Patankar, S.V.,1980, Numerical heat transfer and fluid flow, Hemisphere, Washington, DC., USA.

Rashid, F.L., Hussein, A.K., Malekshah, E.H., Abderrahmane, A., Guedri, K. and Younis, O., 2022", Review of Heat Transfer Analysis in Different Cavity Geometries with and without Nanofluids," Nanomaterials, 12, 2481. https://doi.org/10.3390/nano12142481

Rahman, M.H., Aljibory, M.W., Rashid, F.L., Hussein, E.Q., 2022, "Sweep Angles Influence on the Aerodynamic Characteristics of NACA 2412 Wing with Supersonic Flow",Mathematical Modelling of Engineering Problems, 9, 757-761. https://doi.org/10.18280/mmep.090323

Sahim, K., Puspitasari, k., 2017, "Convective heat transfer of one row arrangement of elliptical cylinder",Frontiers in Heat and Mass Transfer, 8(16), 1-6. http://dx.doi.org/10.5098/hmt.8.16

Tahseen, T. A., Ishak, M., and Rahman, M.M.,2014, "Performance predictions of laminar heat transfer and pressure drop in an in-line flat tube bundle using an adaptive neuro-fuzzy inference system (ANFIS) model",International Communications in Heat and Mass Transfer, 50, 85–97. https://doi.org/10.1016/j.icheatmasstransfer.2013.11.007

Tahseen, T. A., Ishak, M., and Rahman, M.M.,2013,"laminar Forced Convection heat transfer over Staggered Circular Tube Banks: A CFD Approach",Journal of Mechanical Engineering and Sciences, 4,418-430. https://doi.org/10.15282/jmes.4.2013.6.0039

Tang, L., Du, X., Pan, J, and Sundén J., 2019. "Air inlet angle influence on the airside heat transfer and flow friction characteristics of a finned oval tube heat exchanger",International Journal of Heat and Mass Transfer,145, 118702. https://doi.org/10.1016/j.egypro.2013.06.770

Wang, Z., Wang, Z., and Yang, M., 2021, “Transversal flow and heat transfer of two cylinders with a flapping reed between them”,Frontiers in Heat and Mass Transfer, 17(10), 1- 7. https://doi.org/10.5098/hmt.17.10

Toolthaisonga, S., and Kasayapanand, N.,2013, "Effect of attack angles on airside thermal and pressure drop of the cross-flow heat exchangers with staggered tube arrangement",Energy Procedia,**34**:417 – 429. https://doi.org/10.1016/j.egypro.2013.06.770

Zeeshan, M., Nath, S., and Bhanja, D., 2019, "Numerical investigation to predict optimum attack angle combination of longitudinal vortex generators in compact heat exchangers for thermo-hydraulic heightened performance",Indian Academy of Sciences, 44,241. https://doi.org/10.1007/s12046-019-1219-5