J. Electrical Systems 20-5s (2024): 2684-2704

[1]Ayad Jawad Kadhim

[2]Mohammed K. Hamza

[3]Nawar S.A. Bakly

[4]Qusai A. Mahdi

Numerical Investigation of Heat Exchanger Enhancement for a Staggered Arrangement with Corrugated Rods in Front

Abstract: - The heat exchanger is used in many industrial and engineering activities.Where, numerous applications confirms on a large heat transfer rate. However, the twisted tube is one of heat transfer rate enhancement methods. Also, the turbulent flows are often used instead of laminar for enhancement heat transfer rate. Therefore, the tube arrangement can be used in a suitable way for utilized this purpose. The shape features of tubes in the heat exchangers are very important for enhancing their performance. As well, the pressure drop and heat transfer rate have a remarkable effect on the thermal efficiency of the heat exchanger. In this research, a numerical investigation with a new way for heat exchanger tube staggered arrangement is presented. Where new solid spiral rods with oval sections are inserted in front of main tubes to induce high turbulent flow intensity as well as redistribution airflow over exchanger tubes. However, the new air distribution will help the airflow pressure over all the tubes to push the thermal boundary symmetrically behind the tubes. Therefore, two different sectional tubes were presented (circular and elliptical). These cross-sectional areas for both types have the same values. The main tubes have no twisted to prevent decreases in pressure. The numerical analysis used the K- ε model at a low Reynolds number to the simulation of this situation. Where, the eddy will generate in the vicinity region to pipe walls due to twisted pipe as well as the obstruction the cross flow. However, there are a good improvement of heat transfer rate without spend any energy depending on the control the shape and arrangement of pipes. The results show that the corrugated rod will increase the Nusselt number at different values of Reynolds. However, the elliptic tube gives the best results for heat transfer rate and maximum Nusselt number. The results compared with others works and give a good agreement.

Keywords: Heat exchanger; Numerical; corrugated tubes; thermal heat performance

1. Introduction

This work studies the laminar and turbulent flow around several shapes of the cylinder at a low Reynolds number. Most engineering applications need to improve the exchange of heat between fluids to get the best results for industrial products at minimum cost. One of the amazing devices is the use of a heat exchanger. Heat exchangers are found in numerous applications, like the generation of steam in a boiler or air cooling in the coil of an air conditioner and in food preservation, medical and oil refinery and many other applications. The most common flow problem is under-predicting the flow conditions. For a suitable design for a heat exchanger, the flow across tube behavior must be controlled. Usually, tube banks are arrayed in an in-line or staggered mode. Consequently, many tube shapes and arrangements are produced and accomplished. However, the shell tube is a type of heat exchanger that used different shapes and arrangements of tubes for flow control to improve its thermal performance. One of the common types of tube arrangement is the staggered tube bundle, in which the cross-flow is utilized achieving a high rate of heat transfer, satisfying low-pressure drop, and simple manufacturing. Consequently, many engineers and designers have produced new shapes of heat exchangers especially its pipes shape to satisfy the maximum benefit. The turbulent model can be produced as a method of enhancement of heat transfer rate, for example of applications ribs and conical nozzles. These shapes are used to enhance their performance from the point of view of heat treatment and transfer rate as well as decrease the pressure drop.

The pressure drop refers to the energy loss and it is the commitment to the friction factor. Moreover, there is a close relationship between thermal enhancement and fluid properties. Also, the numerous new pipe shapes are presented in another type of heat exchanger like of double pipe type. Where these novel shapes have been presented flow characteristics can be used to improve other types of heat exchangers. For instance, Corcoles et al [1] have been presented numerically and experimentally investigated the turbulent flow for the varying pith diameter ratio of corrugated inner pipe of the double pipe heat exchanger. Where the pipes under consideration

[1] Remote Sensing Department, College of Remote Sensing and Geophysics, Alkarkh University of Science, Baghdad, Iraq

[2,3,4]Department of Polymers and Petrochemical Industries, College of Material Engineering, University of Babylon, Iraq

have the same computational domain and characteristics of actual uses. Zaid et al [2] presented spirally corrugated tubes where the suggested new format increased the heat exchanger performance. While, Bharat et al [3] enhanced the heat exchanger tube by increasing the turbulent intensity in order to induce a swirl. However, this swirl is produced by inserting a helical tape within the inner tube.

Some researchers presented new pipe shapes to satisfy the increasing thermal performance of heat exchangers with declining flow pressure. Where, Amin et al [4] presented a numerical study for the oval tubes. In order to produce turbulent flow, the tube was twisted in such a way as to get the maximum enhancement of heat rate. Thus, Pethkool et al [5] presented the experimentally work on a helically corrugated tube with different pitch diameter ratios. They found that for this shape at a low Reynolds number, the pressure drop and heat transfer will be greater than the smooth one. Moreover, Fares et al [6] have thermally studied the several types of oval and flatness tubes. They observed that the heat transfer increased with the axis with both types of tubes. Nakhchi [7] presented a numerical investigation for double V-cut twisted tapes for pipes that were carried by Nanofluid for analysis and improved the performance of heat exchangers. John et al [8] presented a numerical solution for the heat transfer rate of the helically double pipe heat exchanger with a corrugated inner tube. They found that the pressure drop will be bigger in the corrugated pipe compared with the smooth one in spite of increasing the heat transfer rate with the corrugated tube.

The properties of flowing fluid play a good effect on improving the heat transfer rate. Many works have been presented for tubes bundle with innovative manufacturing getting maximum heat transfer rate and trying to regulate the pressure drop hold values. Gray et al. [9] modelled a numerical code to investigate the Staggered-tube heat exchanger thermal load. Ahmed et al. [10] carried out a heat exchanger that has a tubes bank arrangement in a staggered way. Where these tubes are circular and hold a splitter attachment to increase the performance of the heat exchanger. From another point of view, there are many procedures to increase the heat exchanger performance by the increasing heat transfer rate with a low-pressure drop, like the work of Halil et al. [11]. They observed by numerical computation that, the baffle spacing for shell and tube heat exchanger can be modified to improve its performance. Also, Dawid et al. [12], studied the heat transfer coefficient (HTC) for a heat exchanger having four rows of tubes. Also, they show correlations individually for the air-side Nusselt numbers that describe each row of tubes alone. Donkyoun et al [13] presented a new correlation for bank tube inline that can show the values of heat transfer coefficient individually for each pipe row of this type of heat exchanger. This correlation satisfies by numerical investigation of the effect of the longitudinal pitch in a tube bank using commercial software Fluent. Chidanand K. et al. [14] presented the cam-shaped tube as a new shape for a bank in line tubes. Moreover, they presented the flow friction effect for Reynolds range 11500 to 42500. However, they found an obvious decrease in friction. While the performance of the heat exchangers is very clear. The new way for the arrangement of banks of twisted tubes was presented by Nidal H. et al. [15]. The new arrangement is unique and differed from the situation of inline and staggered arrangements. The numerical solution gives moderate improvement by decreasing the friction as well as increasing of thermal performance of the new heat exchanger. Ahmadali Gholami et al [16] developed a new numerical investigation by introducing the corrugated fin oval tube heat exchanger type. The new shape helps to decrease the pressure drop and enhancement the thermal performance of the heat exchanger. Eiamsa-ard et al. [17] Presented a new shape of heat exchanger pipe with dual twisted tapes. The twisted tapes, as well as Nano-fluid, are used to improve the circular tube thermal applications. Haolin Ma et al. [18] Presented a numerical simulation of semi-circle cross-sectioned slotted tubes. The tubes are placed in a staggered and inline arrangement so to get a maximum enhancement of flow thermal characteristics. On the other hand, the acquired tube shape will increase the pressure drop.

The numerical methods used in most works are modified to improve heat exchanger performance. Mangrulkar et al [19] examined three models with different grid procedures and sizes near the pipe wall. They found that the results of Nusselts number and friction factor at high Reynolds number commitment to grid generation method and grid size. On the other hand, at a low Reynolds number, the viscous forces that are found in the vicinity region of the pipe wall clearly affect making the grid size effect unclear. Joseph et al [20] presented a finned tube with filled the polymer. They found that the polymer heat exchanger gives a superior thermal performance. Experimental research presented by Vinous [21] for double pipe heat exchangers can be used to manufacture and design a suitable heat exchanger depending on thermal results. K. Ravikumar et al [22] have calculated the fin-tube heat exchanger where these fins have several values of fin thickness. The numerical study was done using FLUENT to enhance this type (fins tube) heat exchanger. The inclined angle of a bundle of tubes for cross-flow has no clear change in flow characteristics and thermal performance; these results were presented by Yongqing et al. [23]. The pressure drop is one of the important design parameters for heat exchangers. Parikshit et al [24] presented a simple model to calculate the pressure drop using the finite element method. Moreover, they found that Zukauskas friction factor has a suitable value for the angle of twisted 30o and 90o for tubes bank. Yifan Zhang et al. [25] experimentally investigated the enhancement of heat exchangers and drag reduction due to corrugated tube conveyed xanthan gum (XG) solutions. They found that the tube shape and the characteristics of polymer solution have remarkable effects on thermal improvement and drag reduction for heat exchanger behaviour. Xiuzhen Li et al. [26] numerically investigated the pressure drop performance bundle tubes with the inline arrangement. These tubes are twisted and have an oval cross-section area. They found that the heat transfer rate decreased as twist pitch increased. LI Xiuzhen et al. [27] present the numerical solution for bundle tubes with the staggered arrangement. These tubes have an elliptical cross-sectional area and corrugated variable twisted steps.

The obvious remarkable relation between the heat transfer rate and tube characteristic shape is due to the induced new way for turbulent intensity and flow. The experimental work is very important for validation and presented a suitable correlation for indicating the thermal characteristics of the heat exchanger. M. Ishak et al. [28] presented an experimental work for calculating the flow and heat features for a bank of tubes with an oval cross-sectional area. Moreover, they suggested a correlation with very high confidence as measured by the heat and flow parameters. They found that the pressure drop reverses rotation with the Reynolds number. Moreover, many researchers deal with a new method for improving the heat exchanger performance by inserting a twisted shape in tubes. Pengxiao Li et al. [29] numerically investigated the special tube shape to enhance the thermal performance when used with a heat exchanger. Also, many parameters are examined for their effect on the thermal performance, especially, the twisted tape effect on and addition to the number of turns. They found this twisted tap will increase the thermal performance of tubes that may be used with heat exchangers even with a low Reynolds number. Indri Yaningsih et al. [30] also, suggested an innovative physical shape to increase the turbulent flow this will modify the Nusselt number producing a higher performance for heat exchangers. On the other hand, the friction factor will obvious improvement. Where the friction factor decrease as the Reynolds number increase. This new shape has been represented by a louvred strip inserted in heat exchanger concentric tubes. Agung et al. [31] have presented a numerical simulation to investigate the effect of short-length twisted tape located inside a heat exchanger pipe on heat transfer rate. For validation, the numerical conclusions, have been compared with the experimental results and correlations presented by other researchers empirically. They found that tape-length ratios improved the Nusselt number as well as the friction factor as the Reynolds number increased. Turbulent flow due to twisted tape will improve the performance of convection even low Reynolds Number.

In this work, try to increase the thermal performance of the heat exchanger which has been accomplished by introducing two corrugated oval rods in front of three tubes, where, these tubes are arranged in staggered types. This will leads to enhancing the thermal performance through heat transfer by increasing the heat dissipation with low air velocity as well as keeping the pressure drop through pipes path constants. The numerical simulation and solution are produced for turbulent flow over two parts of cross-sectional types of tubes (circular and elliptical) using fluent software Ver.16.1. As presented in previous research above, the increase in the spiral pitch of tubes will increase the pressure drop. Therefore, the tubes in this work have not corrugated. Therefore, the flow turbulent is achieved by obstruction flow by corrugated two oval rods that have been placed in front of tubes adding to the drag induced by the tubes themselves. However, the flow will be turbulent and the swirl almost will be generated throughout the flow over and behind the rods and tubes as well as the boundary layer maybe perform. Consequently, the k-epsilon equation has been used to analyze the turbulent flow that is satisfied by the models under consideration. On the other hand, the Zhukauskas model is used for calculating the thermal characteristics and friction factor for (circular and elliptical) models, which, this work will be elaborated in subsequent articles.

2. Physical model of the problem

The model under numerical investigation consists of a test section with dimensions 210 mm x 200 mm. Two parts of tubes with equal cross-sectional area are used for exchanging heat flux with flow air across them. However, the first part represents a circular cross-sectional area for three tubes. While the second part is presented by three tubes that have an elliptical cross-sectional area. To make the results be compared with the experimental work of M. Ishak et al. [28] the new two models have the same cross-sectional area that for the Ishak model. The new two models represented in this work and the Ishak model can be shown in figure (1). The airflow for each part has ranged from 0.6 to 1 m/s. The parameters for each part will be illustrated in detail as below:

1.1. Part 1

Part (1) can be shown in figure (2). As shown in this figure the model represents three circular tubes diameter of 14.212 mm and a length of 200mm that has been arranged in such a way to give the staggered arrangement for one cell. This part consists of two sections. Section (a) refers to three circular tubes St=40 mm, Sd=56.56 mm and each pipe length equal to 200mm, insert inside the test section see figure (1) (d). The cross-section of (b) is the same as that of (a) adding two corrugated rods in front of the circular pipes as shown in figure (2) b.

1.2. Part 2

Part (2) can be shown in figure (2) in two sections (c) and (d). As shown in figure (2) section (c) is presented three oval tubes with the section dimensions (B=10.09 mm and A=5 mm) to get the same cross-sectional area as part one. While part (d) distinguishes the presence of two twisted rods at the front of the tubes. These rods are exactly similar in cross-sectional area to the part (1), but with an elliptical cross-section and have the same length, vertical St and lateral Sd for part 1 see figure (1) part e. This study has taken into account the development and impact of oval shape on flow and heat transfer improvement, especially for low-capacity.

flow, that is, the practical application of low cost, high performance and easy to use, which has been fully studied.

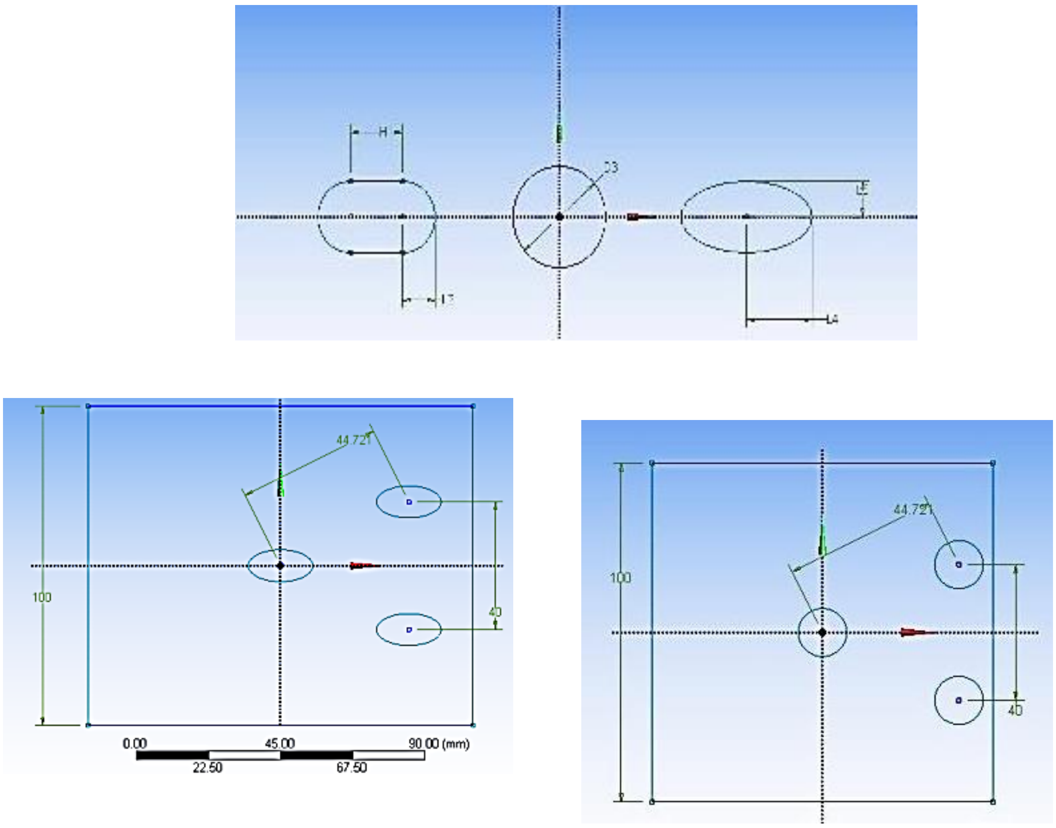


Fig. 1. The cross sectional with the same area of tubes for a) ref. [28], b) circular part, and c) elliptical part. The St and Sd for both parts under investigation are same as well as pipe and rods cross sectional area.

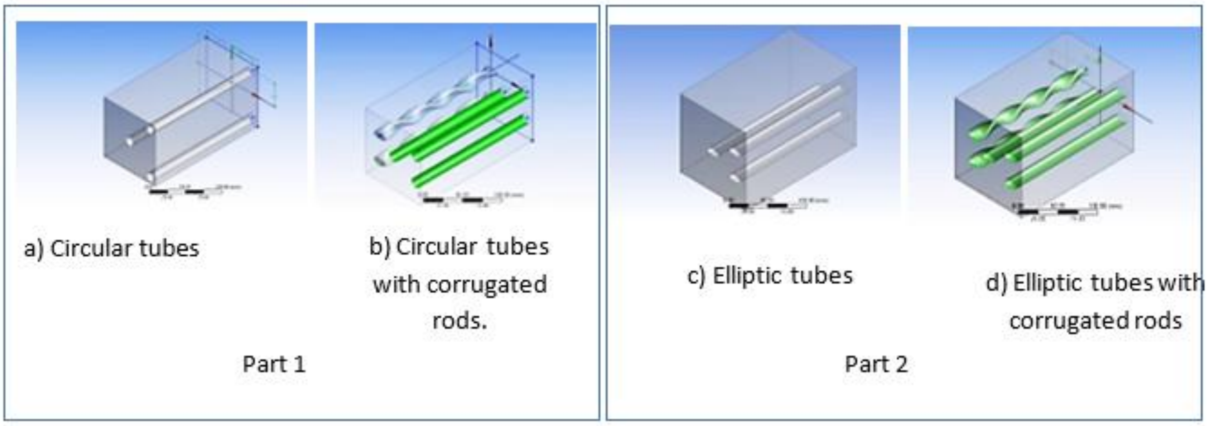


Figure 2. The control volumes of two-part of models under investigation: Part (1) circular tube and part (2) elliptical tube, without corrugated rods represent by a and c for part 1and part 2 respectively. Also, corrugated rods represented b and d for part 1and part 2 respectively.

The locations of minimum area for airflow over tubes can be shown in the figure below at distances L/8 and L/2 respectively. The minimum area represents the location of maximum velocity or refers to the location of variation of air velocity. The variation of air velocity around the twisted tubes will induce flow disturbance around tubes, especially in the vicinity region of tubes. The eddy produced by this method does not need to drop the pressure and not spend any excess energy. Flow-through tubes with irregular paths need the energy to produce turbulent flow through the tubes, as presented by most previous researchers. This energy is abbreviated as that used by this method. In addition, the irregular duct section will increase the pressure drop through the pipes, this method reduces the requirements of extra turbulent flow.

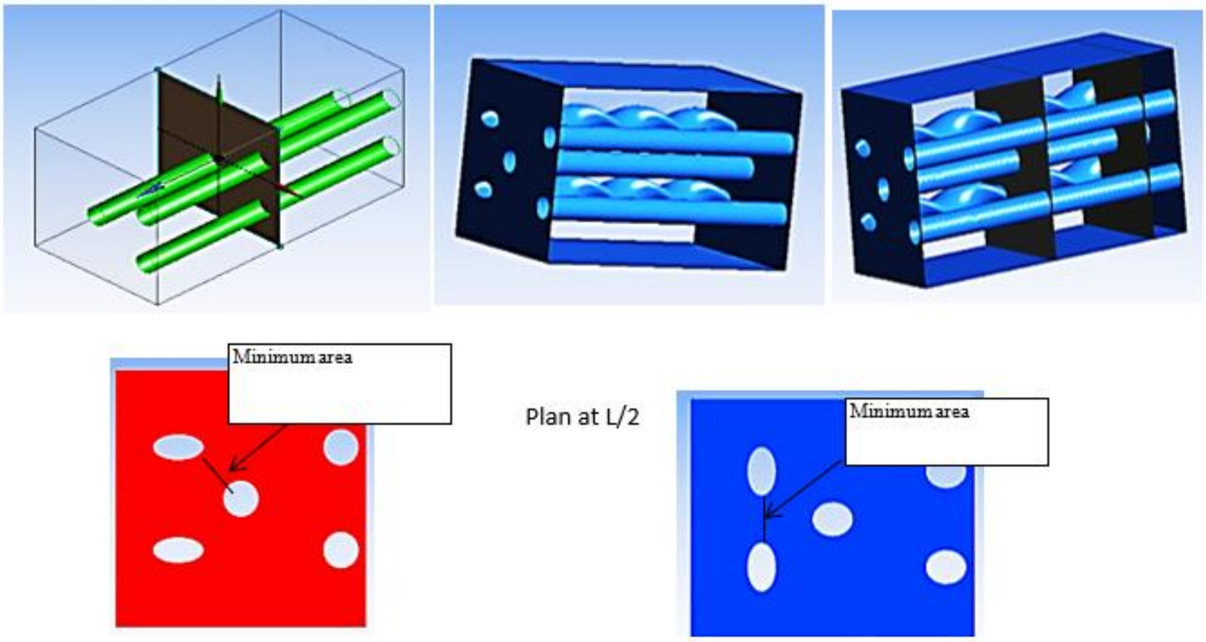


Figure 3.The plane's position (L/2) and (L/8) that were used to display the temperature, pressure and velocity distribution for the circular tube model.

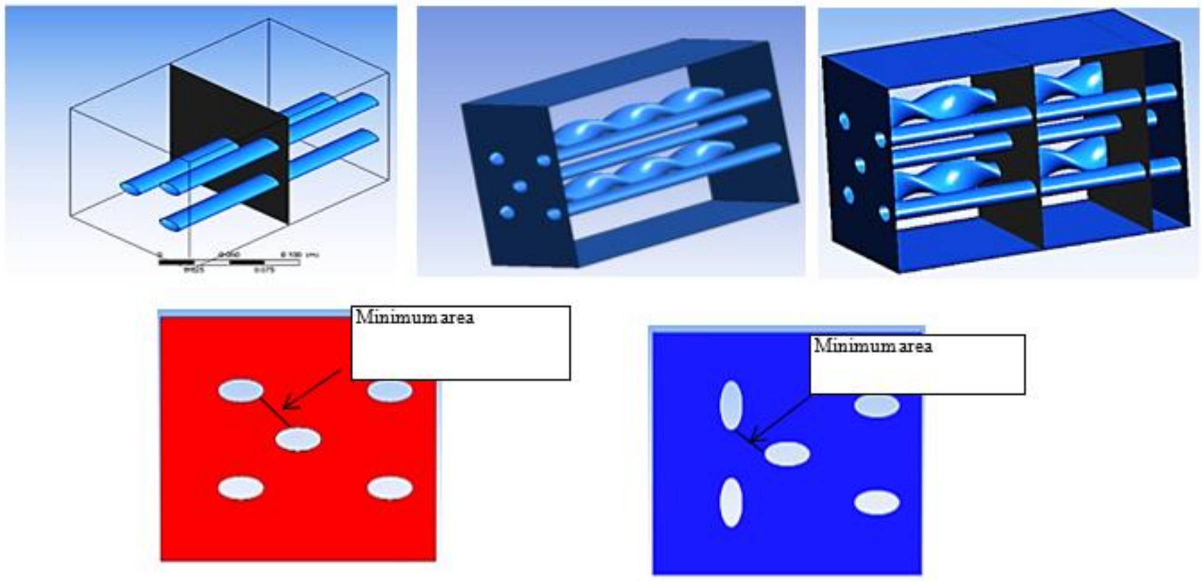


Figure 4. The plane's positions (L/8) and (L/2) that used to display the temperature, pressure and velocity distribution for the elliptical tube model.

The Nu number is affected by the temperature and velocity distribution of the fluid outside the tube. The area near the surface walls of the pipes is the most important area for heat flow. The region closes where $f_{\mu<1}$, the place of the special boundary layer, in this region the eddy will be smaller due to low Reynolds number or high viscosity. The value of this region is the main region under numerical study. Consequently, the inflation method has been used in several areas around tubes.

1. Governing Equations and Boundary Conditions

The governing equations of continuity, momentum, and heat transfer rate that are used for numerical modelling of the turbulent boundary layer and temperature distribution induced by airflow over each modelling part and thermal features above tubes will be presented as follows [32,33]and [28]:

Continuity equation

$$\nabla.\rho U = 0 \tag{1}$$

Momentum equation

$$\rho U.\nabla U = -\nabla P + \mu\nabla^2 U \tag{2}$$

Energy equation

$$\nabla.\rho UCpT = \nabla.(\alpha\nabla T) \tag{3}$$

The assumption is the same for the two parts presented in this work and can be summarized below:

Airflow is steady crossing pipes for both parts.

Air velocity is uniform at the inlet side of the test section, moreover, it's turbulent over the corrugated rod and when flowing across the pipe for each part.

The heat flux will be applied at the pipe surface only.

The sectional area of circular tubes and elliptical tubes are identical.

The rod has an oval section and equal cross-sectional area for both parts under consideration.

For pipes that are presented in both parts and corrugated rods that place in front of airflow, the boundary layer is assumed to be no slip.

Since the flow has been obstructed by corrugated rods adding to the tube drags, the swirl will be performed and the turbulent boundary layer separation is indispensable. The eddies have been formed, Motivate the flow to move away from the surface of the pipe and this effect changes according to the shape of the pipe in addition to the number of turns in the front pipes of the model used which are diffused away from the outer surface of all pipe types that presented in this work. Consequently, the K-ε must be introduced with the enhancement of the wall effect for more details can be seen by [26].

Therefore, at a low Reynolds number at the near-wall damping effects. For this reason, these models are called ”Low-Reynold More precisely, in the k – ε model, the constants cμ, c1 and c2 are multiplied by fμ, f1 and f2 respectively. Where, this function $0 < f\mu \leq 1$, $1 \leq f1$ and $0 < f2 \leq 1$ and which depend on two local Reynolds numbers depends on Lam-Bremhorst[1981]Low Reynolds k – ε model.

$$f_{\mu=}\left[1-\exp\left(-0.0165R_y\right]^2\left(1+\frac{20.5}{R_t}\right) \quad (4)$$

$$f1=1+\left(\frac{0.05}{f_\mu}\right)^3, f2=1-\exp\left(R_t{}^2\right)^3 \quad (5)$$

$$R_t=\frac{\text{Turbulent Forces}}{\text{Viscous Forces}}=\frac{\rho k^2}{\mu\varepsilon} \quad (6)$$

The wall boundary conditions for u, k and ε are u = 0, k= 0 and ∂ε/∂y= 0. For more details see reference [34-36]. If Rt small refer to that, the viscous effect dominate.

The boundary flow conditions are represented by airflow velocity and the geometry characteristics of each part under consideration and the mutual influence with thermal conditions which are represented by heat flux values and its location as well as temperature distribution through fluid and over solid parts surfaces with controlling the suitable governing equations that presented above are solved numerically by the software program ANSYS 16.1. The numerical solution and simulation by fluent depend on the finite volume method. The hybrid scheme has been used for discretizing the governing flow and energy equations area which perform the computational region. The Reynolds number can be given by:

$$Re=\frac{\rho VD_h}{\mu} \quad (7)$$

Where Dh is hydraulic diameter and μ is dynamic viscosity. The Nusselt Number can be given by

$$Nu=\frac{h\,D_h}{K} \quad (8)$$

And the heat transfer coefficient can be given by

$$h=\frac{1}{As}\int_0^{As}\frac{Q}{T_w-Ta}dA \quad (9)$$

Where Ta represents the airflow temperature in the vicinity region to the surfaces of the tubes and As is the local tubes surfaces area and Tw is the averaged surface tubes wall temperature.

To calculate the pressure drop through the tubes for both presented models, the Zhukauskas pressure drop mode has been used [29].

$$\Delta p=Nx\left(\frac{\rho V_{max}{}^2}{2}\right)f \quad (10)$$

There are very suitable figures to calculate f and x depending on the value of the Reynolds number. For more details see ref. [29].

ReT:is the Reynolds number that charactrises the strenth of the near wall turbulent relative to the viscosity

2. Validations

The validation of the numerical simulation method can be satisfied by comparing the pressure distribution with ref.[26] as shown in figure (3) below. For independence solutions for mesh generation, several trial calculations are done to get the best values of grid refinement at Reynolds equal to 500 and the overall heat flux equal to 3 kW/m^2 velocity 1 m/s with air vas shown in Table 1 below.

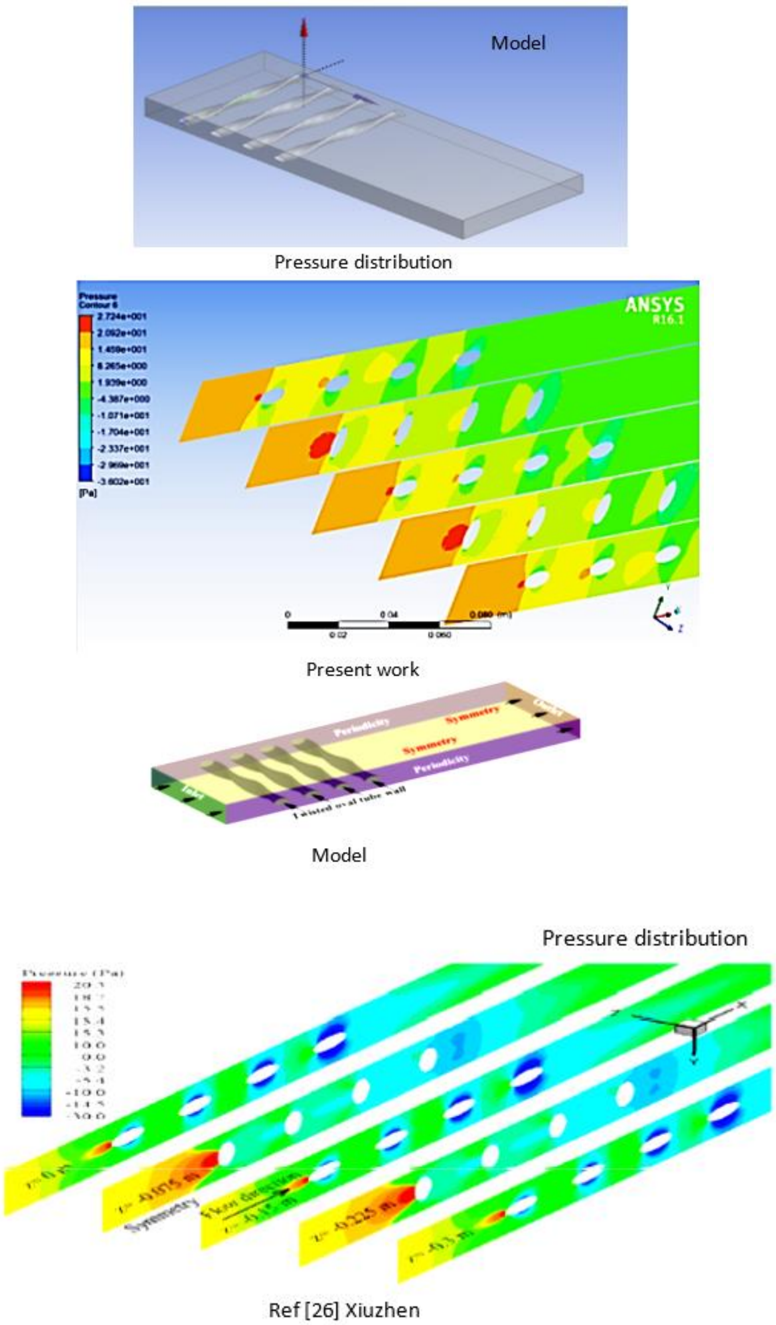


Figure 5 The validation and comparison between the present work and that of Xiuzhen Ref. [26] for pressure distribution along corrugated oval four tubes.

Table 1 Grid generation and corresponding Nusselt number

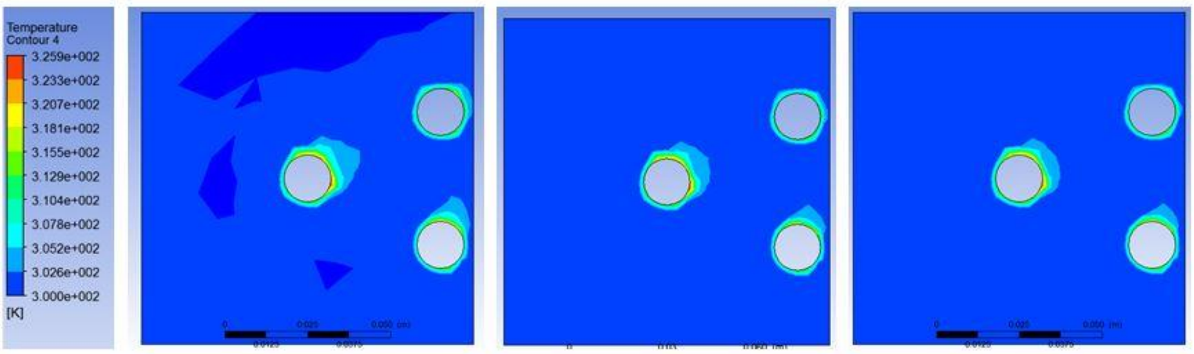
Number	No. of Nods	Nusselts number
1	66540	19.3450
2	233504	19.5187
3	2548723	19.5209

3. Results and discussions

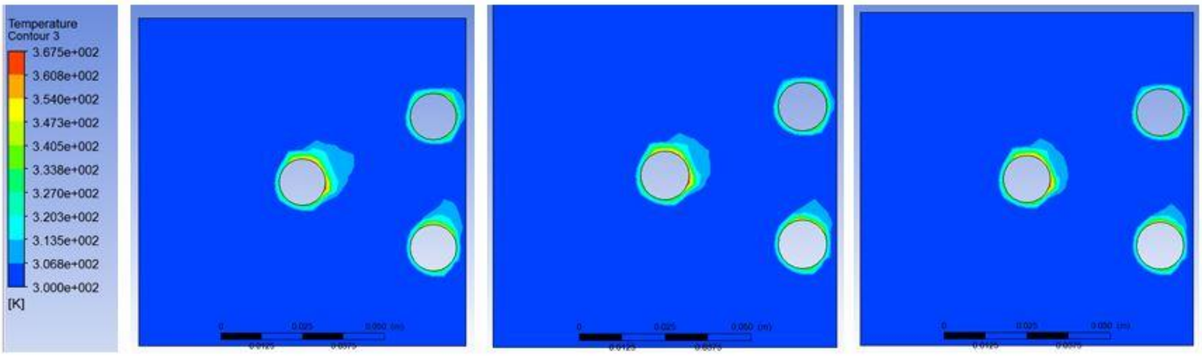
The main objective of the present numerical simulation is to investigate heat transfer and flow characteristics of flow over tube banks in a cross-flow way with introduced corrugated rods in front of crossing airflow with two parts. Therefore, control volumes are presented as shown in figures (3) and (4) containing circular tubes in part (1) and elliptical tubes in part (2). The benefit of corrugated rods is to induce turbulent flow before cross tubes which convey the hot liquid. The turbulent boundary layer intensity will improve the heat transfer dissipation from the outer surface of tubes. However, in this work, the shape of tubes and airflow characteristics is the key incentive for the convection heat coefficient between airflow and tubes surface walls. Moreover, the rods corrugated compensate for a decline in pressure through pipes. So, in this work, the pipes have a constant cross-sectional area along their length. The previous researchers improved the heat exchanger performance by predicting a new shape for tubes producing a high loss due to pressure drops. The innovative result is to improve the thermal performance without increasing the pressure drop. As shown figure (6) represents the variation of temperature distribution around the circular tubes with and without corrugated rods, the temperature of airflow will increase as airflow increases. Row (a) represent the flowing of air on circular tubes with the overall heat flux due to fluid flow through pipe equal to 1.5 KW/m2 and without corrugated rods. While, row (b) refers to the heat flux equal to 3 kW/m2 but, without corrugated rods too. The same values for overall heat flux are applied on the third row (c) with 1.5 kW/m2 and 3 kW/m2 applied with the row (d) with optimization by adding two corrugated oval rods at plane L/2 from tube length. So, at the length L/2 the section of corrugated rods elliptic horizontal section. When the rods are placed in front of tubes, the surface temperature of tubes will decrease with an opposite effect on that of airflow temperature. This refers that, the corrugated rods will enhance the heat transfer rate. Each row presented in figure 6 has been repeated four-time along the tube length for two parts and for all models. The same things have been presented with the second part which has oval tubes as shown in figure (7). The rows (a), (b), (c) and (d) have the same heat flux values as that of the first part which is illustrated in figure 6. Also, both rows (c) and (d) contain corrugated oval rods in front of oval tubes with the same cross-sectional area. For comparison between figures 6 and 7, the oval tubes with corrugated rods introduced more thermal enhancement to the circular tubes. Where the temperature of airflow at the exit region of the second part is greater than that of the first model. Also, the oval tube surface temperature of the second model is less than the surface temperature of the first model.

Air velocity in m/s

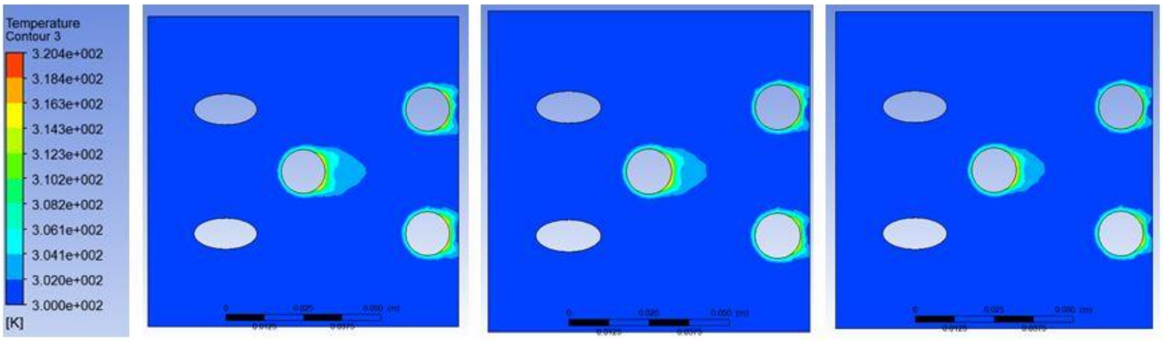
0.6 0.8 1



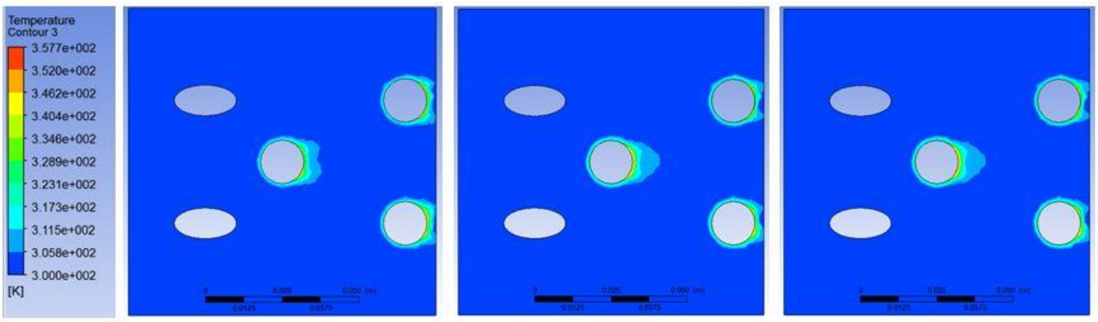
a)



b)



c)

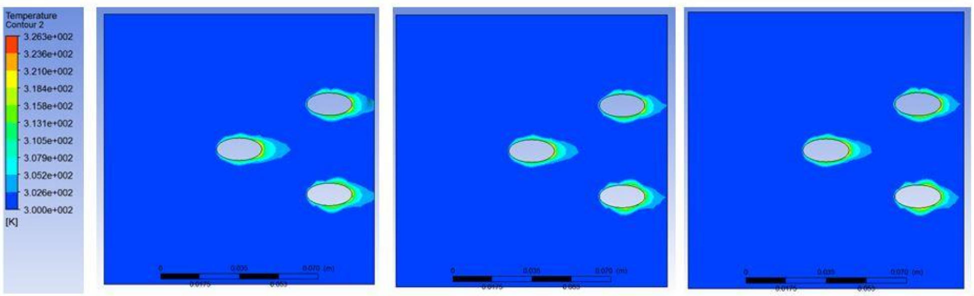


d)

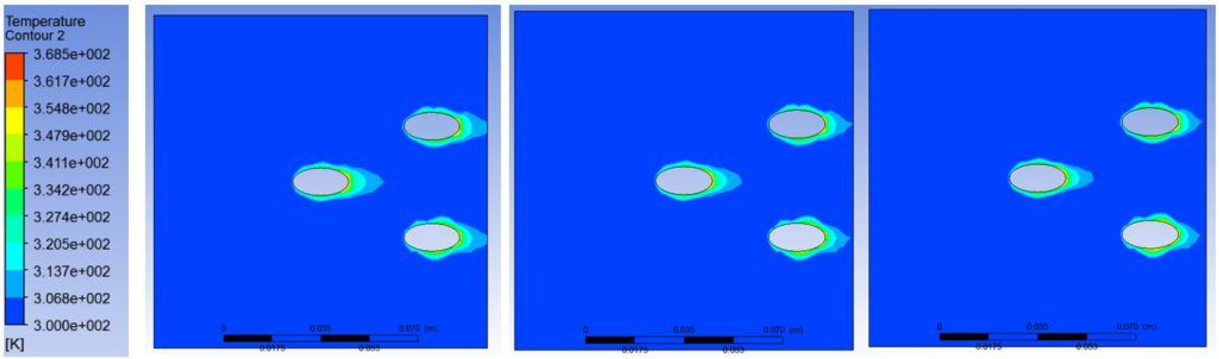
Fig. 6. The Temperature distribution for circular model at different Reynolds number with variation heat flux, where, a and b without corrugated rods while c and d with corrugated rods.

Air velocity in m/s

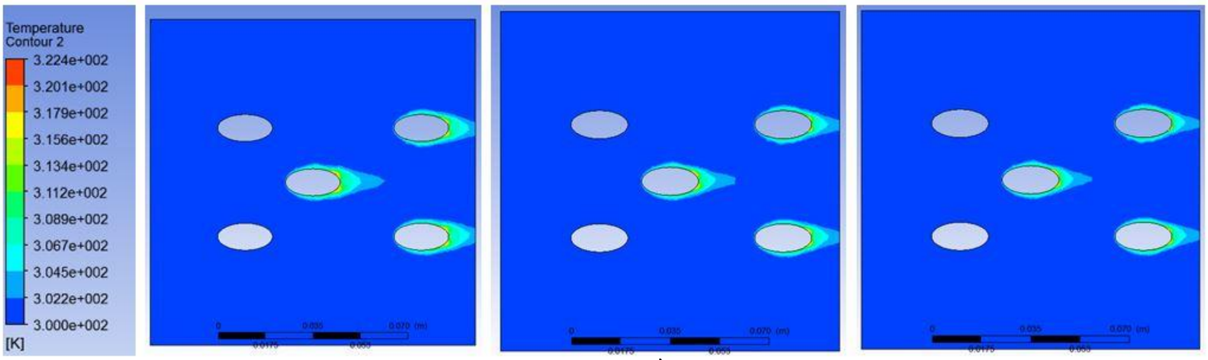
0.6 0.8 1



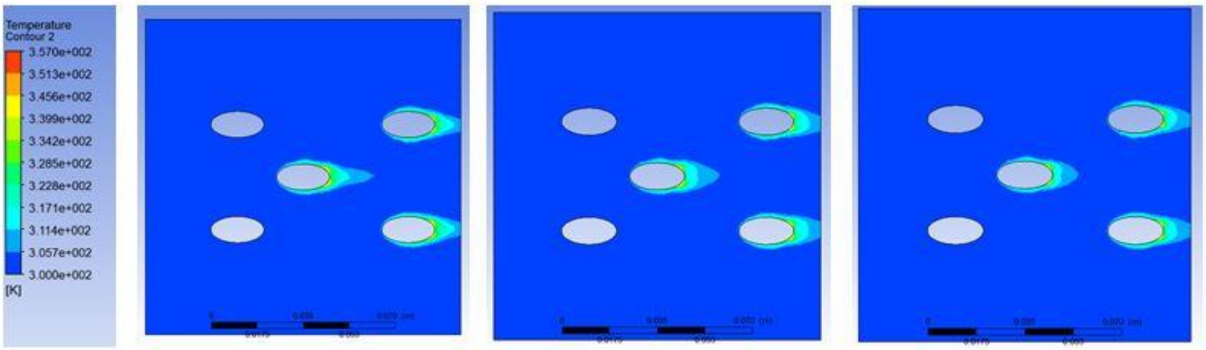
a)



b)



c)



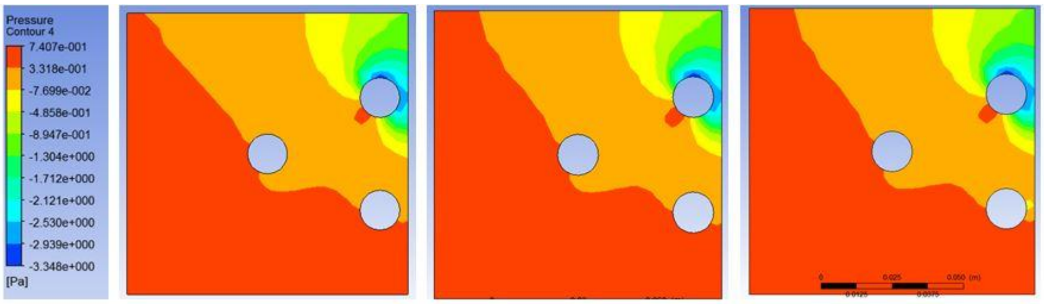
d)

Fig. 7.The Temperature distribution for elliptical at different Reynolds number with variation heat flux, where, a and b without corrugated rods while c and d with corrugated rods

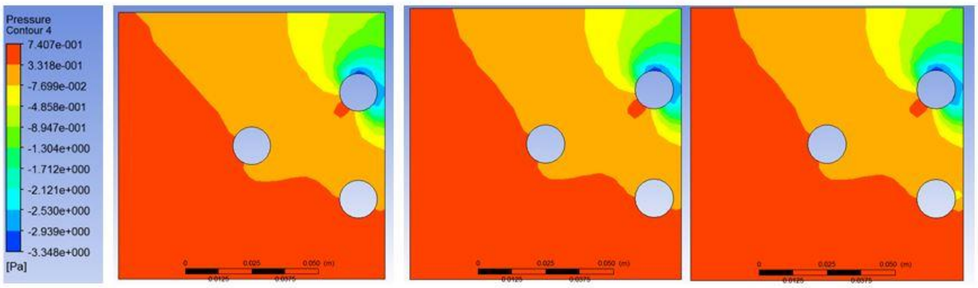
The reason is due to the redistribution of pressure and velocity airflow over circular and oval tubes as shown in figures (8, 9, 10 and 11) respectively. Moreover, the thermal boundary layer in the vicinity region and behind the thickness of the circular tube has been increased due to improving pressure distributions interconnected effect with air-flow velocity increase as shown in figures (6) rows (c) and (d). However, similar behaviour can be seen in figure (7) with the same rows. The corrugated rods not only re-distribution the pressure and velocity, but also, will change the thermal boundary layer thickness. Moreover, these rods will produce swirl flow by pushing airflow sideways direction producing an irregular flow in most the directions given the best region for heat dissipation from tube surfaces to the airflow. This means as increase the number of corrugated pitches will increase irregular with flow-induced more turbulent intensity for flow around tubes. From the above figure, the fluctuation of pressure and temperature is concentrated in the vicinity region of the pipes although the flow is laminar. Therefore, the k-ε equation for a low Reynolds number is a very suitable method to simulate the behaviour of the flow.

Air velocity in m/s

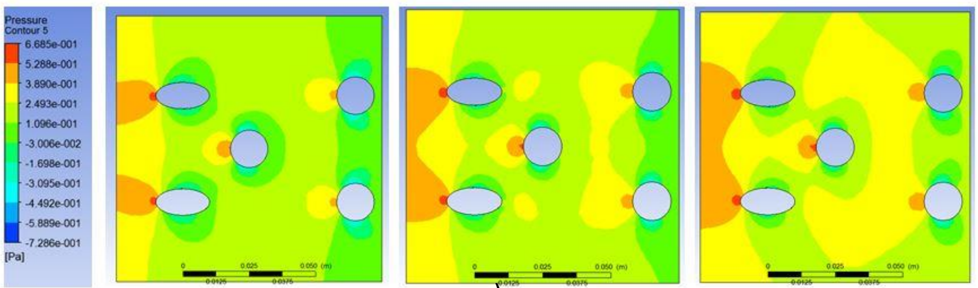
0.6 0.8 1



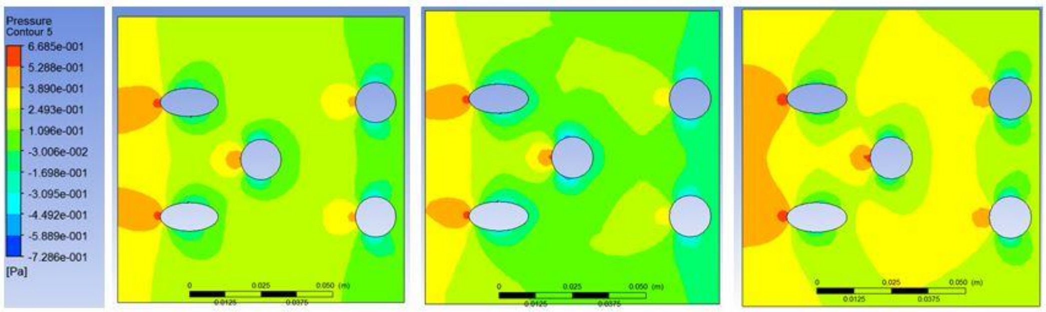
a)



b)



c)



d)

Fig. 8.Pressure distribution for the circular model at different Reynolds numbers with variable heat flux, where, a and b without corrugated rods while c and d with corrugated rods .

Air velocity in m/s

0.6 0.8 1

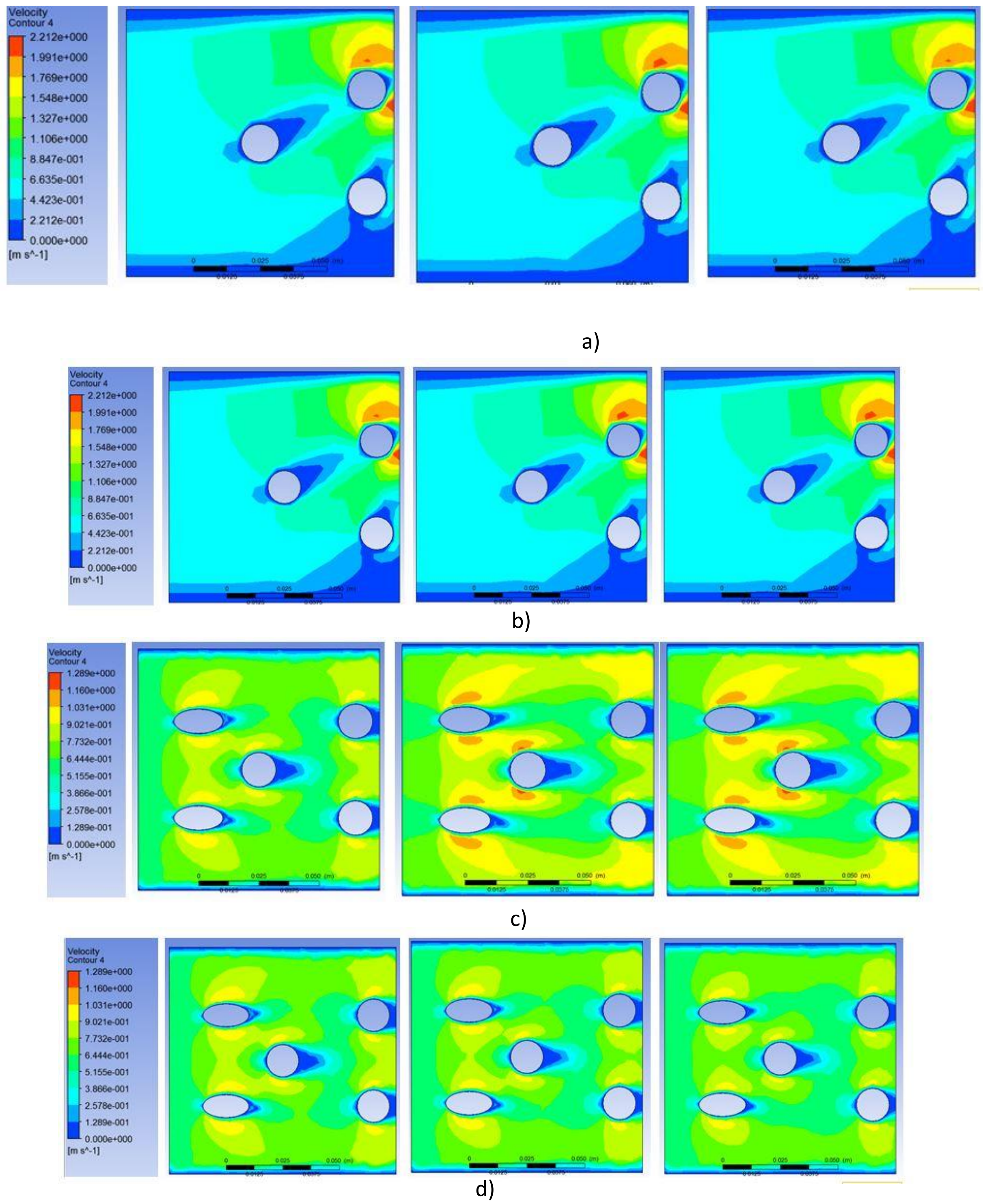
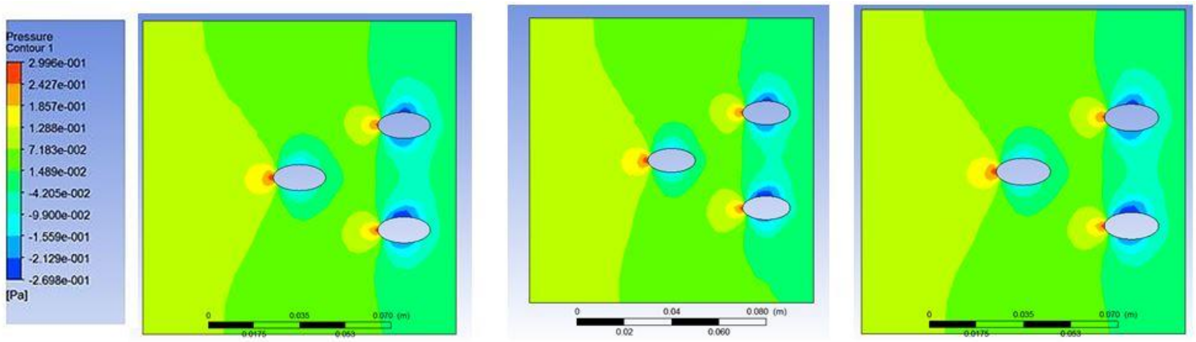


Fig. 9.The velocity distribution for the circular model at different Reynolds numbers with variable heat flux, where, a and b without corrugated rods while c and d with corrugated rods .

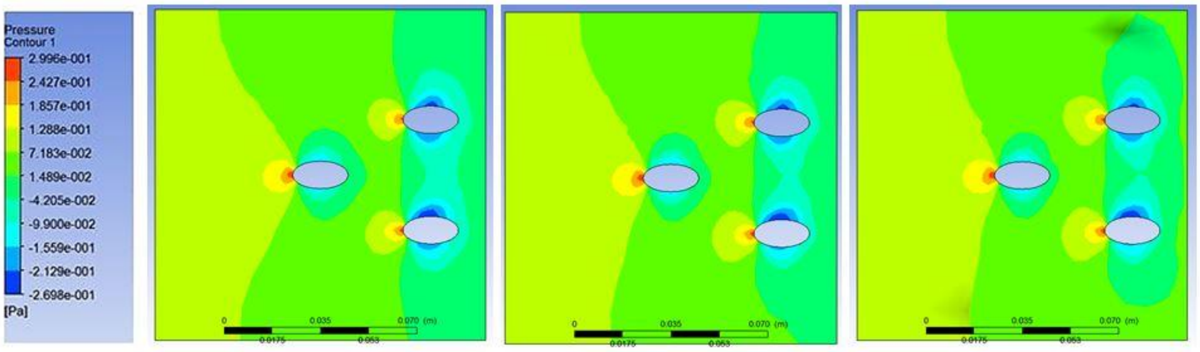
Also, the continuous inclination of the oval along the tube will enhance heat transfer. These results are entirely identical to the results of the reference [38]. Moreover, the wake region is very clear for all figures. Where the wake of the circular section shown in figure (6) is shorter than the wake or ellipse tail shown in figure (7), this refers to that, the temperature difference between the surface area of the tubes with the ellipse section has more diffusivity compared with the circular one. This behaver will increase the heat transfer from tubes to the ambient. Therefore, the pressure distribution and velocity distribution will be modified too in ellipse shape compared with circular tubes and more effective in point of view of enhancement of the heat transfer diffusions as shown in figures (8,9,10 and 11). On the other hand, the rod in front of tubes can be used to redirect the flow toward the pipes and away from the outer walls as shown in figures (9 and 11).

Air velocity in m/s

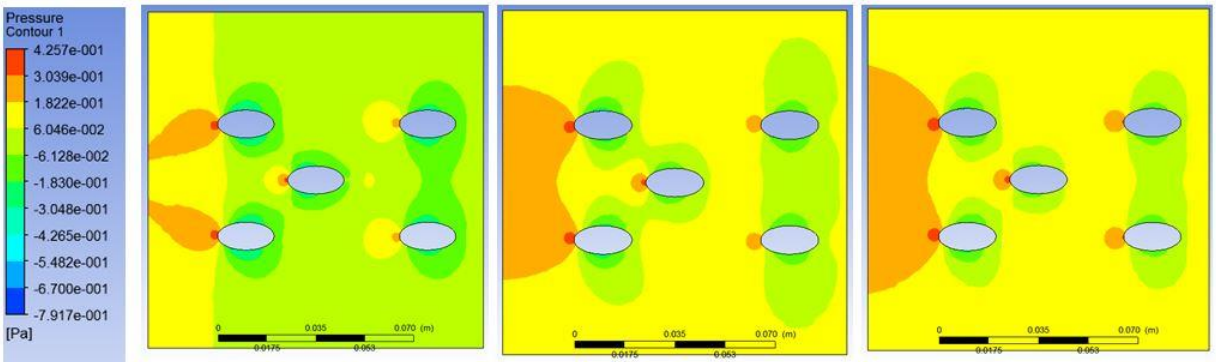
0.6 0.8 1



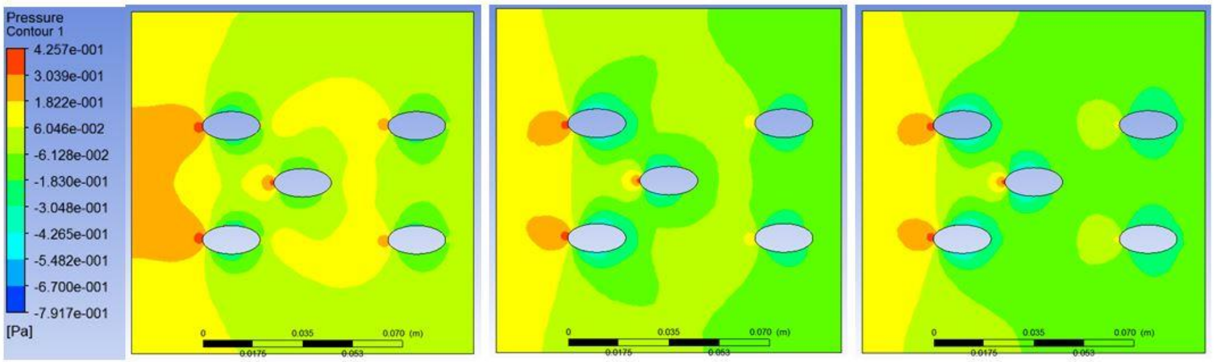
a)



b)



c)

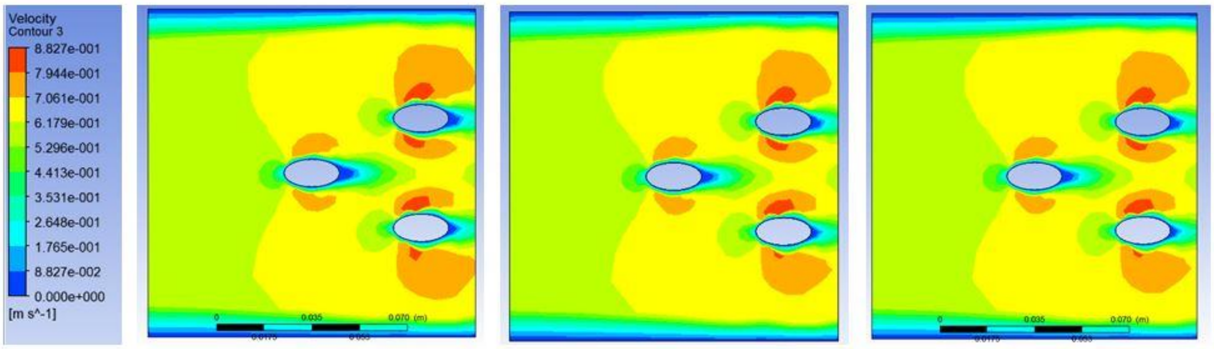


d)

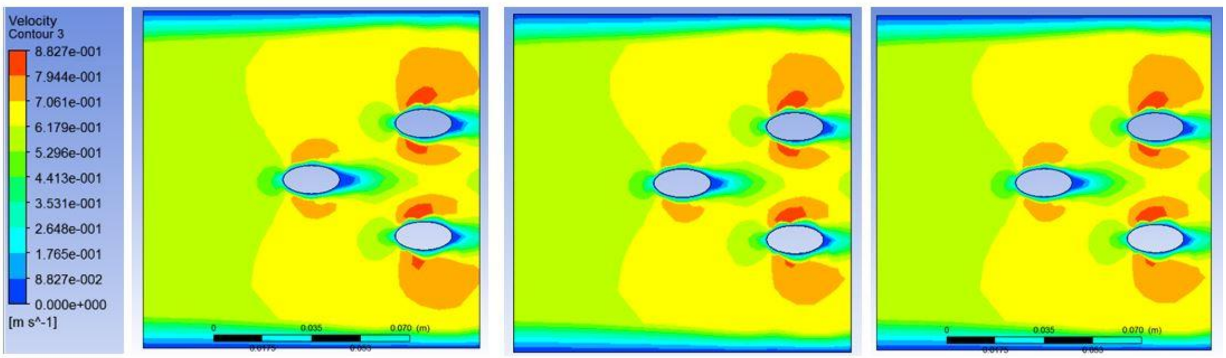
Fig. 10:The pressure distribution for the elliptical model at different Reynolds numbers with variable heat flux, Where a and b without corrugated rods while c and d with corrugated rods .

Air velocity in m/s

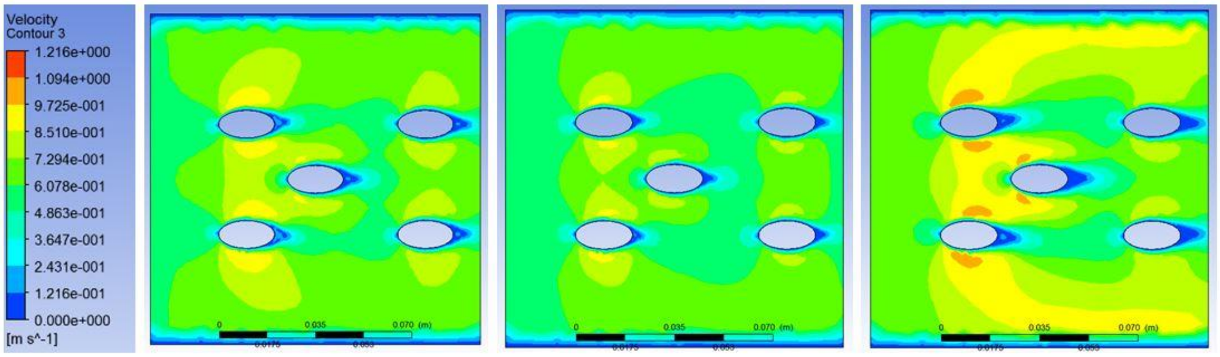
0.6 0.8 1



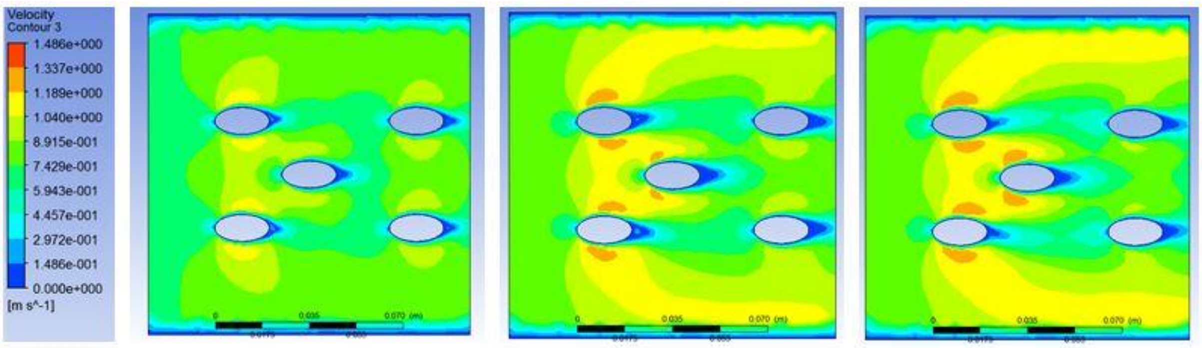
a)



b)



c)



d)

Fig. 11. The velocity distribution for the elliptical model at different Reynolds numbers with variable heat flux, where, a and b without corrugated rods while c and d with corrugated rods .

From the above, it can be observed that the ellipse-shaped tubes have better heat transfer performance as compared to the circular array of tubes with the same cross-section area at a low Reynolds number of airflow over tube surfaces.

Figure 12 presented the relationship between the overall heat flux and the average Nusselt number. As shown in this figure the Nusselt increase as the heat flux increase. These results are expected since the amount of heat transfer will increase as the temperature difference between tubes and surrounding air increases. However, the temperature of the surface of the tube increase as increasing heat flux that applied to it. Moreover, the relations between Reynolds number and pressure drop are illustrated in figure 13 compared with Ref. [28]. While figure 14 represents the relation between the Reynolds number and the average Nusselt number. The results are compared with the experimental results of reference [28], where the cross-section of the tubes for this reference is equal to two parts presented by this work but with a greater surface area. As shown in figure 13 the pressure drop is less than that of Ref. 28. On the other hand, figure 14 shows that almost all the surface area pipes presented by [28] are greater than that of this work but the effect of the corrugated rod on the flow characteristic and thermal boundary layer makes the newly presented model with oval tubes is more efficient.

The rotational direction of an elliptical section changes flows behaviour. The flow characteristics like pressure variation around the ellipse, eddy formation, drag coefficient, as well as heat transfer enhancement will be affected by the orientation of the ellipse. The effect of ellipse orientation is presented by reference [37]. Moreover, the twisted rod will produce mainstream flow to be skewed, two adjacent twisted rods will increase the disturbance of streamlines in front of fluid flow. The type of tube arrangement will obviously affect the quality and properties of the airflow and thus the amount of heat energy dispersed.

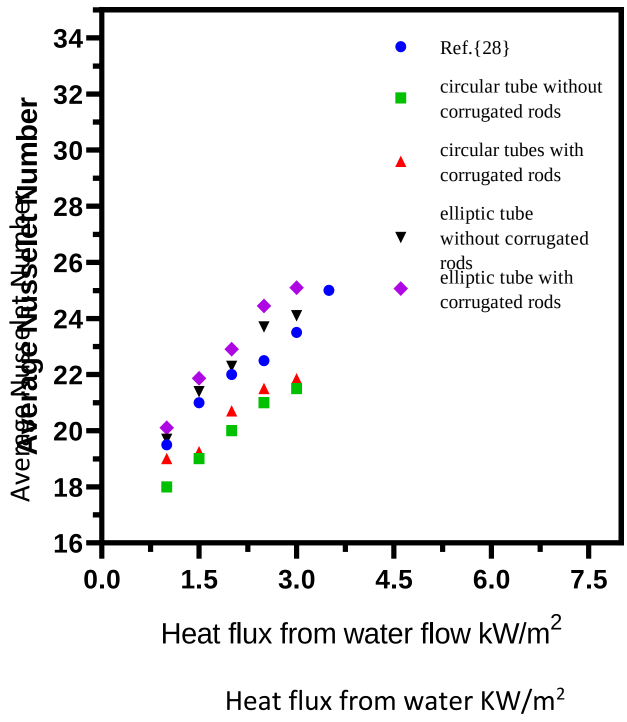


Fig. 12. The relation between the overall heat flux and the average Nusselt number.

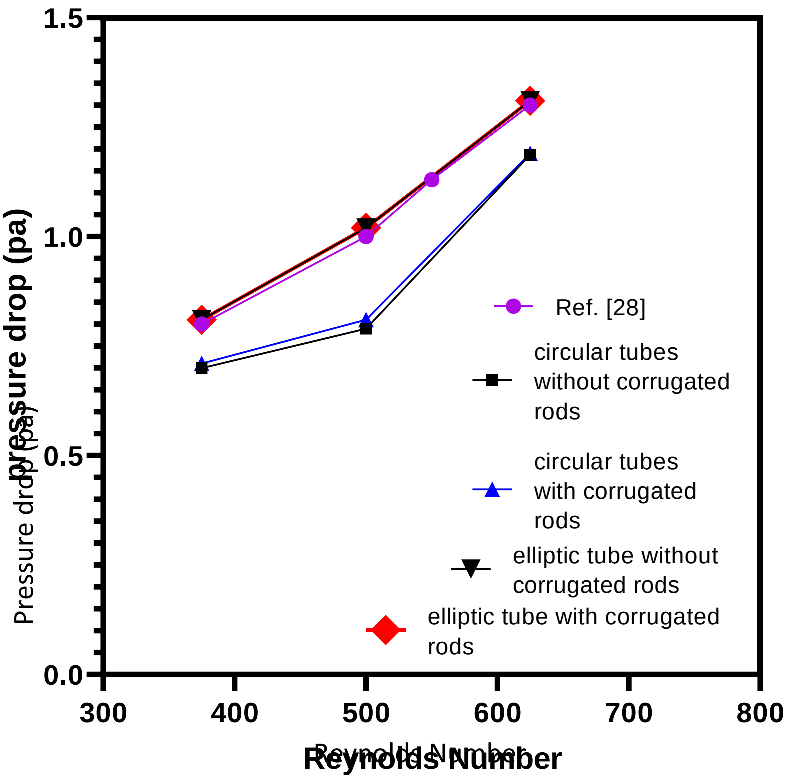


Fig. 13. The relations between Reynolds number and pressure drop.

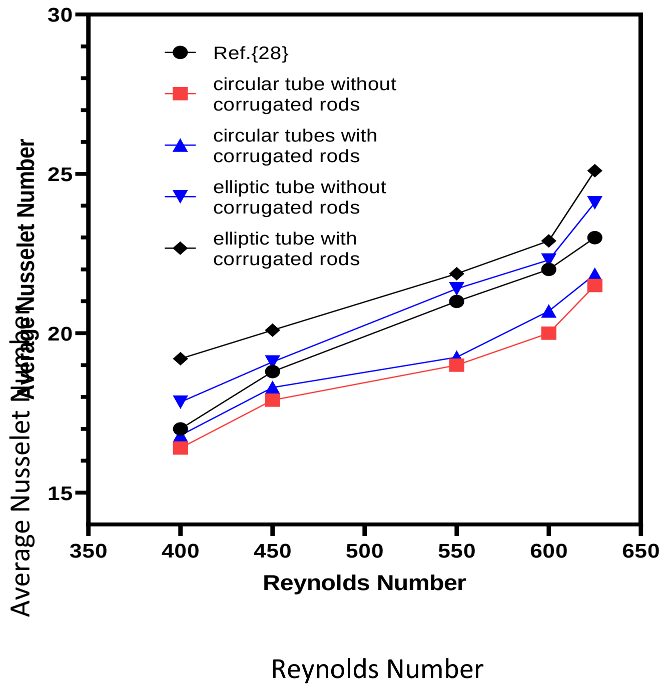


Fig. 14. The relation between Reynolds number and the average Nusselt number.

6. Conclusions

The conclusion of numerical models for two parts, part 1 represent by circular tubes behind two corrugated rods and part 2 with elliptical cross-section tubes with the same corrugated rods in from of it, can be summarized as follow.

1. The model circular tubes and elliptic cross-sectional area with corrugated rods in front of airflow can be applied over a wide range of heat exchanger types with low cost and with superior thermal performance.

2. The heat transfer coefficients for tube banks depend on flow properties and the tube geometries.

3. The shape of corrugated rods has an uneven effect on the pressure distribution of flow around the tubes in front of the flow. So, the separation boundary layer and swirl or any features that increase the turbulent intensity depend on rod geometries.

4. The increase in airflow velocity will enhance the heat dissipation from tube wall surfaces.

5. This work produces flow with eddy at a low Reynolds number to enhance thermal performance without presenting or spending any new energy or inducing extra pressure drop to achieve.

6. The ellipse-shaped tubes have better heat transfer performance as compared to that of the circular array of tubes with the same cross-section area at a low Reyn-olds number of airflow over tube surfaces.

7. The corrugated rods will produce an irregular flow by pushing airflow side-ways and this will give the best region for heat transfer from tube surfaces to the airflow.

8. The Low Reynolds K- ε model for low Reynolds number can be given good results for cross-flow over non-uniform pipe shapes.

9. Rotating the ellipse affects the flow characteristics such as the pressure difference around the ellipse, vortex formation, drag coefficient, as well as heat transfer.

10. The wake of the ellipse rod is clearer and benefits from enhancing the increased heat exchanger performance.

11. The rods in front of tubes can be used as directional control of fluid flow.

Recommendation and Suggestions for Future Work

In order to improve heat exchanger performance, it would be suggested the following points:-

1. Investigated the several materials that used in pipe manufacturing.
2. Numerical simulation of enhancement of heat transfer rate using new fluid flow with different nanofluids.
3. Induced new shapes of corrugated outer pipe surface for increasing turbulent intensity.
4. Study of two concentric tubes containing twisted and opposite walls

REFERENCES

[1] J.I. Corcoles , J.D. Moya-Rico, A.E. Molina, J.A. Almendros-Ibanez "Numerical and experimental study of the heat transfer process in a double pipe heat exchanger with inner corrugated tubes", International Journal of Thermal Sciences, Volume 158, December 2020, 106526. https://doi.org/10.1016/j.ijthermalsci.2020.106526

[2] Zaid S. Kareem و Shahrir Abdullah , Tholudin. Lazim, M.N. Mohd Jaafar, Ammar. Abdul Wahid a "Heat transfer enhancement in three-start spirally corrugated tube: Experimental and numerical study", Chemical Engineering Science, 134,2015, 746-757. https://doi.org/10.1016/j.ces.2015.06.009

[3] Bharat Bhushan Verma, Saurabh Kumar, "CFD Analysis and Optimization of Heat Transfer in Double Pipe Heat Exchanger with Helical-Tap Inserts at Annulus of Inner Pipe" IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684, p-ISSN: 2320-334X, Volume 13, Issue 3 Ver. VII (May- Jun. 2016), PP 17-22. DOI: 10.9790/1684-1303071722

[4] Amin Ebrahimi , Ehsan Roohi "Numerical Study of Flow Patterns and Heat Transfer in Mini Twisted Oval Tubes", international Journal of Modern Physics C,Vol 26 Issue 12, 2015. https://doi.org/10.1142/S0129183115501405

[5] S. Pethkool , S. Eiamsa-ard , S. Kwankaomeng, P. Promvonge, "Turbulent heat transfer enhancement in a heat exchanger using helically corrugated tube" International Communications in Heat and Mass Transfer, Volume 38, Issue 3, March 2011, Pages 340-347. https://doi.org/10.1016/j.icheatmasstransfer.2010.11.014

[6] Fares Djeffal Lyes Bordja , Redha Rebhi, Mustafa Inc, Hijaz Ahmad, Farouk Tahrour , Houari Ameur , Younes Menni ,Giulio Lorenzini, Sayed K. Elagan and Taghreed M. Jawa "Numerical Investigation of Thermal-Flow Characteristics in Heat Exchanger with Various Tube Shapes", Appl. Sci. 2021, 11, 9477. https://doi.org/10.3390/app11209477.

[7] M. E. Nakhchi, J. A. Esfahani, "Numerical investigation of turbulent CuO–water nanofuid inside heat exchanger enhanced with double V cut twisted tapes", Journal of Thermal Analysis and Calorimetry,Springer, 21 may,2020.

[8] John M. Gorman, Kevin R. Krautbauer, Ephraim M. Sparrow, "Thermal and Fluid Flow First-Principles Numerical Design of an Enhanced Double Pipe Heat Exchanger", Applied Thermal Engineering, 2016. https://doi.org/10.1016/j.applthermaleng.2016.06.134.

[9] Gary L. Hammock "Cross-Flow, Staggered-Tube Heat Exchanger Analysis for High Enthalpy Flows", Thesis, The University of Tennessee, Knoxville, May 2011.

[10] Ahmed M. Nagib Elmekawy a, Alaa A. Ibrahim a, Abdalrahman M. Shahin a, Sara Al-Ali a, Gasser E. Hassan b, "Per-formance enhancement for tube bank staggered configuration heat exchanger – CFD Study" Chemical Engineering and Processing - Process Intensification, Volume 164, July 2021, 108392.

[11] Halil Bayram and Gökhan Sevilgen, "Numerical Investigation of the Effect of Variable Baffle Spacing on the Thermal Performance of a Shell and Tube Heat Exchanger", Energies 2017, 10, 1156; doi:10.3390/en10081156.

[12] Dawid Taler, Jan Taler and Katarzyna W˛eglarz," Thermal Calculations of Four-Row Plate-Fin and Tube Heat Exchanger Taking into Account Different Air-Side Correlations on Individual Rows of Tubes for Low Reynold Numbers, Energies 2021, 14, 6978. https://doi.org/10.3390/en14216978.

[13] Dongkyun Lee a, , Joon Ahn b, Seungwon Shin ,"Uneven longitudinal pitch effect on tube bank heat transfer in cross flow "Applied Thermal Engineering 51 (2013) 937e947. https://doi.org/10.1016/j.applthermaleng.2012.10.031

[14] Chidanand K. Mangrulkar a,*, Ashwinkumar S. Dhoble b , Amey R. Deshmukh c , Sachin A. Mandavgane" Numerical investigation of heat transfer and friction factor characteristics from in-line cam shaped tube bank in crossflow. Applied Thermal Engineerin, 2016. https://doi.org/10.1016/j.applthermaleng.2016.08.174

[15] Nidal H. Abu-Hamdeh, Rashad A. R. Bantan, Ashkan Alimoradi "Thermal and hydraulic performance of the twisted tube bank as a new arrangement and its comparison with other common arrangements" Journal Pre-proof, Volume 157, May 2020, Pages 46-57.https://doi.org/10.1016/j.cherd.2020.03.003

[16] Ahmadali Gholamia,, Hussein A. Mohammedc, , Mazlan. Wahidb , Mehdi Khiadanid "Parametric design exploration of fin-and-oval tube compact heat exchangers performance with a new type of corrugated fin patterns" International Journal of Thermal Sciences, Volume 144, October 2019, Pages 173-190. https://doi.org/10.1016/j.ijthermalsci.2019.05.022

[17] S. Eiamsa-ard, K. Kiatkittipong , W. Jedsadaratanachai, "Heat transfer enhancement of TiO2/water nanofluid in a heat exchanger tube equipped with overlapped dual twisted-tapes " Engineering Science and Technology, an International Journal, 2015. https://doi.org/10.1016/j.jestch.2015.01.008

[18] Haolin Ma, Dennis E. Oztekin, Seyfettin Bayraktar, Sedat Yayla and Alparslan Oztekin, "Computational Fluid Dynamics and Heat Transfer Analysis for a Novel Heat Exchanger", Journal of Heat Transfer, 2015, MAY 2015, Vol. 137. https://doi.org/10.1115/1.4029764

[19] C K Mangrulkar, J D Abraham, A S Dhoble," Numerical studies on the near wall y + effect on heat and flow characteristics of the cross flow tube bank" Journal of Physics: Conference Series, 2019. **DOI** 10.1088/1742-6596/1240/1/012110

[20] Joseph Jabbour a,b,c, Serge Russeil a,c* , Mohammed Mobtil a,c , Daniel Bougeard a,c Marie-France Lacrampe b,c, Patricia Krawczak, "High performance finned-tube heat Exchangers based on filled polymer" Elsevier, 2019. https://doi.org/10.1016/j.applthermaleng.2019.04.036

[21] Vinous M. Hameed , Sarah M. Awad, "Experimental investigation of double pipe heat exchanger temperature distribution" , International Journal of Multidisciplinary Sciences and Advanced Technology Vol 2 No 1 (2021) 10–1.

[22] K. Ravikumar, Ch.Naga Raju , Meera Saheb , "CFD Analysis of a Cross-flow Heat Exchanger with Different fin thickness" International Journal of Dynamics of Fluids. ISSN 0973-1784 Volume 13, Number 2 (2017), pp. 345-362.

[23] Yongqing Wang, Xin Gu, Zunlong Jin ⇑, Ke Wang, "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, Volume 93, February 2016, Pages 584-594. https://doi.org/10.1016/j.ijheatmasstransfer.2015.10.018

[24] B. Parikshit ⇑ , K.R. Spandana, V. Krishna, T.R. Seetharam, K.N. Seetharamu, "A simple method to calculate shell side fluid pressure drop in a shell and tube heat exchanger" International Journal of Heat and Mass Transfer, 84, (2015), 700-712. https://doi.org/10.1016/j.ijheatmasstransfer.2015.01.068

[25] Yifan Zhanga, Fubao Zhoua, Jianhong Kang, "Flow and heat transfer in drag-reducing polymer solution flow through the corrugated tube and circular tube" Engineering, Volume 174, 25 June 2020, 115185. https://doi.org/10.1016/j.applthermaleng.2020.115185

[26] Xiuzhen Li a,b,c,d, Dongsheng Zhu a,b,c,⇑, Yingde Yin a,b,c, Aimin Tu a,b,c, Shijie Liu, "Parametric study on heat transfer and pressure drop of twisted oval tube bundle with in line layout" , International Journal of Heat and Mass Transfer 135 (2019) 860–872. https://doi.org/10.1016/j.ijheatmasstransfer.2019.02.031

[27] LI Xiuzhen1, WANG Lin, FENG Rong, WANG Zhanwei, ZHU Dongsheng," Thermal-Hydraulic Characteristics of Twisted Elliptical Tube Bundle in Staggered Arrangement", Journal of Thermal Science, Institute of Engineering Thermophysics, CAS and Springer-Verlag GmbH Germany, part of Springer Nature 2021.

[28] M. Ishak, Tahseen Ahmad Tahseen And Md. Mustafizur Rahman, " Experimental Investigation on Heat Transfer and Pressure Drop Characteristics of Air Flow Over A Staggered Flat Tube Bank In Crossflow", International Journal of Automotive and Mechanical Engineering, Volume 7, pp. 900-911, January-June 2013. **DOI:** https://doi.org/10.15282/ijame.7.2012.7.0073

[29] Pengxiao Li , Zhichun Liu , Wei Liu, Gang Chen" Numerical study on heat transfer enhancement characteristics of tube inserted with centrally hollow narrow twisted tapes", International Journal of Heat and Mass Transfer 88 (2015) 481–491. https://doi.org/10.1016/j.ijheatmasstransfer.2015.04.103

[30] Danar Susilo Wijayanto, Soenarto, Mochamad Bruri Triyono, Wuri Prasetyo, Indah Widiastuti. Analysis of Longitudinal Finned Pipes in Cross-Flow Heat ExchangerInternational Journal of Heat and Technology Vol. 39, No. 6, 2021, pp. 1909-1916. DOI: https://doi.org/10.18280/ijht.390627

[31] Agung Tri Wijayanta, Mirmanto and Muhammad Aziz, Heat transfer augmentation of internal flow using twisted tape insert in turbulent flow", HEAT TRANSFER ENGINEERING 2019 Taylor & Francis Group, LLC .https://doi.org/10.1080/01457632.2019.1637149

[32] Ravindra Kumar Pasupuleti, Manindra Bedhapudi, Subba Reddy Jonnala, Appa Rao Kandimalla. Computational Analysis of Conventional and Helical Finned Shell and Tube Heat Exchanger Using ANSYS-CFD. International Journal of Heat and Technology, Vol. 39, No. 6, 2021, pp. 1755-1762
1755-1762. DOI: https://doi.org/10.18280/ijht.390608

[33] Wisam Abed Kattea Al-Maliki, Nabeel Sameer Mahmoud, Hussein MH Al-Khafaji, Falah Alobaid, Bernd Epple "Design and Implementation of the Solar Field and Thermal Storage System Controllers for a Parabolic Trough Solar Power Plant"Applied Sciences, 2021. **https://doi.org/10.3390/app11136155**

[34] C K Mangrulkar, J D Abraham, A S Dhoble, "Numerical studies on the near wall y+ effect on heat and flow characteristics of the cross flow tube", 2019 J. Phys.: Conf. Ser. 1240 012110. **DOI** 10.1088/1742-6596/1240/1/012110

[35] Hasanain A. Abdul Wahhab Wisam Abed Kattea Al-Maliki. "Application of a Solar Chimney Power Plant to Electrical Generation in Covered Agricultural Fields" IOP Conf. Series: Materials Science and Engineering 671 (2020) 01. **DOI** 10.1088/1757-899X/671/1/012137

[36] J. P. Holman," Heat Transfer", book, Copyright © 2010 by The McGraw-Hill Companies, Inc.

[37] Eyub Canli, Sercan Dogan, Muammer Ozgoren, Ozgur Solmaz and Gokhan Ozseker, " Numerical investigation of the wake flow of an ellipse cylinder for different angles of attack", Conference: The 6th International Congress of Energy and Environment Engineering and Management (CIIEM15), At: Paris, FRANCE,July 2015.

[38] Siddhant Singh Yogesh, Arun Saco Selvaraj, Dinesh Kumar Ravi, Thundil Karuppa Raj Rajagopal, " Heat transfer and pressure drop characteristics of inclined elliptical fin tube heat exchanger of varying ellipticity ratio using CFD code", International Journal of Heat and Mass Transfer,119 , (2018),26-39. https://doi.org/10.1016/j.ijheatmasstransfer.2017.11.094