RESEARCH ARTICLE OPEN ACCESS

Numerical Investigation of Heat Transfer from Two Different Cylinders in Tandem Varieties Arrangements

Mofreh M. Nassief[1].
[1]*Associated Professor,Faculty of engineering, Zagazig University, Zagazig, Egypt*

ABSTRACT
A two dimensional technique has been studied numerically to predict the heat transfer from two different cylinders in tandem arrangement (one is circular and the other is elliptical) using finite element technique with RNG k-ε turbulent model, taking into consideration the effect of gap ratio (L/Deq) and Reynolds number , where the distance between the centers of cylinders is L (L=30 mm and 37 mm), the equivalent diameter of cylinder is Deq=22.5mm and the range of Reynolds number is $2x10^3 < Re_{eq} < 21x10^3$.The commercial CFD software FLUENT was used to get the thermofluid characteristics (temperature, velocity, kinetic energy and pressure contours ,coefficient of friction , heat transfer coefficient , Stanton number …… etc) of the flow around cylinders. The dependency of the heat transfer coefficient, Stanton number (St_a), pressure drop, and friction factor for circular and elliptical cylinders on the gap ratio is clear from the results. Results show that, for circular cross section, the heat transfer coefficient is increased as velocity, and gap ratio increase. On the other hand St_a decreased as velocity increase. The pressure drop and hence the friction factor increase for circular cylinder as gap ratio increases. For elliptical tube the heat transfer and St_a are relatively equal to that for circular one at the same gap ratio, but the overall power consumption and friction factor for elliptical tube is lower than that of circular one. As the elliptical cylinder fixed on the second position the heat transfer and St_a increase, on the other hand the pressure drop and hence the friction factor decreases. For all studied arrangements the highest heat transfer is observed for the arrangement of circular-first and elliptical-second cylinder and the minimum pressure drop and hence the friction factor are for the elliptical one.

***Keywords*:** Circular and Elliptical Cylinders, Tandem Arrangement, Numerical Solution, Heat Transfer Rate.

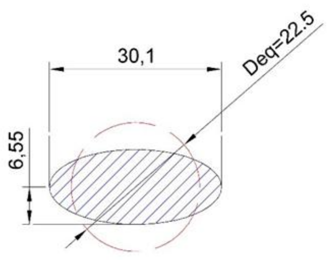
I. INTRODUCTION

The applications of convection heat transfer from bluff bodies can be noticed for examples in heat exchangers, cooling towers, oil and gas pipelines and electronic components cooling. The performance of heat exchangers can be improved to perform a certain heat transfer duty by heat transfer enhancement techniques and due to the curvature of the tubes, as fluid flows through curved tubes, centrifugal force is generated. A secondary flow induced by the centrifugal force has significant ability to enhance the heat transfer rate. Single-phase heat transfer characteristics in the helically coiled tubes have been widely studied by researchers both experimentally and theoretically. There are several advantages of the numerical method, e.g. large volume of the results obtained from the parametric studies, low cost. In addition, due to some complexity of the heat transfer processes in the helically coiled tubes, experimental studies are very difficult to handle and numerical investigations are needed, Naphon [1]. Ahmed et al.[2] demonstrated an experimental and numerical study to clarify fluid flow characteristics and pressure drop distributions of a cross-flow heat exchanger employing staggered wing- shaped tubes at different angles of attack, they concluded that the values of pressure drop coefficient increased with the angle of attack from 0^o to 45^o, while the opposite was true for angles of attack from 135^o to 180^o. Kyung Kim. [3] have demonstrated that some of the parabolic type problems encountered in such branches of engineering as heat conductions with a moving source can be analyzed successfully by means of finite element method with adapted mesh generation technique is implemented. Sadri et al.[4] investigated that one type of applicable analytical method, differential transformation method (DTM), is used to evaluate the efficiency and behavior of a straight fin with variable thermal conductivity and heat transfer coefficient also fins are widely used to enhance heat transfer. Kanta et al. [5] presented an analytical solution of one dimensional free convection flow past an infinite vertical cylinder in stratified fluid medium. The dimensionless unsteady coupled linear governing equations are solved by Laplace transform technique. Hwang et al.[6] stated that among tubular heat exchangers ,fin tube types are the most widely used in refrigeration and air condition equipment. Efforts to enhance the performance of these heat exchangers included variations in the fin shape from a plain fin to a slit and louver type. Elsayed et al. [7] have studied the effect of a slotted shield on thermal performance of a heat sink, they found that increasing the slot width enhances the flow performance over the heat sink and this improvement continues as the number of slots increases, but the thermal performance on the other hand, decreases and the slots work as a flow by pass and create jets to destroy eddies and vortices created by the shield. Elsayed et al. [8] have investigated experimentally and numerically the heat transfer and pressure drop characteristics of a set of pin- fins with uniform heat flux. They found that heat sinks having fin heights of 20 and 29 mm operated at lower Reynolds number, reached minimum value for thermal resistance when the fin 10x10 which means the optimum number of fins for this case, also friction factor increased with a decrease in by pass flow area.

From the above discussion, it can be deduced that much less attention has been given to the studies for flow past two cylinders of different cross section with the variation of Re_a and gap ratio. With these points in mind, the present study focuses on **A)** the effect of different cross section area of tube on the thermofluid characteristics. **B)** the effect of Re_a and gap ratio on heat transfer rate**.** Also, the significance of this work is to have a better understanding of the convective heat transfer characteristics from heat exchanger tubes, with the effect of changing the cross section area of tube from cylindrical to elliptical, when subjected to cross flow of air. By using RNG k-ε turbulent model (some results of this work are compared with the experimental results of Mohamed [13]) and a computational fluid dynamics (CFD) program, FLUENT [9], the preceding points will be analyzed in detail.

II. PROBLEM DESCRIPTION AND BOUNDARY CONDITIONS

The cross section profile of the cylinder comprised two parts of circular and elliptical ones with the same equivalent diameter. The cylinder have identical diameters equal to D = 22.5 mm and the corresponding major and minor axis of the elliptical one is 30.1 mm and 13.1 mm respectively as shown in Fig 1. The distance between their centers are 30 mm and 37 mm. Characteristic length for this tube is the diameter of an equivalent circular cylinder whose circumferential length is equal to that of the elliptical cylinder as shown in Fig. 2.



All dimensions are in mm.

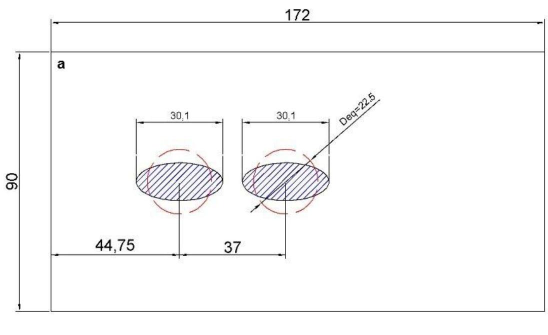
Fig. 1. Circular and elliptical tubes cross sectional dimensions.

Since, the tube length is much greater than its equivalent diameter, the flow across the tube is considered two-dimensional. The typical solution domain and the cylinder boundary definition and nomenclature used in this work are shown in Fig. 3. The numerical solution is carried out by solving the governing equations of mass, momentum and energy under the following assumptions; the flow is incompressible, steady and turbulent, fluid properties are constant, the effect of buoyancy force and radiation are neglected, FLUENT [9],so the governing equations are:

$\frac{\partial}{\partial x_i}(\rho V_i) = 0.0$	(1)
$\frac{\partial}{\partial x_j}(\rho V_i V_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j}$	(2) (2)
$\frac{\partial}{\partial x_i}[V_i(\rho E + p)] = \frac{\partial}{\partial x_i}\left(k \frac{\partial T}{\partial x_i}\right)$	(3) (3)

Where, *i*: is a tensor indicating 1 (V_1=u) and 2 (V_2=v), τ_{ij} is the viscous stress tensor, and k is the fluid effective thermal conductivity.

The software Commercial CFD FLUENT 6.3.26 was used to get the solution of the problem. The turbulence model RNG k-ε is more effective to get the characteristics of flow strain rate. This significantly improves the accuracy for rapidly strained flows, such as the case in reactive swirling flows, Ahmed et al. [2]. Also the RNG k-ε model shows better performance than the standard k-ε model in the prediction of heat transfer as examined by Zehua [10] and assured by many other authors. The RNG k-ε model has shown an excellent agreement between numerical and experimental results for an isothermal flow over backward facing step Yakhot [11]. The RNG k-ε model with enhanced wall treatment is capable of predicting the flow separation and recirculation behind a bluff body. As a consequence, the turbulence model used in this study is the RNG k-ε model with Enhanced Wall Function approach in the near-wall regions to fit the wall boundary conditions, Ahmed et al.[2].



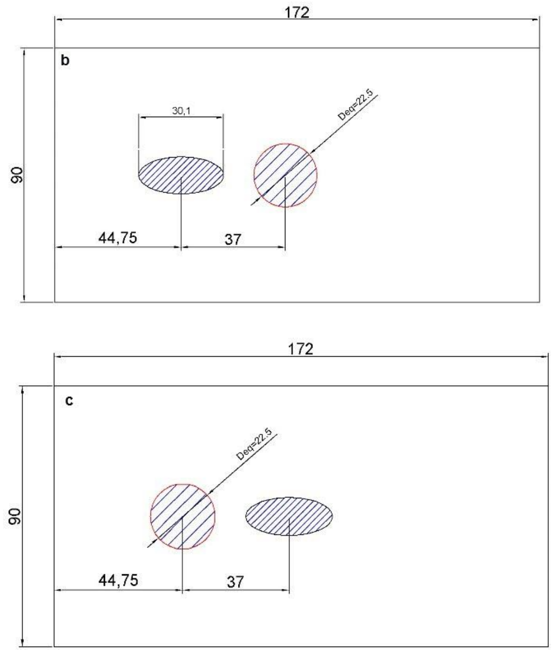


Fig. 2. Schematic plane of the test section, all dimensions are in mm.

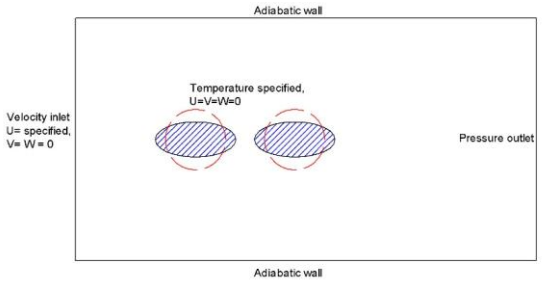


Fig.3. Boundary conditions for the numerical domain.

2.1. Mesh Generation and Discretization:

The problem geometry and mesh generation of the computational model had been done using GAMBIT 2.4.6 and boundary layer meshing scheme was used with mesh refining near walls and sharp edges. As shown in Fig. 4 the computational domain mesh with using finite element discretization method and second order upwind schema for momentum, turbulent kinetic energy, and turbulent dissipation rate was applied, also a simple-based solution algorithm of the velocity–pressure combined with a segregated solver was applied. With running the algorithm ,we find that the residuals of the energy equation and other equations reach 10^{-7} and10^{-4} respectively.

As we know the pressure drop coefficient, P_{dc},.is the ratio between the total pressure drop of the flow over tubes to its dynamic pressure as in Eq.(4).

$$P_{dc} = \frac{2\Delta p_a}{\rho_{af} V_{ai}^{2}} \tag{4}$$

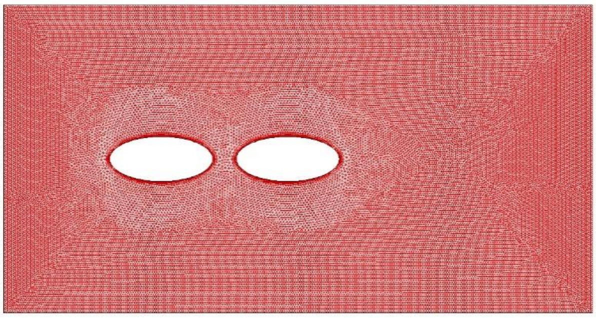


Fig. 4. Mesh Configurations for the Domain of the Tubes.

The Reynolds number, Re_a was given by:

$$Re_a = \frac{\rho_{af} V_{ai} D_{eq}}{\mu_{af}} \tag{5}$$

The Nusselt and Stanton number was attributed as:

$$Nu_a = \frac{h_a D_{eq}}{k_{af}} \tag{6}$$

$$St_a = \frac{h}{\rho V C_p} \tag{7}$$

Where, D_{eq} is the outer equivalent diameter of the tube and k_{af} is the air thermal conductivity, W/m.K and h is the air-side average heat transfer coefficient W/m^2.K.

III. RESULTS AND DISCUSSION

In this section a detailed analysis of different flow patterns, revealed in this study is presented in terms of heat transfer coefficient, Nusselt number, friction coefficients, streamline patterns, temperature, velocity and kinetic energy contours for all study cases. According to Mofreh [12] and for the purpose of the validation of the solution procedure, it is essential that numerical simulations be compared with experimental data. Fig. 5 compares the Nusselt number of elliptical cylinder with the results of Mohamed [13]. This figure shows a good agreement between the present numerical results and experimental measurements. It can therefore be concluded that the CFD code can be used to solve the flow field for similar geometries and conditions.

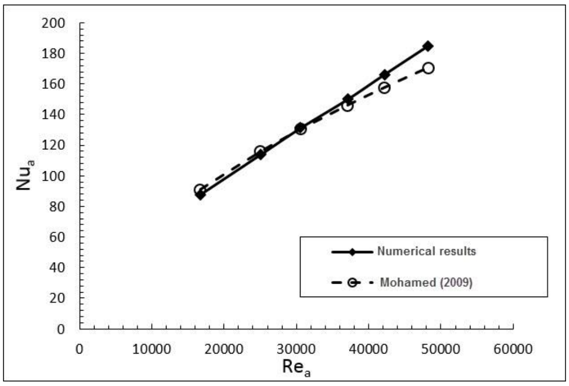


Fig. 5. Comparison of experimental and numerical heat transfer.

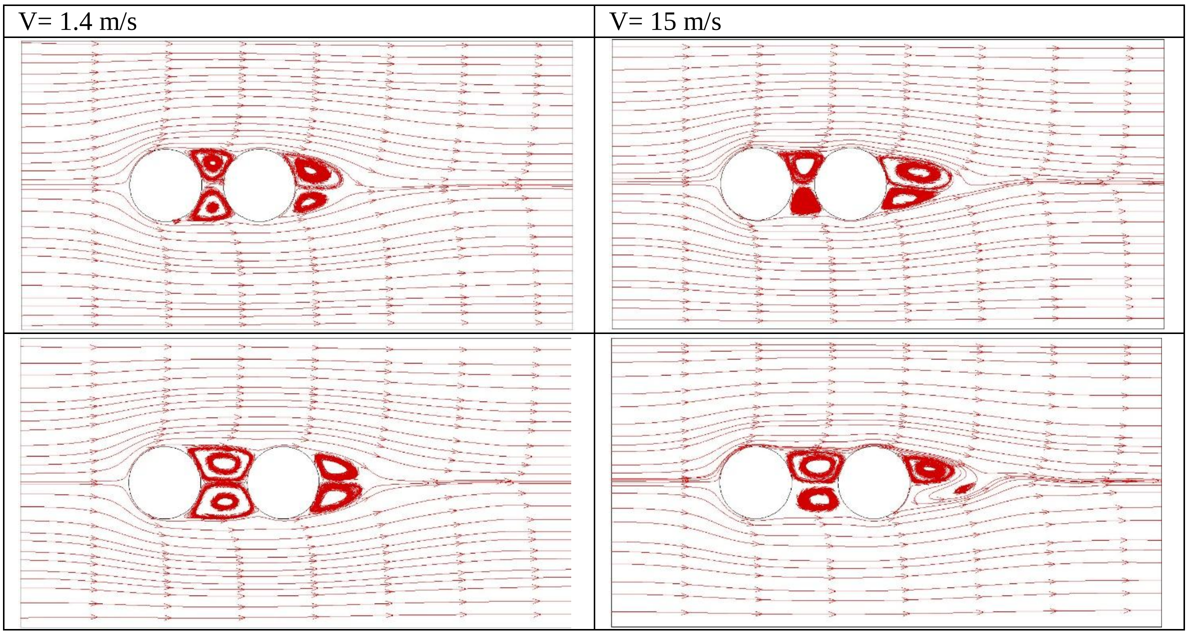
The streamlines patterns of the tested tubes at different arrangements are shown in Fig. 6. As shown in this Figure, there are flow separation-zones at the downstream surfaces of the tubes. Due to travelling of the boundary layer far enough against an adverse pressure gradient, flow separations are occurred and this makes the velocity of the boundary layer falls, almost, to zero, and also as a result of the increase in pressure drag caused by the flowing air between the upstream and downstream surfaces of the tube, as in Ahmed et al. [2]. On the other hand, due to increasing of gap between cylinders the shear layers move further inside the gap and due to this movement we observe wider stagnant recirculation For the elliptical tube the flow separation at the second rear tube disappears due to the surface geometry. On the other hand, the flow separation increase on the rear region for circular one.

Velocity contours (m/s) for different arrangements is shown in Fig. 7. as shown in the figure, for the arrangements of circular followed by the elliptical, the level of turbulence and formation of vortices through the passages of the tubes are higher compared to these other arrangements and due to increasing velocity the level of turbulence and formation of vortices through the passages of the cylinders are more for small gap compared to the other gap.

Fig.8 presents the turbulent kinetic energy contours through the different tubes arrangements for both low and high Re_a, which is defined as the mean kinetic energy per unit mass associated with eddies in turbulent flow. It is found that the turbulent kinetic energy is larger in magnitude in the case of the higher Re_a. This, in turn, enhances the convective heat transfer coefficient h_a and consequently increases Nu_a.

Temperature contours for the air flowing across the tubes are depicted in Fig. 9. The maximum difference between the inlet and outlet air temperatures and consequently the maximum quantity of heat transferred are obtained for the arrangement of circular tube followed by elliptical one, this due to the high differences in pressure between inlet and outlet

Fig.10 demonstrates the static pressure contours through the tubes. It can be seen that the pressure has the highest values at the stagnation point on the frontal portion of each tube, this is because the flow velocity tends to be zero at this point. The static pressure drop for the bundle with elliptical and circular cylinder or two tandem circular cylinder is high.



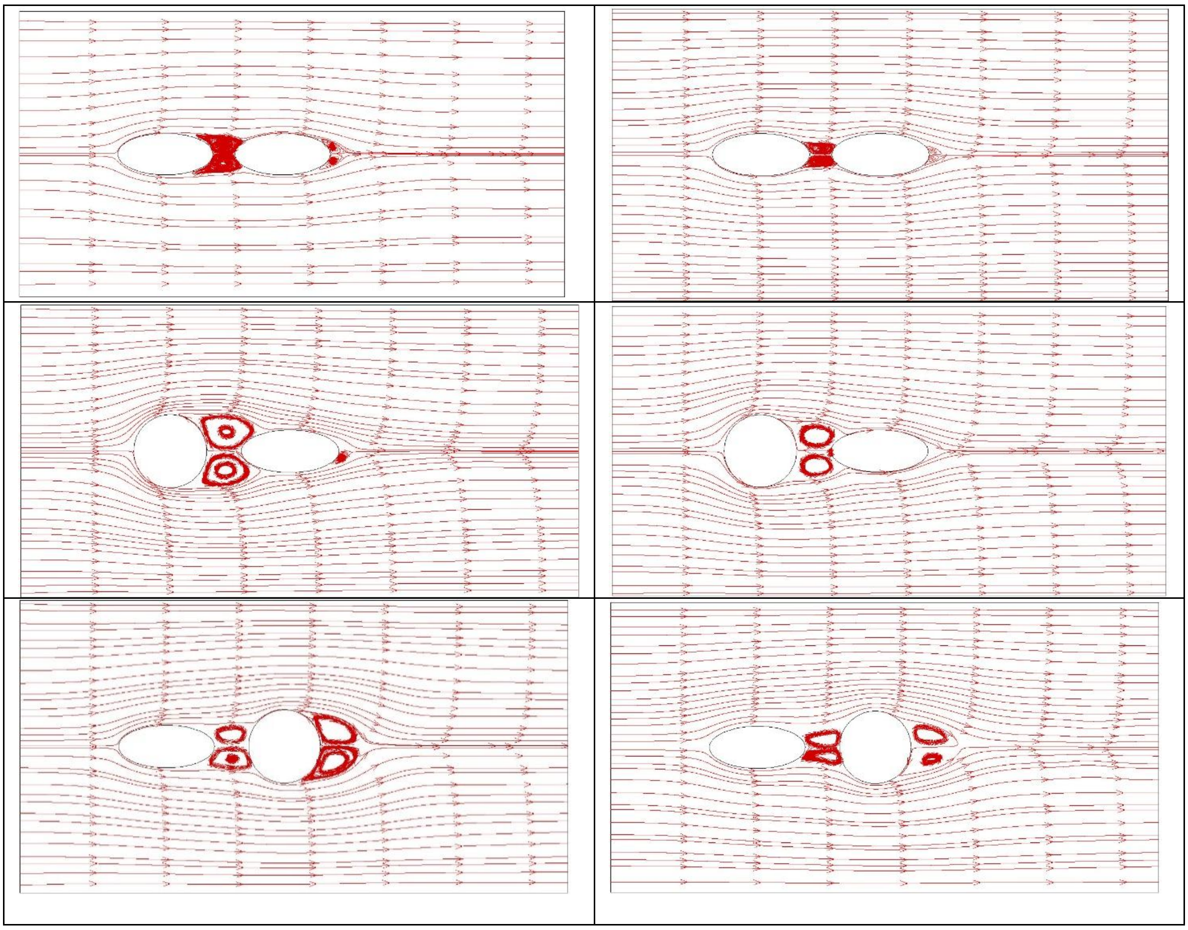
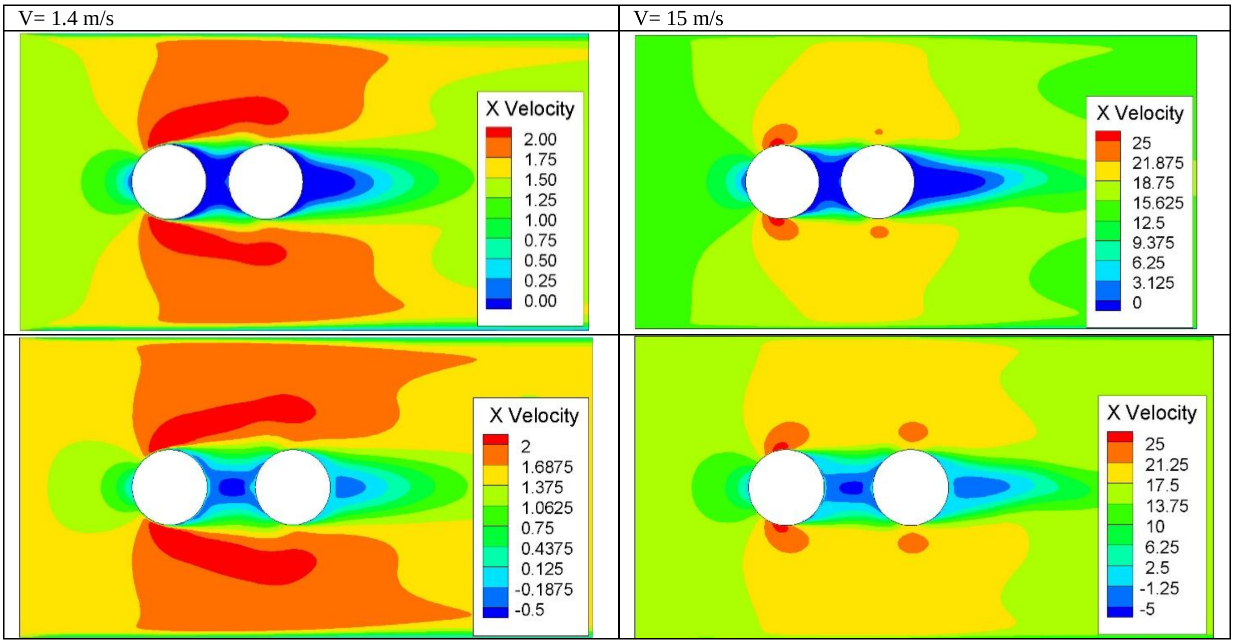


Fig. 6. Streamlines patterns of the tested tubes at different arrangements.



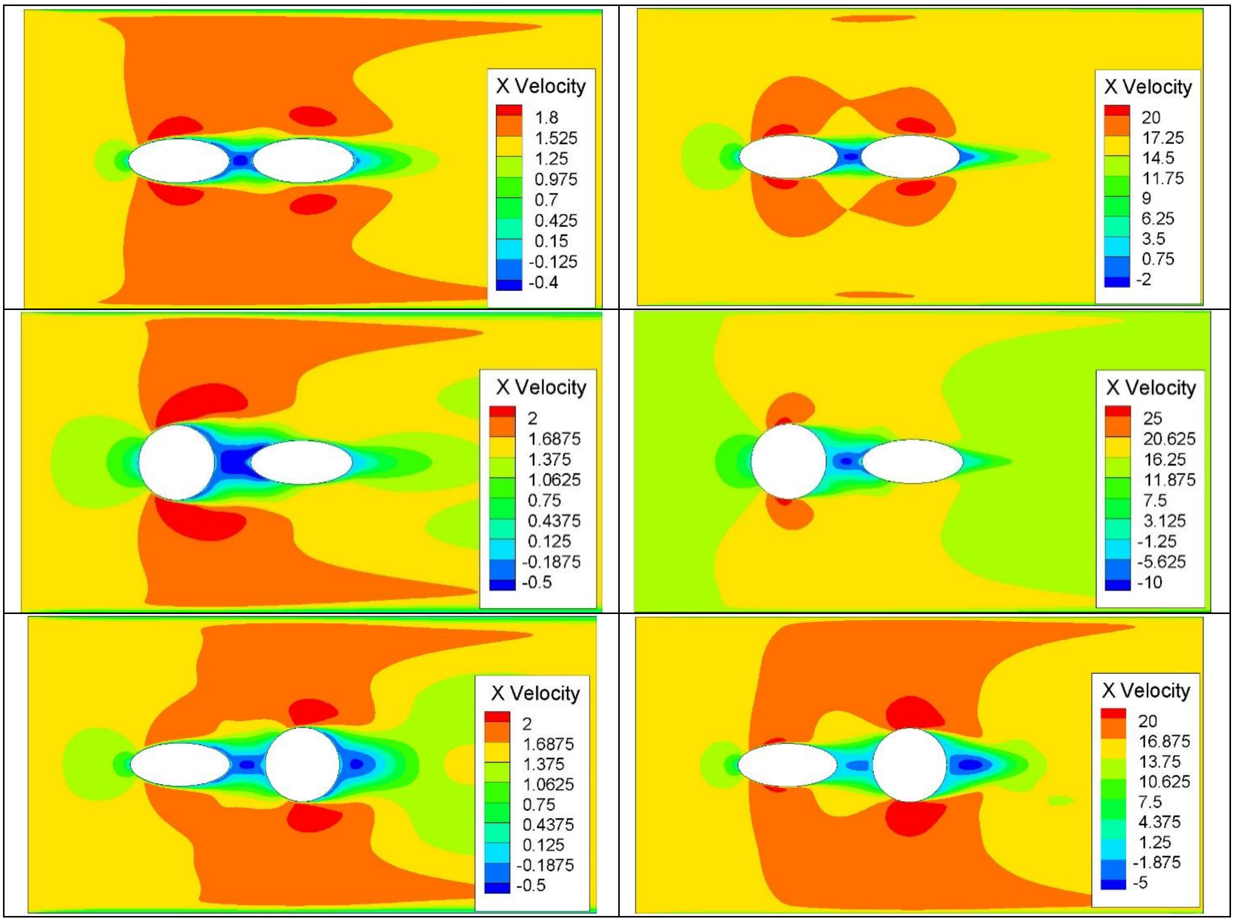
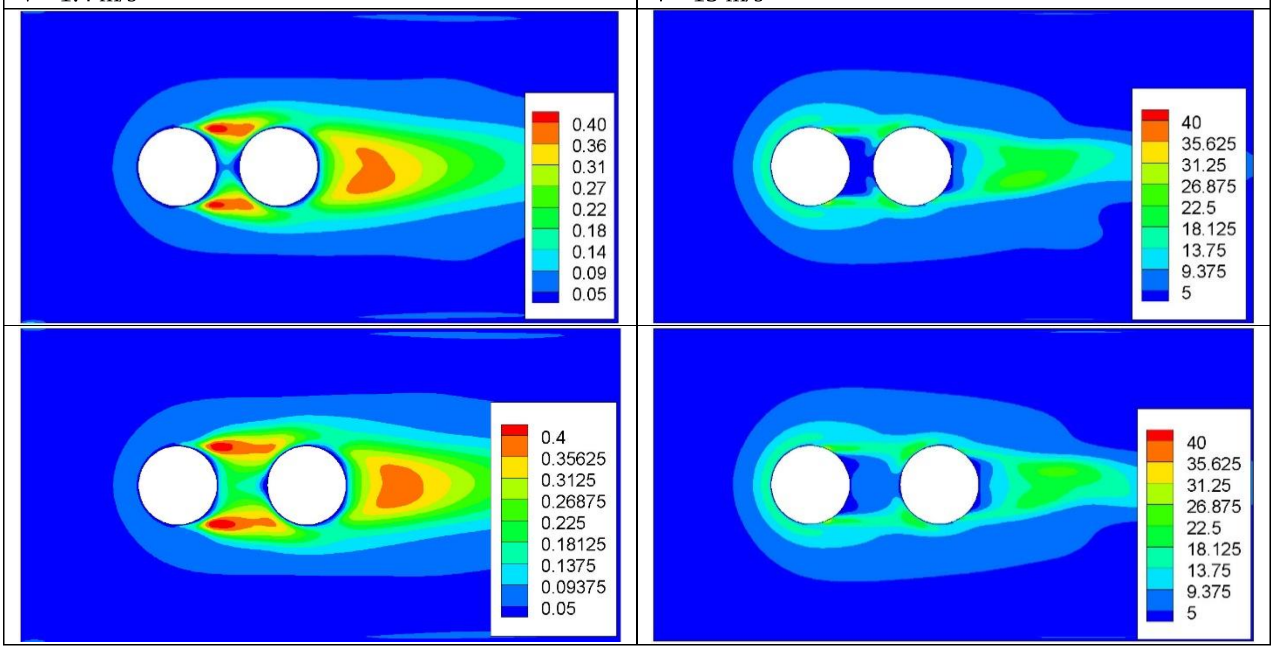


Fig. 7. Velocity contours (m/s) for different arrangements.

V= 1.4 m/s	V= 15 m/s



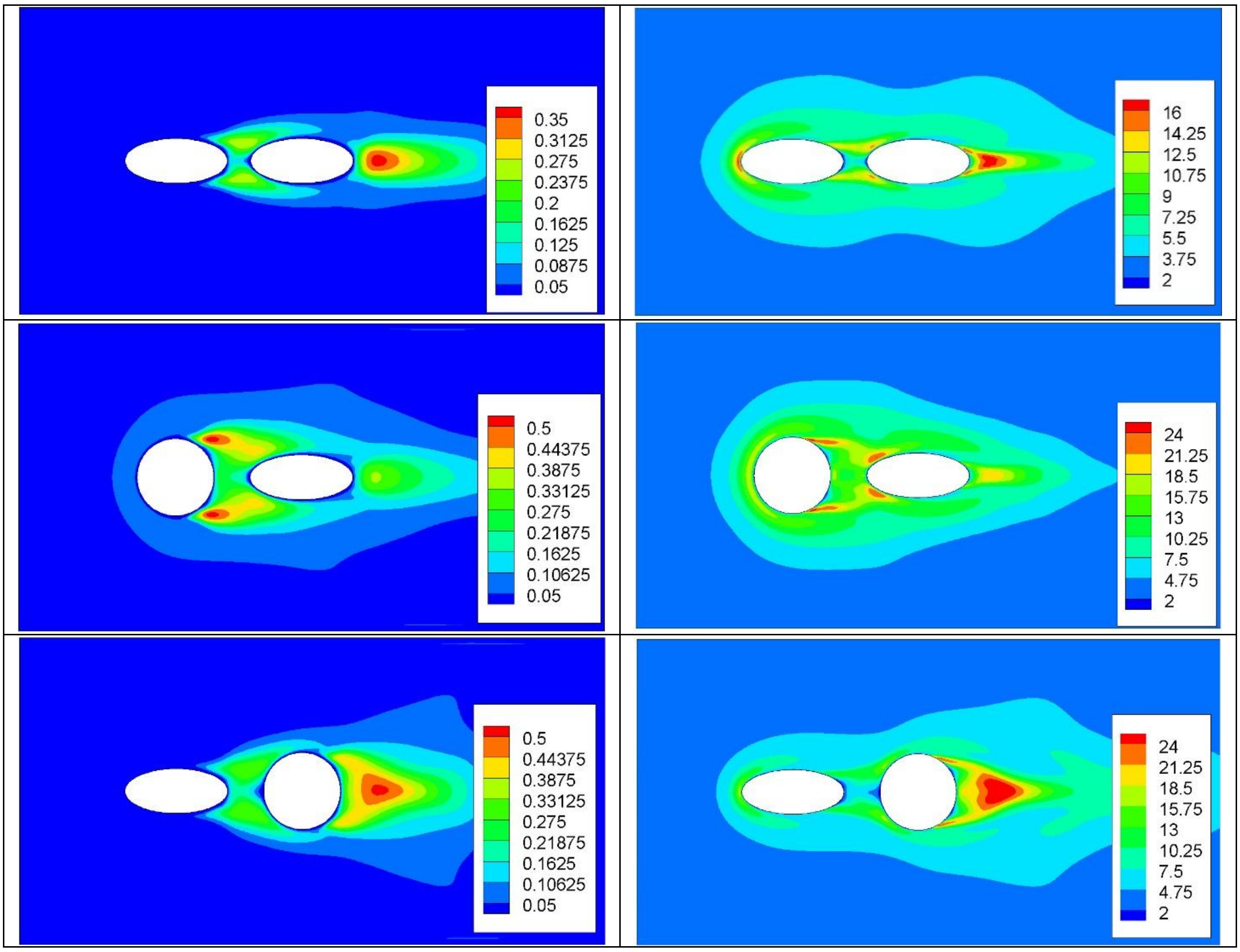
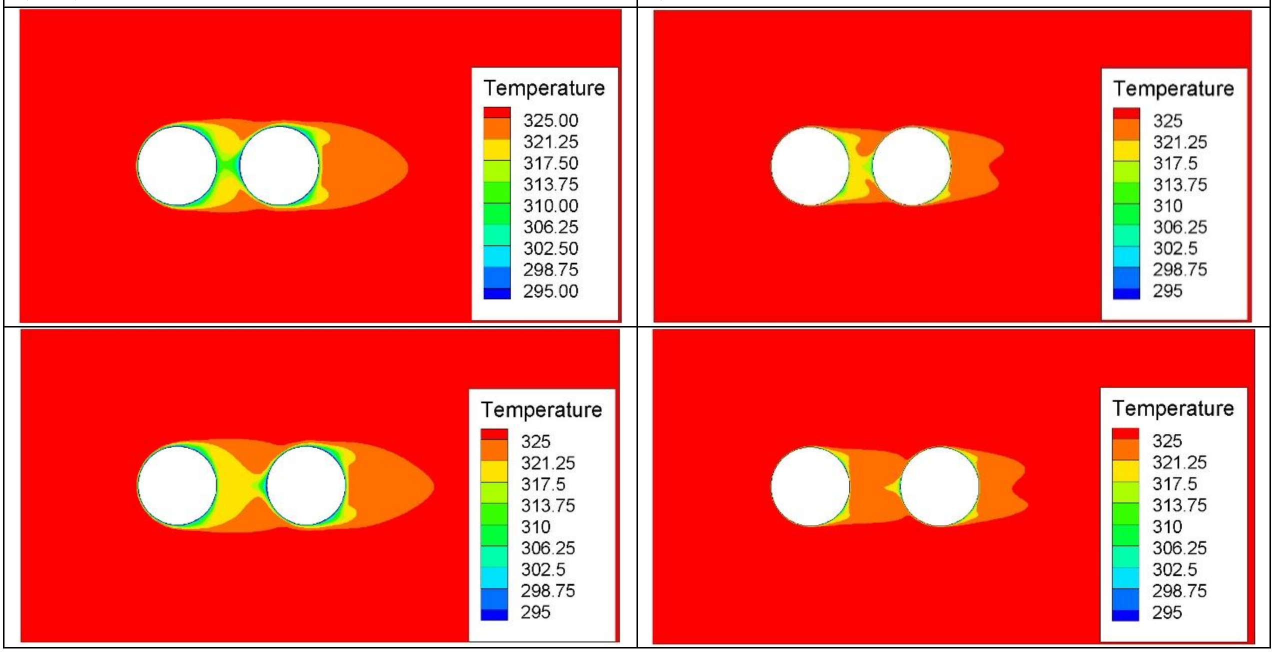


Fig. 8. Turbulent kinetic energy contours for different arrangements.

V= 1.4 m/s | V= 15 m/s



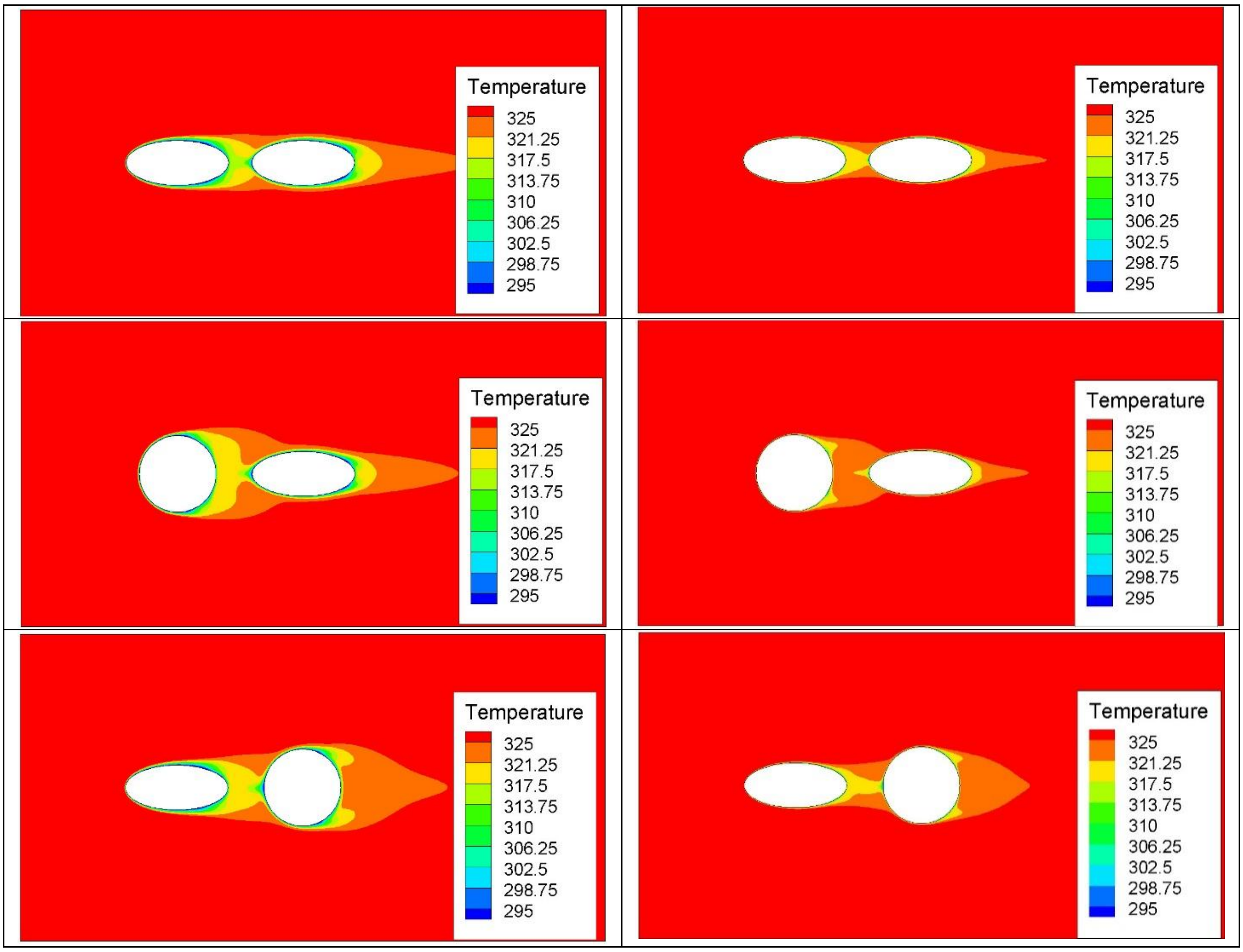
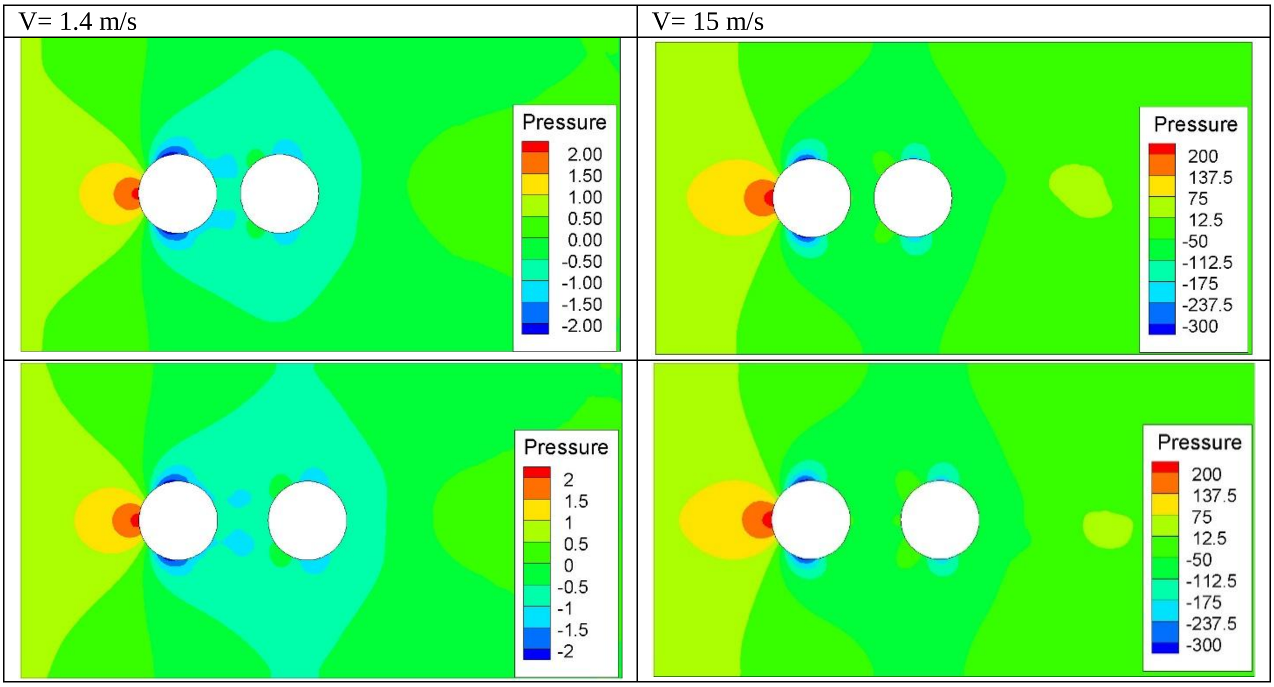


Fig. 9. Temperature (K) contours for different arrangements.



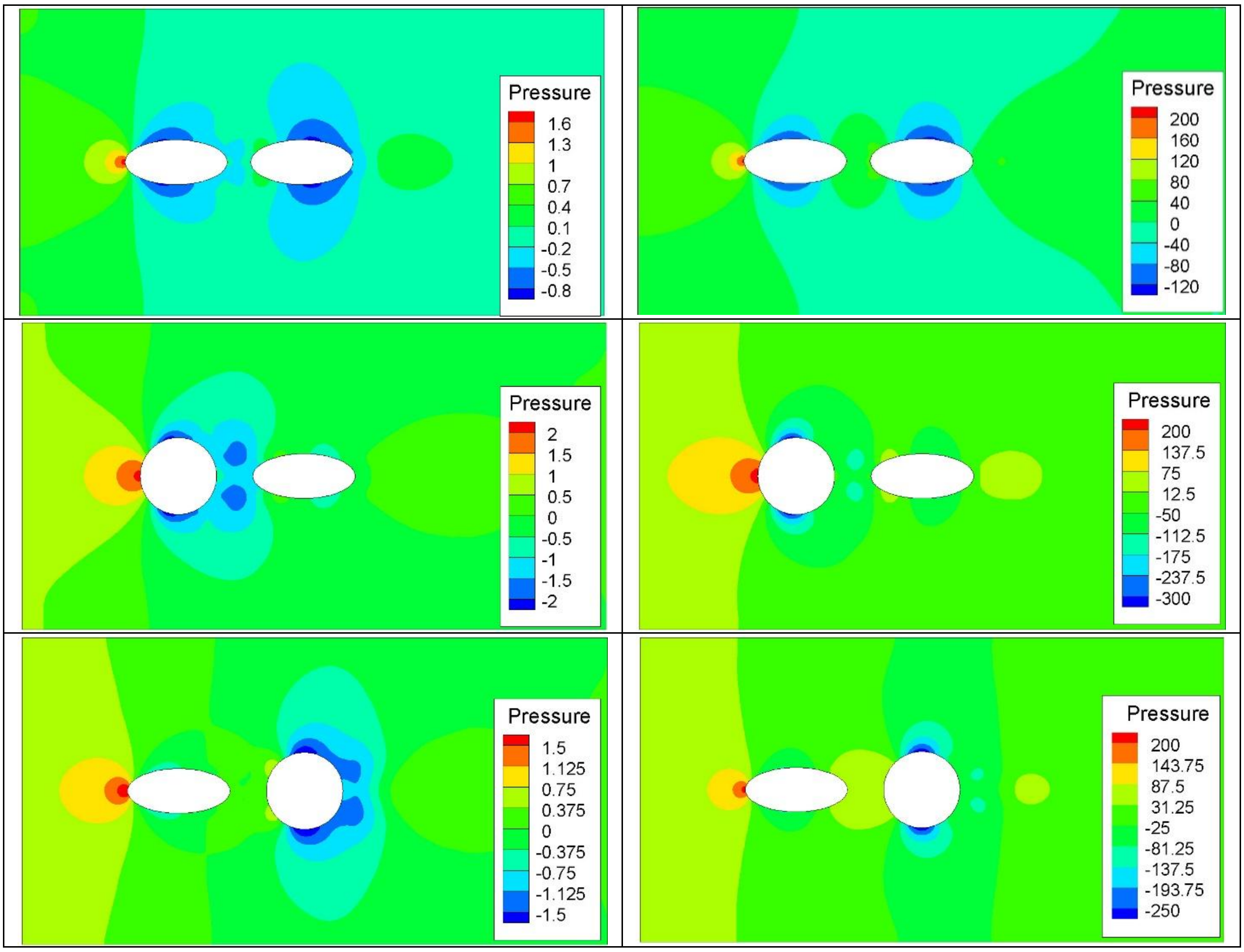


Fig. 10. Static pressure (Pa) contours at different arrangements.

As shown for circular cross section, the heat transfer coefficient is increased as velocity, and gap ratio

In small gap, therefore as the gap increases with increasing velocity the movement of particles increases and so turbulence intensity increases and this enhances the heat transfer coefficient.. On the other hand St_a in Fig. 12 decreases as velocity increases, since the relation between St_a and velocity is an inverse proportional relation and also due to the increasing of kinetic energy as result of increasing velocity. As velocity increases the pressure increases and hence the friction factor decreases for circular cylinder with increasing gap ratio, Fig. 13 and 14.

For elliptical tube the heat transfer and St_a are relatively equal to that for circular one at the same gap ratio, but the overall pressure drop for elliptical cylinder is lower than that of circular one so the power consumption and friction factor for elliptical tube is lower than that of circular one as shown in Fig. 15 to 18.

When changing the position of circular and elliptical one, some changing appears on heat transfer and fluid flow characteristics. As the elliptical cylinder fixed on the second position the heat transfer and St_a increase, on the other hand the pressure drop and hence the friction factor decreased as shown in Fig. 19 to 22.

For all studied arrangements the highest heat transfer is observed for the arrangement of circular-first and elliptical-second cylinder and the minimum pressure drop and hence the friction factor are for the elliptical one as shown in Fig. 23 to 26.

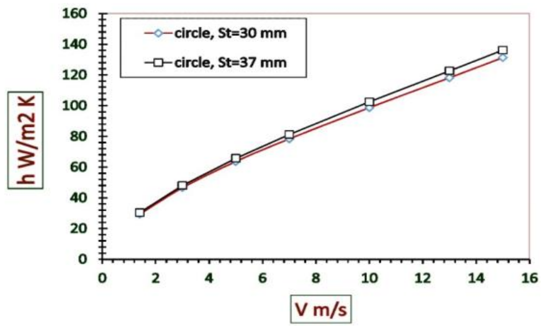


Fig. 11.Effect of longitudinal pitch on h (W/m2K) for circular cross section.

increase Fig. 11, this due to the vortices are not fully developed i

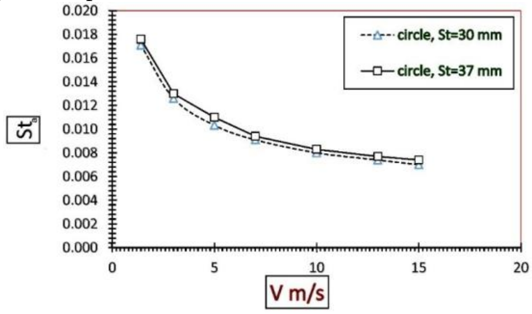


Fig. 12. Effect of longitudinal pitch on St_a for circular cross section.

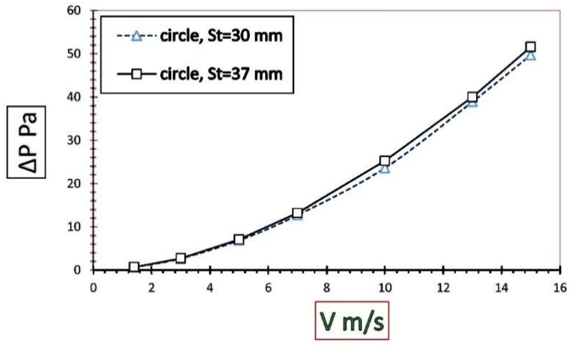


Fig. 13. Effect of longitudinal pitch on ΔP (Pa) for circular cross section.

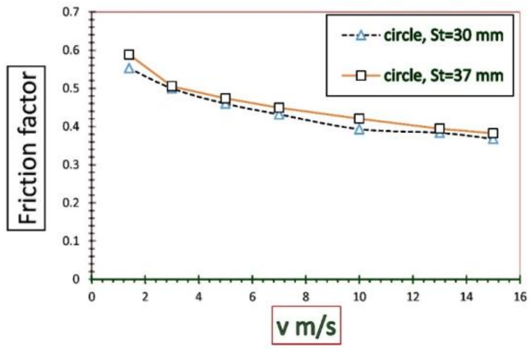


Fig. 14. Effect of longitudinal pitch on friction coefficient for circular cross section.

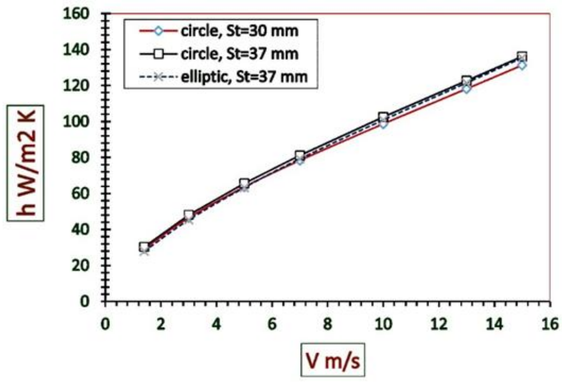


Fig. 15. Effect of tube cross section on h (W/m2K).

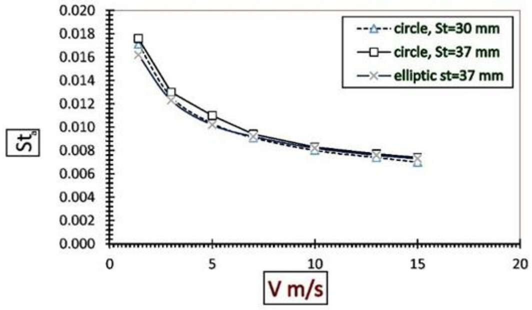


Fig. 16. Effect of tube cross section on St_a.

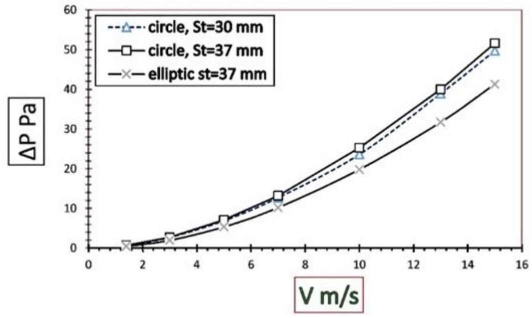


Fig. 17. Effect of tube cross section on ΔP (Pa).

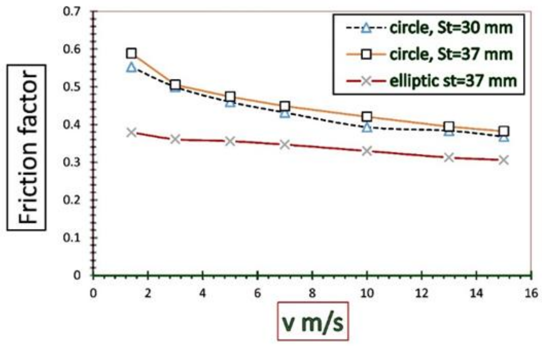


Fig. 18. Effect of tube cross section on friction coefficient.

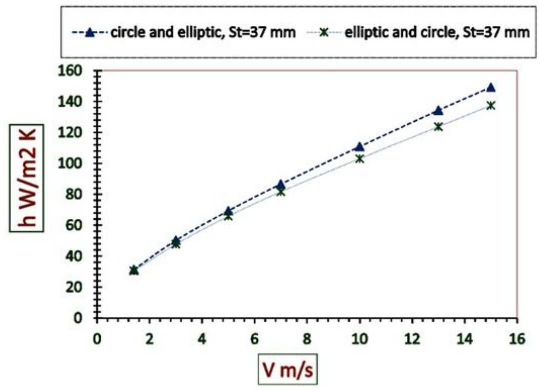


Fig. 19. Effect of tube arrangement on h (W/m2K).

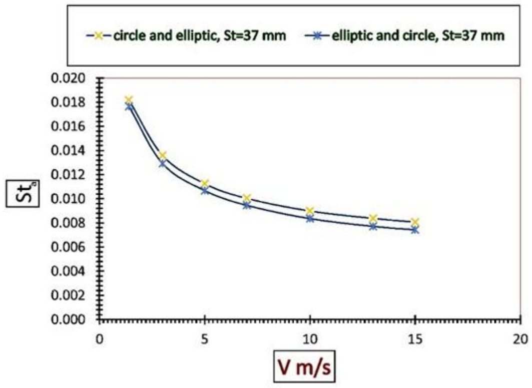


Fig. 20. Effect of tube arrangement on St_a.

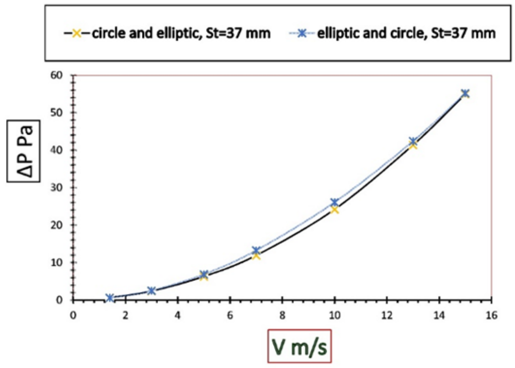


Fig. 21. Effect of tube arrangement on ΔP (Pa).

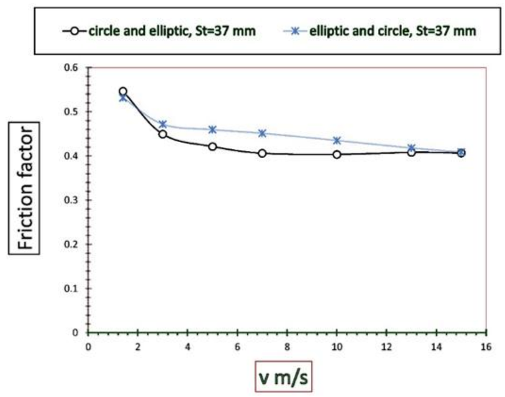


Fig. 22. Effect of tube arrangement on friction coefficient.

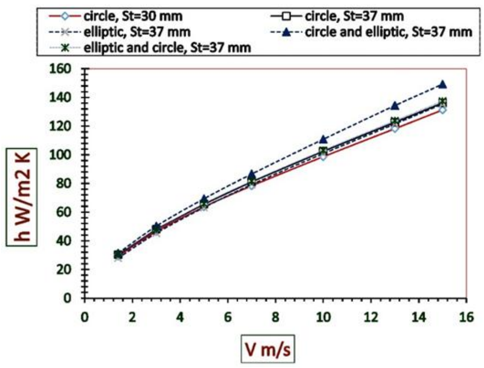


Fig. 23. h (W/m2K) vs. V (m/s) for different studied arrangement.

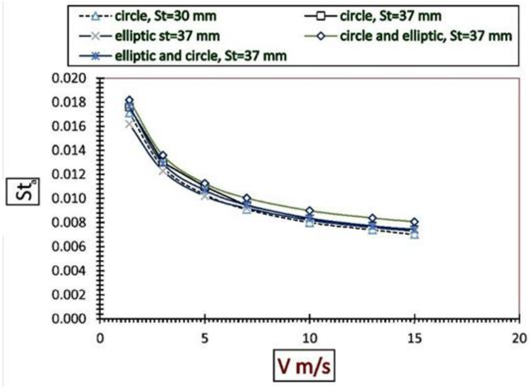


Fig. 24. St_a vs. V (m/s) for different studied arrangement.

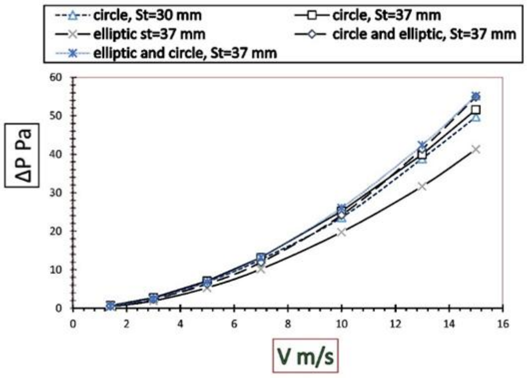


Fig. 25. ΔP (Pa) vs. V (m/s) for different studied arrangement.

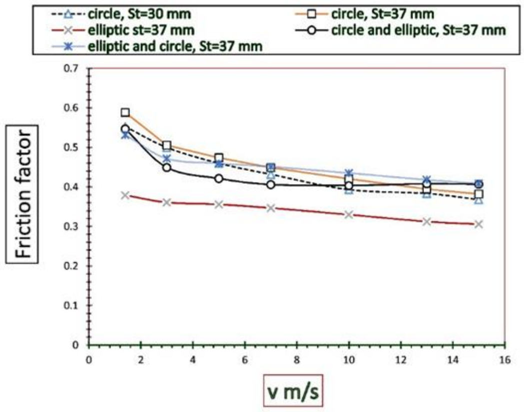


Fig. 26. Friction coefficient vs. V (m/s) for different studied arrangement.

IV. UNCERTAINTY ANALYSIS

4.1 Uncertainty of Numerical results

According to [14], the error in a fine and coarse grids (E_1^{fine}, E_2^{coarse}) and grid convergence index (GCI) are given by: [$E_1^{fine} = (f_2\text{-}f_1)/\ (1\text{-}r^p)$], [$E_2^{coarse} = r^p \times E_1^{fine}$], [$GCI_1^{fine}$ = FS x (abs E_1^{fine})], and [GCI_2^{coarse} = FS x (abs E_2^{fine})].

From the data of both fine and coarse grids (r=2.5 and p = 0.99) and according to the numerical solution, let f_1 and f_2 be the stagnation pressure and [FS = 3]. Table 1 shows the errors and grid convergence index values. In addition to this table, Fig. 27 represents the residuals during solutions

Table 1. Errors and grid convergence index values.

No. of fine grid points	No. of coarse grid points	$P_{fine, Pa}$	$P_{coarse, Pa}$	E_1^{fine}	E_2^{coarse}	GCI_1^{fine}	GCI_2^{coarse}
73078	43332	2.7	2.83	-0.0880044	-0.2200112	0.264013	0.660033

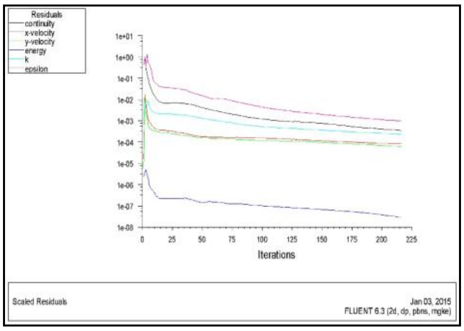


Fig. 27. Residuals of the numerical solutions.

V.CONCLUSIONS

In this paper the heat transfer from two circular and elliptical cylinders in tandem arrangement had been numerically predicted for $2 \times 10^3 \le Re_a \le 21 \times 10^3$. Different tubes arrangements were studied at the considered Re_a range. Also, the commercial CFD software FLUENT was used to get the flow field around the cylinders. To get the effect of different parameters on the heat transfer characteristics, we use the temperature contours. The dependency of the heat transfer coefficient, St_a, pressure drop, and friction factor for circular and elliptical cylinders on the gap ratio is quite clear from the results. For circular cross section, the heat transfer coefficient is increased as velocity, and gap ratio increase. On the other hand St_a decreased as velocity increase. The pressure drop and hence the friction factor increase for circular cylinder as gap ratio increase. For elliptical tube the heat transfer and St_a are relatively equal to that for circular one at the same gap ratio, but the overall power consumption and friction factor for elliptical tube is lower than that of circular one. As the elliptical cylinder fixed on the second position the heat transfer and St_a increase, on the other hand the pressure drop and hence the friction factor decreased. For all studied arrangements the highest heat transfer is observed for the arrangement of circular-first and elliptical-second cylinder and the minimum pressure drop and hence the friction factor are for the elliptical one. According to the uncertainties analysis and the good agreement between the present results and the results of Mohamed [13], so the 2-dimensional analysis of the paper is so good to express any practical application in this field.

AKNOWLEDGMENT

The author would like to thank Eng. Mohamed Attia for his valuable discussions toward the paper material

Nomenclature

f_1: A fine grid numerical solution obtained with grid spacing h_1.

f_2 : A coarse grid numerical solution obtained with grid spacing h_2.

FS : Factor of safety.

r : Refinement factor between the coarse and fine grid.($r = h_2/h_1>1$)

P : Formal order accuracy of the algorithm.(P<1)

p : Pressure, pa. (lb/inch^2)

k: Thermal conductivity, w/m K.

T : Absolute temperature, K.

u, v: Velocity-components in x and y directions, m/s.(inch/s)

ρ : Density of fluid, kg/m^3.(lb/inch^3)

μ : Dynamic viscosity, Pa s.(poise)

REFERENCES

[1]. Naphon p."A Review of Flow and Heat Transfer Characteristics in Curved Tubes". *Renewable and Sustainable Energy Reviews*, Vol.10,(2006),pp.463-490.

[2]. Sayed Ahmed, Sayed E., Emad Z. Ibrahiem, Osama M. Mesalhy and Mohamed A. Abdelatief. " Effect of Attack and Cone Angles on Air Flow Characteristics for Staggered Wing Shaped Tubes Bundle". *Heat and Mass Transfer*, DOI:10.1007/500231,(2015),1473-3.

[3]. Chi-Kyung Kim. " Adaptive Finite Element Solution for The Heat Conduction With a Moving Heat source". *Journal of Mechanical Science and Technology*,Vol. 26-3, (2012),pp.967-972.

[4]. Somayyeh Sadri, Mohamed Reza Raveshi and Shayan Amiri. "Efficiency Analysis of Straight Fin With Variable Heat Transfer Coefficient and Thermal Conductivity." *Journal of Mechanical Science and Technology*, Vol.26-4, (2012),pp.1283-1290.

[5]. Rudra Kanta, Deka and Ashish Paul. "Transient Free Convection Flow Past an Infinite Vertical Cylinder With Thermal Stratification." *Journal of Mechanical Science and Technology,* Vol.26-8, (2012),pp.2229-2237.

[6]. Seong Won Hwang, Dong Hwankim, June Kee Min and JI Hwan Jeong. " CFD Analysis of Fin Tube Heat Exchanger With a Pair of Delta Wing at Vortex Generators.", *Journal of Mechanical Science and Technology*, Vol.26-9, (2012),pp.2949-2958.

[7]. Elsayed, Mohamed L. and Osama Mesalhy. "Effect of a Slotted Shield on Thermal and Hydraulic Performance of a Heat Sink.", *Journal of Electronic Packaging*, Vol.137,(2014), 6 pages.

[8]. Elsayed, Mohamed L. and Osama Mesalhy. "Studying the Performance of Solid/ Perforated Pin- Fin Heat Sinks Using Entropy Generation Minimization ", *Heat and Mass Transfer*,Vol.51, (2015), pp.691–702.

[9]. FLUENT 6.3.26 User's Guide, FLUENT Inc, (2006).

[10]. Zehua H, Sun D," Predicting Local Surface Heat Transfer Coef-Ficients by Different Turbulent k-ε Models to Simulate Heat and Moisture Transfer During Air-Blast Chilling". *Int J Refrig* 24(7), (2000),702–717.

[11]. Yakhot V, Orszag SA, Thangam S, Gatski TB, Speziale GG, " Development of Turbulence Models for Shear Flows by a Double Expansion Technique". *J.Phys Fluids* A 4,(1992),1510–1520,

[12]. Mofreh M. N. "Numerical Solution Of Flow Past Two Circular Cylinders In Different Arrangement Rotating in Various Directions.", *Journal of Mechanical Science and Technology*, (2015).

[13]. Mohamed M. Mosa, "Study of Circular and Elliptical Tube Arrays as Cross Flow Heat Exchangers", *Master thesis, University of Windsor, Canada*, (2009).

[14]. P. J. Roache, "Quantification of Uncertainty in Computational Fluid Dynamics", *Journal of annular review fluid mechanics*, vol. 29, (1997), pp. 123-160.

Mofreh Melad Nassief is currently an associated professor

in Power Mechanical Engineering Department, Zagazig University, Egypt. He received his PhD in 1995. His research interests include, fluid structure interaction, and heat and mass transfer.