Flow and Heat Transfer over two Bluff Bodies from Very Low to High Reynolds Numbers in the Laminar and Turbulent Flow Regimes

A. Etminan
Department of Mechanical Engineering,
Neyriz Branch, Islamic Azad University, Neyriz, Iran
E-mail: etminan.amin@gmail.com

Received: 13 May 2012, Revised: 29 July 2012, Accepted: 25 August 2012

Abstract: Flow structure and heat transfer characteristics around two-equal in-tandem square cylinders in the two dimensional laminar and turbulent flow regimes are simulated numerically for Reynolds and Prandtl numbers, $Re=1\text{-}1\times10^5$, $Pr=0.71$, respectively. The investigation is based on an implicit finite volume scheme for integrating the unsteady Navier-Stokes equations and use of standard κ-ε model to Reynolds stresses and scalar fluxes terms modelling. In this study, the instantaneous and mean streamlines, vorticity and isotherm patterns for different Reynolds numbers and distance between the cylinders are presented and discussed. In addition, the global quantities such as drag coefficients, RMS lift and drag coefficients, Strouhal number and Nusselt number are determined. An interesting phenomenon has been observed in the flow patterns depending upon the Reynolds number and the distance between the cylinders. A switch over in the nature of the fluctuations of the lift and drag coefficients has been also observed with the increase of Reynolds number and the distance between the cylinders. The numerical results are in good agreement with the experimental and numerical data available in the literature.

Keywords: Heat Transfer, Turbulent Flow, κ-ε Model, Vortex Shedding, Square Cylinder

Reference: Etminan, A., "Flow and heat transfer over two bluff bodies from very low to high Reynolds numbers in the laminar and turbulent flow regimes", Int J of Advanced Design and Manufacturing Technology, Vol. 6/ No. 2, 2013, pp. 61-72.

Biographical notes: A. Etminan received his BSc from Persian Gulf University on Fluid Mechanics in 2004. Then he received his MSc at Yazd University on Fluid Mechanics/Energy Conversion in 2007. Since then he has held position in the Department of Mechanical Engineering in the Islamic Azad University as the head of Department at Neyriz Branch for two years. His current research interest includes turbulent flows and cooling of gas turbine blades.

1 INTRODUCTION

One of the most important problems in the fluid dynamics is flow around bluff bodies with different cross sections. The main anxiety about flexible structures exposed to the flow is vibration and fluctuating forces, which are caused by vortex shedding phenomenon. Flow over a bluff body usually creates a large region of separated flow and a massive unsteady wake region downstream and they have susceptibility to flow-induced vibration. Vortex shedding observed in the wake of these bodies generates unsteady-periodic lift and drag forces. These forces are combination of pressure and viscous components. Also, the linear bodies such as airfoils when the angle of attack is large, rolls same manner of bluff bodies.

The interaction of the flow about groups of cylinders is becoming an increasingly popular field of many studies. Flow interference is responsible for several changes in the characteristics of the flow around pairs of cylinders, which can provide a better understanding of the vortex shedding and aerodynamic forces, in cases involving more complex arrangements. Some examples include group of electrical transmission lines, tall buildings and skyscrapers in a city, cooling towers, chimneys, pipelines, cables and bundles of tubes in heat exchangers. These flows usually include some complex events such as; separation flow, wake region, shear flow and eddy shedding.

Square cylindrical geometries often appear in many structures and industrial applications. Although these structures are very simple, the flow pattern around them is not. The square cylinder is a bluff body and can form a large separated region and a massive unsteady wake region. Separated wake and its patterns are nearly impossible to predict analytically and hence must be solved either through experimental or numerical methods.

The fundamental fluid dynamics problems of single circular and square cylinders have been examined extensively in both numerical and experimental studies [1, 2]. However, there are not enough results for the flow over the cylinders in tandem, especially with the square cross section. It seems that there is some similarity between the flow structures of the single and tandem bluff bodies. In tandem cases, there are more flow complexities than the single ones because of the existence of two bodies and their reciprocal effect on the flow pattern (spacing effect).

Wong et al. studied experimentally the aerodynamic forces acting on two square cylinders with different size in side-by-side arrangement and vortex shedding phenomenon [3]. Their study was in the range of Reynolds numbers, $\mathrm{Re} = 2.5\times10^4 - 5\times10^4$. Their results show that maximum pressure coefficient acts on small cylinder for distance between the cylinders equal by 1.75 times of the cylinders width. Also flow separation point was located on inner surface of larger cylinder and distance between the cylinders equal by 1.25 times of the cylinders width. However for smaller cylinders, this may occur for larger distance between the cylinders. By increasing the distance between the cylinders, effect of upstream cylinder on flow patterns around the downstream cylinder and its vortex shedding have been reduced. The skew of von-Kármán street of small cylinder by increasing of distance between the cylinders coming down too.

The flow around two bars with square cross section in tandem and side-by-side arrangements in channel have been simulated numerically by Alvarez et al. [4]. They studied the effect of bars on pressure loss in channel and heat transfer of channel walls. Their results showed that local and overall Nusselt numbers on channel walls increased highly due to unstable vortex shedding caused by the bars. Also, local heat transfer increased after downstream bar in tandem arrangement. Local friction coefficient has two pick points in tandem arrangement; first at upstream bar position and another one at downstream bar position. The local Nusselt number has same pick points too.

Niceno et al. studied passing flow and heat transfer of wall mounted cubes in channel by using large eddy simulation method [5]. Their investigation for Re=13000 illustrated a non-uniform distribution temperature around the cubes as a result of the complex and rotational flow structure vicinity of them. Moreover, they analyzed flow structure and eddy shedding around cubes to find correlation between local heat transfer and temperature distribution on them.

The unsteady turbulent flow characteristics and heat transfer about side-by-side square have been simulated numerically by Valencia and Cid [6]. In that research, the Reynolds number based on channel height was Re=20000, where channel walls were at constant temperature. They studied the effect of distance between the bars on the flow and heat transfer patterns. Their results showed that the unstable behavior of the flow, pressure loss and heat transfer strongly depended on the distance between the bars; the increase of the mean heat transfer coefficient was considerably smaller than the pressure loss growth.

Lin et al. investigated experimentally the flow around two equal-sized circular cylinders in tandem arrangement for Reynolds number, Re=10000 [7]. They showed that the flow pattern is symmetric in the space between the cylinders when that distance was equal with cylinder diameter. By increasing the distance

between the cylinders vortex shedding occurred of upstream cylinder and the Reynolds stresses amplified.
In another experimental work, Chia hung and Chen [8] studied characteristics of flow around two square cylinders in tandem arrangement. They considered distance between the cylinders from 1.5 to 9 times of cylinder width and Reynolds number, Re=2000-16000. In that research the effect of Reynolds number and distance between the cylinders variations on the drag force, mean and fluctuating pressure coefficients and Strouhal number are investigated. Their discussion showed that the flow characteristics around two square cylinders are strongly depended upon the ratio of distance between the cylinders to the cylinder width and how to change the distance between the cylinders. In addition, that research showed that increasing the distance between the cylinders causing to change the symmetric to asymmetric flow pattern and onset of vortex shedding from upstream cylinder. These phenomena changed highly drag coefficients of cylinders, specially for downstream cylinder.
Mahbub et al. investigated experimentally the acting forces on in tandem square cylinders in Reynolds number, 56000 [9]. They studied effect of a T-shape controller in upstream cylinder on magnitude of acting forces on cylinders. Their results showed that for position of controller in range of 1.5 to 1.9 times of cylinders width had most effect on decreasing drag forces. In this range of controller position, drag force reduced 84% and 66% for upstream and downstream cylinder respectively. The magnitude of drag force of upstream cylinder was dependent upon the position of downstream cylinder. It should be noted that the presented results by references [8, 9] shows the drag coefficient of cylinders tends to be constant at large enough distances between the cylinders.
In other experimental research, Mahbub et al. have investigated the acting aerodynamic forces on two circular cylinders in side-by-side arrangement for Reynolds number, 56000 [10]. Their results showed that for the distance between the cylinders less than 1.2 times of cylinders diameter, flow pattern comes to one side, wake region behind the upstream cylinders narrows and downstream cylinder wake propagates. Also, they found that for the distance between the cylinders larger than 0.2 times of cylinders width, lift drag of cylinders are in opposite directions.
Mahbub et al. studied experimentally the acting fluctuating forces on two in tandem circular cylinders for Reynolds number, 65000 [11-14]. Their results showed that critical distance between the cylinder equal with three times larger than cylinder diameter where drag force coefficients of cylinders changed strongly. In addition, they studied the effect of changing the distance between the cylinders and the angle between them on flow patterns around the cylinders and acting forces on them. In [14] they studied experimentally the effect of T-shape plate on acting forces on two in tandem circular cylinders. This controller obviously reduced drag coefficient of upstream cylinder, whilst controller has no noticeable effect on drag coefficient of downstream cylinder. Their results showed that, for distance between the cylinders larger than three times of cylinder diameter, upstream cylinder drag coefficient was equal with single circular cylinder. In other words, influence of the downstream cylinder was negligible on upstream one. Also, the provided results in [14] indicated that increasing the distance between the cylinders except in the range of 1.5 to 3 times the cylinders diameter without the T-shape plate, reduced upstream cylinder drag coefficient. Etminan simulated numerically the flow over single and two in tandem bluff bodies with square cross section extensively for different distances between them [15].
By considering reviewed literatures, there are some investigations for the flow over the circular and square cylinders in the tandem arrangement, where often concentrated on flow and heat transfer in channel. There are no numerical or experimental results in the range of the Reynolds numbers for this study in the published literatures to compare with the present work results. The only comparable experimental results with the results of this study are presented in [8, 9]. It should be noted, that the presented results in these references are only related to the flow patterns and no temperature pattern and Nusselt number exist.
The objective of the present work is to study free-stream flow over two square cylinders and explore the link between fluctuating forces, Nusselt number and the two dimensional laminar and turbulent flow regimes for various Reynolds number and distance between the cylinders. Although the flow regimes have been studied previously, the physics of flow patterns around two square cylinders, which is the focus of the present work, is largely unexplored.
Another objective of this work is to generate a numerical database for the flow parameters with respect to the Reynolds number for the considered boundary conditions. The considered Reynolds and Prandtl numbers are $Re=1\text{-}1\times10^5$, $Pr=0.71$ respectively. The distance between the cylinders ranges from 1 to 4 by step one. The investigation is based on an implicit finite volume scheme for integrating the unsteady Navier-Stokes equations and use of standard κ-ε model to Reynolds-stresses and scalar fluxes terms simulations.
In this study, the instantaneous and mean streamlines, vorticity and isotherm patterns for different Reynolds numbers and distance between the cylinders are presented. In addition, the global quantities such as drag coefficients, RMS lift and drag coefficients,

Strouhal and Nusselt numbers are determined. An interesting phenomenon has been observed in the flow patterns depending upon the Reynolds number and the distance between the cylinders.

The problem under consideration is depicted in Figure 1 where, two equal-sized tandem square cylinders with sides (*d*) are exposed at zero angle of attack to a constant free stream with uniform velocity represented by u_{in}. Incompressible viscous flow with the constant fluid properties is assumed. All dimensions are scaled with side, *d*. The vertical distance between the upper and lower boundaries, *H*, defines the blockage of the confined flow. It is expected that if the width of the computational domain is chosen adequately large, the lower blockage parameter, the flow in the lower and the upper boundaries goes to the free-stream conditions. In other words, these boundaries should be sufficiently far away from the cylinders to satisfy this boundary condition. The effect of blockage ratio was studied for a single square cylinder by Etminan [15] and Sohankar et al. [16, 17]. They have shown that the free-stream condition is satisfied and that the boundaries have little effect on the characteristics of flow patterns if $B = 5\%$. The effect of blockage ratio ranges from 4 to 10% on the flow structure and parameters studied by Etminan [15, 18-19] and Sohankar and Etminan [17] in detail. They found that blockage ratio from 5 to 4% has negligible changes on the fluid flow and heat transfer results. Thus, it is expected that blockage ratio, 5% is a suitable blockage ratio for further simulations of the present work.

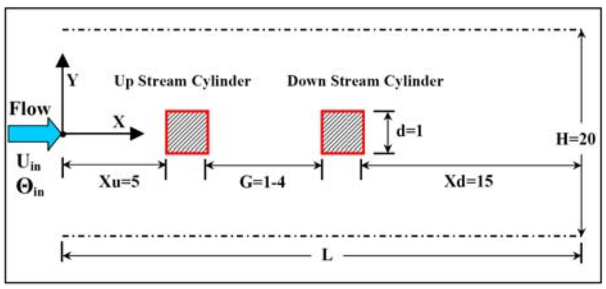

Fig. 1 Computational domain for flow around the square cylinders

2 PROBLEM DESCRIPTION AND GOVERNING EQUATIONS

The governing equations for analysis of heat transfer assessment are Navier-Stokes and energy respectively, which have been averaged against time by Reynolds method. With assumption of existent relationship between Reynolds stresses and mean velocity gradient, the Boussinesq hypothesis has been utilized to calculate the Reynolds stresses to lead the results for finding the turbulent viscosity. In turbulence models that employ the Boussinesq approach, the central issue is how the eddy viscosity is computed. The unsteady Navier-Stoks equations in dimensionless form for incompressible flow by constant properties are as follows:

$$\frac{\partial U_i}{\partial X_i} = 0 \tag{1}$$

$$\frac{\partial U_i}{\partial \tau} + \frac{\partial (U_i U_j)}{\partial X_j} = -\frac{\partial P}{\partial X_i} + \frac{\partial}{\partial X_j}\left[\frac{1}{\mathrm{Re}}\left(\frac{\partial U_i}{\partial X_j} + \frac{\partial U_j}{\partial X_i}\right)\right] - \frac{\partial}{\partial X_j}\left(\overline{u_i' u_j'}\right) \tag{2}$$

$$\frac{\partial \Theta}{\partial \tau} + \frac{\partial (U_j \Theta)}{\partial X_j} = \frac{\partial}{\partial X_j}\left[\frac{1}{\mathrm{Re}\,\mathrm{Pr}}\frac{\partial \Theta}{\partial X_j}\right] - \frac{\partial}{\partial X_j}\left(\overline{u_j' \theta'}\right) \tag{3}$$

Above equations contain unknown Reynolds stresses and scalar fluxes that present $\overline{u_i' u_j'}$ and $\overline{u_j' \theta'}$ respectively. These turbulence diffusion fluxes have great effects on flow patterns and turbulence. To close these equations, the unknown terms must be modeled. By applying standard κ-ε turbulence model and the definition of dimensionless parameters, turbulent kinetic energy and dissipation equations in dimensionless form will come as follows:

$$\frac{\partial K}{\partial \tau} + \frac{\partial (U_j K)}{\partial X_j} = \frac{\partial}{\partial X_j}\left[\left(\frac{1}{\mathrm{Re}} + \frac{1}{\sigma_K \mathrm{Re}_t}\right)\frac{\partial K}{\partial X_j}\right] + \frac{1}{\mathrm{Re}_t}\left(\frac{\partial U_i}{\partial X_j} + \frac{\partial U_j}{\partial X_i}\right)\frac{\partial U_i}{\partial X_j} - \Sigma \tag{4}$$

$$\frac{\partial \Sigma}{\partial \tau} + \frac{\partial (U_j \Sigma)}{\partial X_j} = \frac{\partial}{\partial X_j}\left[\left(\frac{1}{\mathrm{Re}} + \frac{1}{\sigma_\Sigma \mathrm{Re}_t}\right)\frac{\partial \Sigma}{\partial X_j}\right] + \frac{\Sigma}{\mathrm{K}}\left[\frac{C_{\varepsilon 1}}{\mathrm{Re}_t}\left(\frac{\partial U_i}{\partial X_j} + \frac{\partial U_j}{\partial X_i}\right)\frac{\partial U_i}{\partial X_j} - C_{\varepsilon 2}\Sigma\right] \tag{5}$$

The dimensionless parameters are defined as follows:

$$U_i = \frac{u_i}{u_{in}} \quad i,j=1,2 \quad , P = \frac{p}{\rho u_{in}^2} \ , \ \Theta = \frac{T - T_{in}}{T_{Wall} - T_{in}}$$

$$\theta' = \frac{t' - T_{in}}{T_{Wall} - T_{in}} \ , \ \mathrm{K} = \frac{k}{u_{in}^2} \ , \ \Sigma = \frac{\varepsilon d}{u_{in}^3} \ , \ \tau = \frac{t\, u_{in}}{d}$$

where, $C_{\varepsilon 1} = 1.44$, $C_{\varepsilon 2} = 1.92$, $\sigma_{\mathrm{K}} = 1.0$, $\sigma_\Sigma = 1.31$.

In addition, all of the above equations are presented with regard to constant density and viscosity and also negligible viscose dissipation. The Reynolds, turbulence Reynolds, Strouhal and Prandtl numbers are defined as follows:

$$\begin{aligned} &\mathrm{Re} = u_{in} d / \nu \\ &\mathrm{Re}_t = u_{in} d / \nu_t \\ &St = fd / u_{in} \\ &\mathrm{Pr} = \upsilon / \alpha \end{aligned} \tag{6}$$

Where, u_{in}, d, v, v_t, α and f correspond to the inlet flow velocity, width of cylinders, kinematic viscosity, kinematic turbulence viscosity, thermal diffusion coefficient and the frequency of vortex shedding, respectively. Local Nusselt number is calculated as follows:

$$Nu = \frac{hd}{k} = -\frac{\partial \Theta}{\partial n}\bigg|_{Wall} \tag{7}$$

Where, h and k are convective and conductivity heat transfer coefficients respectively. Also, n is the normal direction on the cylinders walls. An incompressible SIMPLEC finite volume code is used employing collocated grid arrangement. The scheme is implicit in time, and a second order Crank-Nikolson scheme, has been used. The convective and diffusive terms are discretized using QUICK and central differencing schemes, respectively. The time-marching calculations are commenced with the fluid at rest and a constant time step $\Delta\tau = 0.025$ is used for all simulations. This value is chosen based on the presented numerical results in Etminan [15] and Sohankar et al. [16].

The employed boundary conditions are as follows:

- Top and bottom boundaries: $\partial U / \partial Y = 0$, $V = 0$
- No-slip condition on the cylinders walls: $U_{i,wall} = 0, \Theta_{wall} = 1$
-Uniform velocity at the inlet: $U_{in} = 1, \mathrm{V}_{in} = \Theta_{in} = 0$
- Outlet boundary condition: $\partial U_i / \partial \mathrm{X} = 0$, $\partial U_i / \partial \mathrm{X} = 0$

Owing to the existence of the cylinders and their effect on the flow, a non-uniform grid is made around the cylinders using a hyperbolic tangent stretching function. This type of stretching functions and their advantages in non-uniform grid distribution have been discussed by Thompson et al. [20]. Beyond this non-uniform region (one d from cylinders surfaces), a uniform grid is established with the same size as the latest generated cell in the places with non-uniform grid distribution. The domain is divided into five and three separate zones in X and Y directions respectively included both uniform and non-uniform grid zones. The grid distribution was made uniform with a constant cell size, 0.18, outside a region around the cylinder that extended upstream, downstream, and sideways. A grid of much smaller size, d, is clustered around the cylinder, with the smallest cell size, 0.0053, over a distance of one unit to adequately capture wake-wall interactions.

3 RESULTS AND DISCUSSION

The presented results by [17] shows that increasing Reynolds number transfers flow separation point from trailing edge to leading edge of upstream cylinder. Meanwhile the position of flow separation for downstream cylinder is always located on its trailing edge. In this research it has been found that for distance between the cylinders, $G \leq 1$ flow patterns stayed symmetric and vortex shedding occurred only from downstream cylinder slowly. At the larger distance between the cylinders vortex shedding phenomenon occurred from both cylinders in the same frequency.

Figure 2 shows instantaneous and time-averaged contours for some flow parameters around the cylinders for Reynolds number Re=2700 and G=1. It is seen that flow patterns in the space between the cylinders is symmetric and vortex shedding occurs just from downstream cylinder. Due to the symmetric region and boundary layer effect, X-velocity component magnitude around and behind of the upstream cylinder is obviously low, compared with other region of domain.

Due to the formation of stagnation zone in the vicinity of the front side of upstream cylinder, pressure increased strongly and large drag force acted on upstream cylinder compared with downstream cylinder one. Also, it is found that due to the boundary layer effects and flow separation the magnitude of Z-vorticity values around the edges of the cylinders will have a noticeable increase for both clockwise and counterclockwise components.

As was mentioned, for the small distances between the cylinders, flow pattern between cylinders was symmetric and vortex shedding occurred just from downstream cylinder. To further understanding this phenomenon, instantaneous streamlines around the cylinders have been shown in Figure 3 for different Reynolds number and distances between the cylinders. It can be seen that flow separation points have been located on leading edges. In addition, symmetrical flow between the cylinders created just for Reynolds number, 2700 and G=1. By increasing, the distance between the cylinders, flow patterns and global quantities suddenly deformed, which will be investigated next.

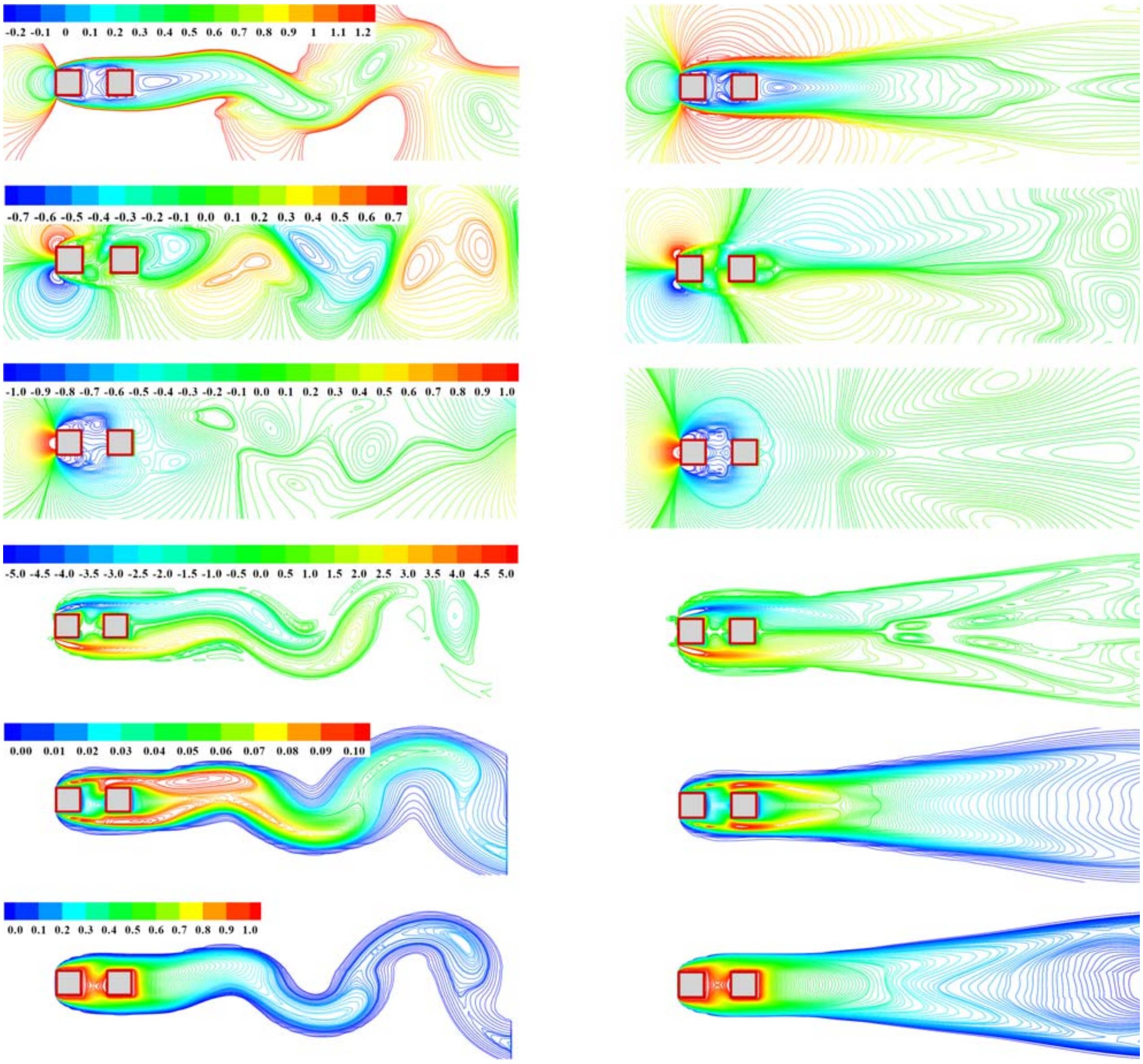

Fig. 2 Instantaneous (left side graphs) and time-averaged (right side graphs) contours from top to down; X-velocity component; Y-velocity component; pressure; Z-vorticity component; turbulent kinetic energy and temperature, respectively around the cylinders for Reynolds number, Re=2700 and distance between the cylinders, G=1

For higher Reynolds number symmetrical flows did not appear. It is found that the flow patterns for different Reynolds number have remained identical and obviously no change was observed. In other words, the flow structure and global quantities are independent of the Reynolds number.

The existence of cylinders caused fluctuations in downstream flow. Also, the level of fluctuations around the downstream cylinder has been increased due to the upstream cylinder interaction to the incoming flow. Therefore, the level of turbulent kinetic energy has growth vicinity of downstream cylinder.

Figure 4 shows the instantaneous streamlines vicinity of cylinders for different Reynolds numbers and the distances between the cylinders.

It should be noted that the flow structure and its pattern and also global quantities such as turbulence kinetic energy have significant variations in the range of Reynolds number, $\mathrm{Re}=2700-16000$. But for higher Reynolds number, $\mathrm{Re}\geq 16000$ a major change in results is not observed and this confirms earlier results that various parameters at high Reynolds numbers fluid flow field and temperature is almost independent of Reynolds number.

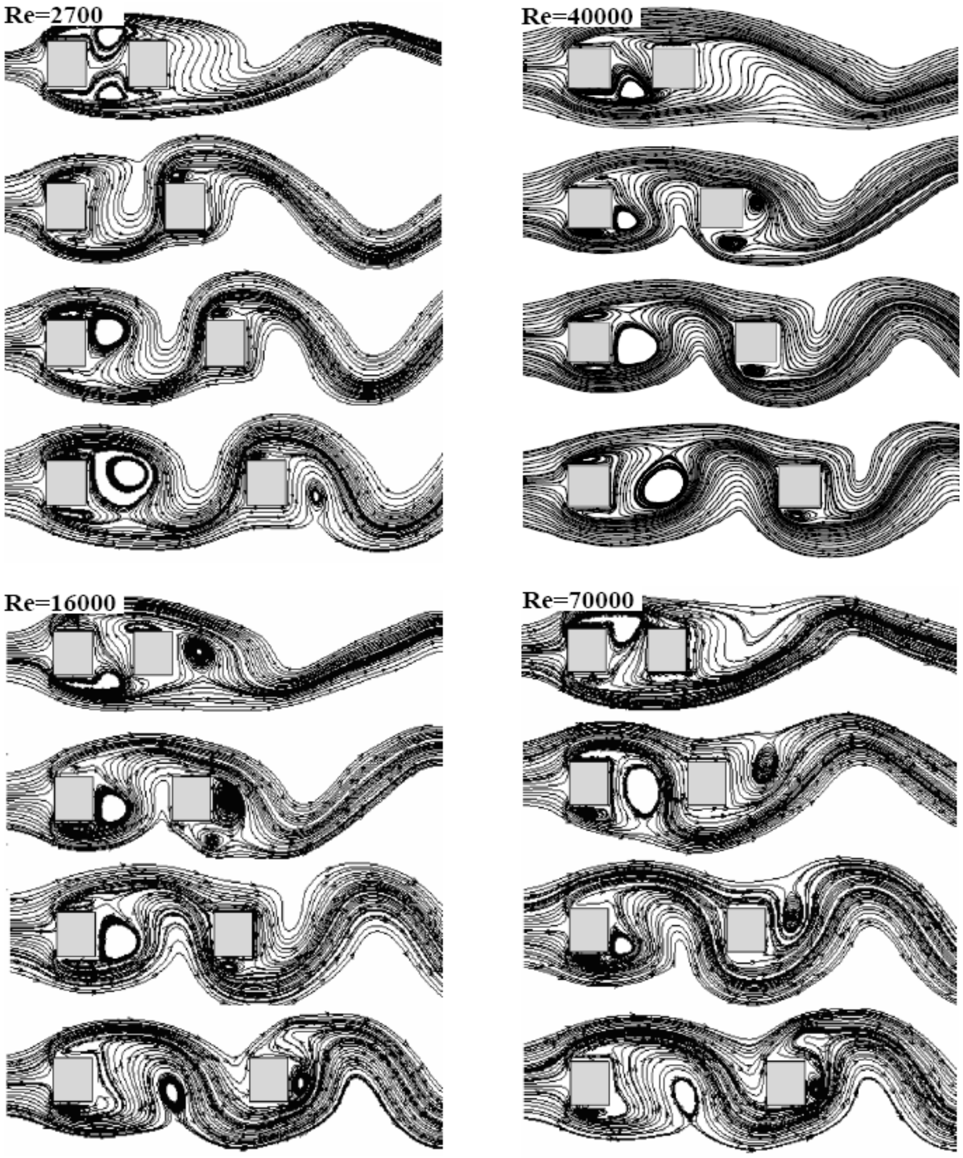

Fig. 3 Instantaneous streamlines vicinity of cylinders for different Reynolds numbers and the distances between the cylinders

From Figure 4, it is found that the turbulence intensity level has been increased due to the sharp edges of cylinders, where turbulence kinetic energy and its dissipation were in highest value range.

As well as, in the range of Reynolds number, $\mathrm{Re} = 2700 - 16000$, the flow fluctuations and turbulence kinetic energy magnitude have been increased; the stream wise length of downstream cylinder's wake and turbulence influence have been decreased too. The variation of turbulence kinetic energy and its dissipation have same manner. The time-averaged streamline colored by velocity and pressure has been shown in Figure 5.

It should be noted that the backward flow formed due to the wake region behind the cylinders where, X-velocity component is negative and vorticity direction can be seen clearly. Also, the stagnation zone is formed near the front side of the upstream cylinder where, the pressure increased highly. The maximum X-velocity component is near top and bottom sides of cylinders and in the range of Reynolds number, $\mathrm{Re} = 2700 - 16000$ in the space between the cylinders the pressure level decreased and X-velocity increased.

As well as, for higher Reynolds numbers, $\mathrm{Re} \geq 16000$ a major change in results is not observed and this confirms earlier results that various parameters at high Reynolds numbers, fluid flow field and temperature patterns are almost independent of Reynolds number. The same manner can be seen for other distance between the cylinders studied in the present work.

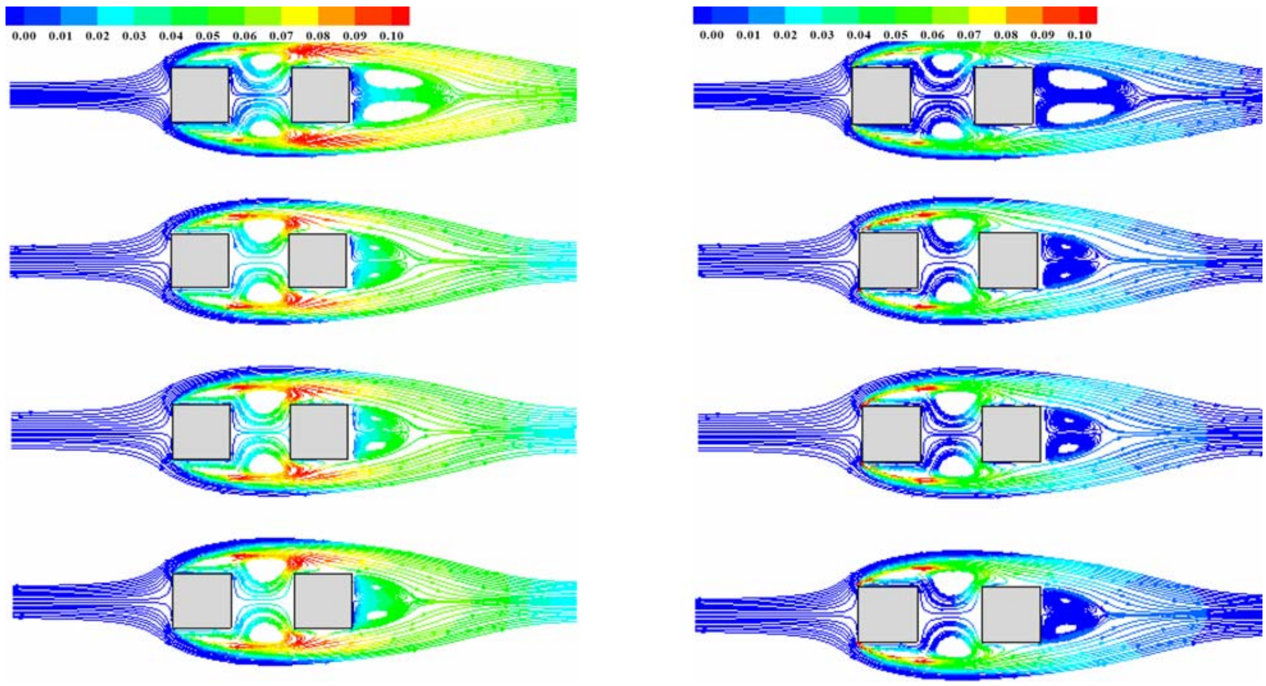

Fig. 4 The time-averaged streamline colored by turbulence kinetic energy (left side graphs) and turbulence kinetic energy dissipation (right side graph) for Reynolds number Re=2700, 16000, 40000, 70000 from top to down, respectively and the distance between the cylinders, G=1

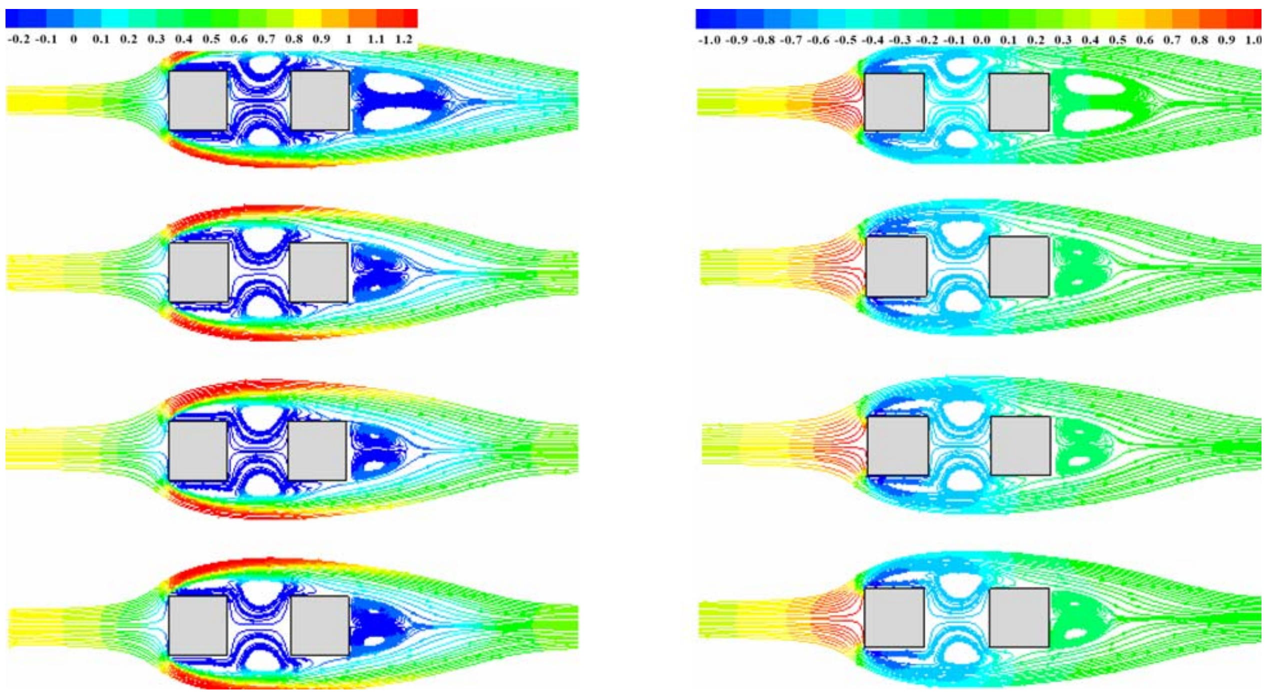

Fig. 5 The time-averaged streamline colored by X-velocity component (left side graphs) and pressure (right side graph) for Reynolds number, Re=2700, 16000, 40000, 70000 from top to down, respectively and the distance between the cylinders, G=1

The growth of Reynolds number may be interpreted as inertia forces rising in comparison with viscous force. This ratio generates the flow disturbances, which do not easily merge with each other in order to reinforce it. Therefore, RMS of lift and drag forces are increased. Since viscous force, location of separation phenomenon on cylinder’s wall as well as flow pattern including wake regions have not considerably been changed, therefore, all of global quantities remain steady. This issue is indicated in reference [16] where, the flow around a single cylinder is investigated and has shown that the flow pattern and global quantities remain almost constant for Re≥20000. The same results are observed in present study, where the RMS of lift force coefficients in the range of Reynolds numbers, Re≥16000 is almost constant (see Figure 6). Figure 6 shows the variation of RMS of lift force coefficient of upstream and downstream cylinders. It is noted that the increase of Reynolds number rises the RMS of lift force coefficient where, it is very impressive in the range of Reynolds number, Re=2700-16000. However, for higher Reynolds number the value of RMS of lift force coefficient is almost independent of Reynolds number and only depends on the distance between the cylinders. Furthermore, RMS of drag force coefficient exhibits the same trend in comparison with RMS of lift as shown in Figure 7. The provided results are same compared with the presented result for single cylinder in reference [16]. As shown in Figure 3, increasing the distance between the cylinders, G=1 to G=2, causes great change of flow structure and isotherm pattern. This flow pattern reformation from steady to unsteady-periodic regimes causes excessive rise in RMS forces.

Figure 7 illustrates the variation of RMS of drag force coefficients of cylinders versus Reynolds number where, due to flow regime changing from steady to unsteady-periodic, the strong jump can be seen about the distance between the cylinders, G=1-2 for upstream cylinder. In addition, while the established eddies from the upstream cylinder strike with the downstream cylinder, the vortexes' strength weakens and forces of RMS lessen. This point can be seen in Figure 7 where, the RMS forces decreased in distance between the cylinders, G=2-4 for all investigated Reynolds numbers. It is noted that the elongation of the distance between the cylinders caused increasing the RMS of forces coefficients of the downstream cylinder. The reason for this could be due to rise of fluctuations in flow and interaction of incoming upstream vortex shedding with downstream cylinder. Figure 8 shows the variation of Strouhal number versus Reynolds number. The vortex transmission velocity or the Strouhal number is a function of the Reynolds number and the distance between the cylinders. The provided numerical results show that the Strouhal number increased versus elongation of the distance between the cylinders for turbulent flow regime and is almost independent of Reynolds number, Re≥16000. Figure 9 shows the instantaneous isotherm pattern contours vicinity of cylinders for different Reynolds numbers and the distances between the cylinders. It could be seen to be identical due to the no-considered change for flow pattern and heat transfer specially, about downstream cylinder for Reynolds number, Re≥16000. On the other hand, it may be predicted that the existence of downstream cylinder in the wake region of upstream cylinder, do not make any change in the temperature pattern in the vicinity of downstream cylinder.

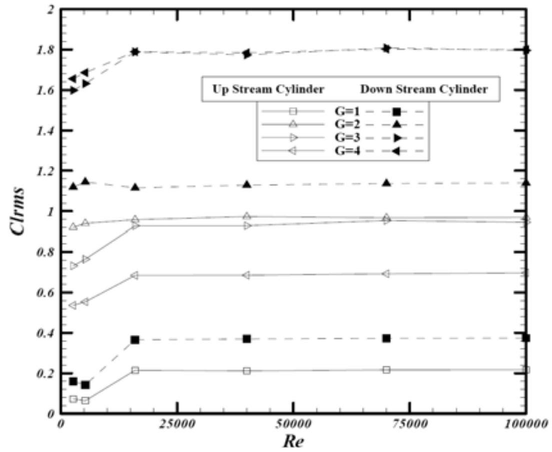

Fig. 6 Variation of RMS of lift force versus Reynolds number

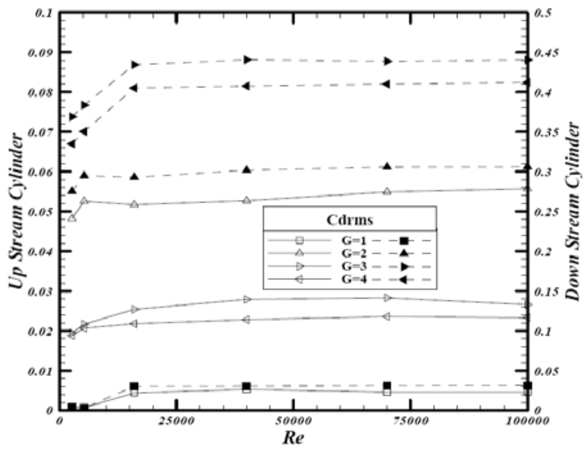

Fig. 7 Variation of RMS of drag force versus Reynolds number

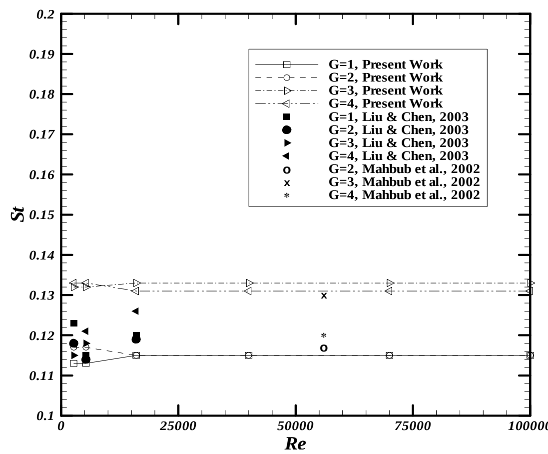

Fig. 8 Variation of Strouhal number versus Reynolds number

While, the ever the increasing of Reynolds number increases the heat transfer from the upstream cylinder. It could be seen, increasing of the distance between cylinders changes the flow regime from steady to unsteady-periodic, see Figures 9-a to 9-d, respectively. Also, increasing the Reynolds number compressed isotherm line around the cylinder's walls where, heat transfer has more flux.

.

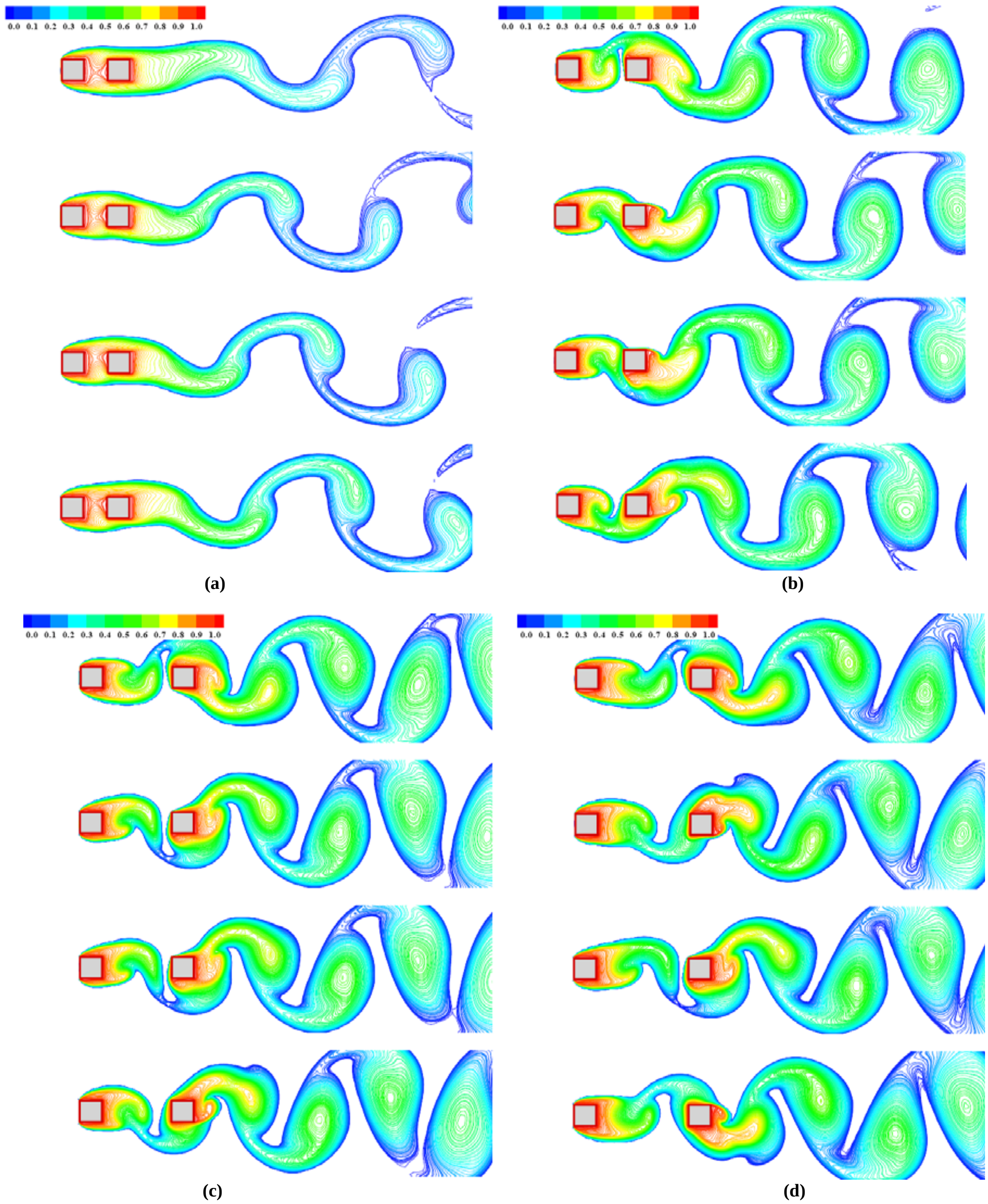

Fig. 9 Instantaneous isotherm pattern contours vicinity of cylinders for the Reynolds number Re=2700, 16000, 40000, 70000 from top to down, respectively: (a) G=1; (b) G=2 ; (c) G=3 ; (d) G=4

In Figure 10 the variation of Nusselt number versus Reynolds number has been shown for different distances between the cylinders. Increasing the Reynolds number and distance between the cylinders may lead to the rise in the Nusselt numbers of cylinders. The lowest Nusselt number is observed for G=1, where rotational regions between the cylinders are symmetric and interaction between them and surrounding zone is in lowest level.

The supplied numerical results indicate that, the total Nusselt number on the front side of upstream cylinder is more than the other sides. Moreover, the lowest Nusselt number relates to the rear side of cylinders due to the eddy region formation near the cylinders walls.

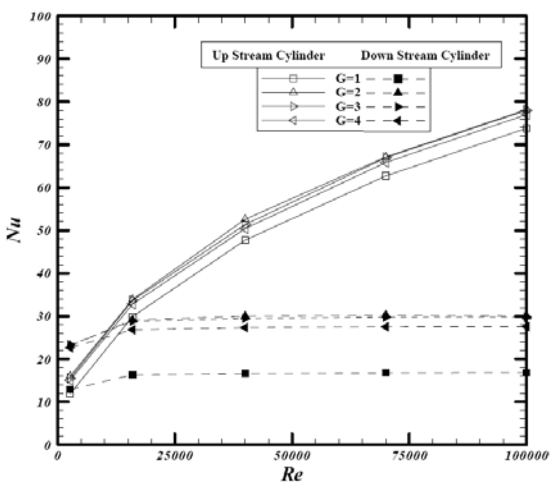

Fig. 10 Variation of Nusselt number versus Reynolds number

4 CONCLUSION

In this research, flow structure and heat transfer characteristics around two equal in-tandem square cylinders in the two dimensional laminar and turbulent flow regimes are simulated numerically for Reynolds and Prandtl numbers, $Re=1\text{-}1\times10^5$, $Pr=0.71$, respectively. An interesting phenomenon has been observed in the flow patterns depending upon the Reynolds number and the distance between the cylinders. A switch over in the nature of the fluctuations of the lift and drag coefficients has been also observed with the increase of Reynolds number and the distance between the cylinders.

It is found that in the range of Reynolds numbers, $Re \geq 16000$ a major change in results is not observed and this confirms that the various parameters at high Reynolds numbers fluid flow field and temperature is almost independent of Reynolds number. In the special distance between the cylinders, flow patterns and global quantities suddenly deformed due to the vast changing in flow pattern and becoming steady flow to periodic-unsteady flow regime.

By elongation of the distance between the cylinders, RMS of lift and drag forces, drag force coefficient of downstream cylinder, Strouhal number and Nusselt number of upstream cylinder have been increased, while Nusselt number of downstream cylinder has no considerable change. The separated heated-rotational regions, propagated at the downstream flow with specific frequency, identifies the rate of heat transfer and the Strouhal number.

5 NOMENCLATURE

Cd	Total drag coefficient
Cdf	Viscous drag coefficient
Cdp	Pressure drag coefficient
Cd_{rms}	Root mean square (RMS) of the drag coefficient
Cl_{rms}	Root mean square (RMS) of the lift coefficient
Cp	Pressure coefficient
d	Width of square cylinders
f	Vortex shedding frequency
G	Spacing between the cylinders
H	Width of computational domain
h	Convective heat transfer coefficient
i	Horizontal axis of coordinate
j	Vertical axis of coordinate
k	Fluid thermal conductivity coefficient
n	Normal direction on cylinders' walls
Nu	Nusselt number
p	Pressure
P	Non-dimensional pressure
Pr	Prandtl number
Re	Reynolds number
Re_t	Turbulent Reynolds number
St	Strouhal number
t	Time
t′	Instantaneous temperature component
T	Fluid temperature
T_{in}	Free stream temperature
T_{wall}	Cylinders' wall temperature
u	Stream wise velocity
U	Non-dimensional stream wise velocity
U′	Instantaneous X-velocity component
u_{in}	Free stream velocity
v	Cross-stream velocity
V	Non-dimensional cross-stream velocity
v′	Instantaneous Y-velocity component
x	Stream wise dimension coordinate
X	Non-dimensional streamwise dimension of coordinate
Xd	Non-dimensional stream wise distance between the rear side of the downstream cylinder and the exit

Xu	Non-dimensional stream wise distance between the inlet plane and the front side of the upstream cylinder
y	Cross-stream dimension of coordinate

Greek letters

α	Thermal diffusion coefficient
ε	Turbulent kinetic energy dissipation
θ	Non-dimensional temperature
θ'	Instantaneous dimensionless temperature component
ρ	Fluid density
κ	Turbulent kinetic energy
ν	Fluid kinematic viscosity
ν_t	Turbulent kinematic viscosity
τ	Dimensionless time
$\Delta\tau$	Dimensionless time step
Θ	Dimensionless temperature
K	Dimensionless turbulent kinetic energy
Σ	Dimensionless turbulent kinetic energy dissipation

ACKNOWLEDGMENTS

I would like to appreciate Prof. Ahmad Sohankar for his expert advices and encouragement to provide this research.

REFERENCES

[1] Lyn, D. A., Einav, S., Rodi, W., and Park, J. H., "A laser-doppler velocimeter study of ensemble-averaged characteristics of the turbulent near wake of a square cylinder", Journal of Fluid Mechanics, Vol. 304, 1995, pp. 285-319.

[2] Williamson, C. H. K., "Three-Dimensional wake transition", Journal of Fluid Mechanics, Vol. 328, 1996, pp. 345-407.

[3] Wong, P. T. Y., Ko, N. W. M. and Chiu, A. Y. W., "Flow characteristics around two parallel adjacent square cylinders of different sizes", Journal of Wind Engineering and Industrial Aerodynamics, Vol. 54-55, 1995, pp. 263-275.

[4] Alvarez, J., Pap, M. and Valencia, A., "Turbulent heat transfer in a channel with bars in tandem and in side by side arrangements", Int. Journal of Numerical Methods For Heat and Fluid Flow, Vol. 10, No. 8, 2000, pp. 877-895.

[5] Niceno, B., Dronkers, A. D. T. and Hanjalic, K., "Turbulent heat transfer from a multi-layered wall-mounted cube matrix: a large eddy simulation", Int. Journal of Heat and Fluid Flow, Vol. 23, 2002, pp. 173-185.

[6] Valencia, A. and Cid, M., "Turbulent unsteady flow and heat transfer in channels with periodically mounted square bars", Int. Journal of Heat and Mass Transfer, Vol. 45, 2002, pp. 1661-1673.

[7] Lin, J. C., Yang, Y. and Rockwell, D., "Flow past two cylinders in tandem: instantaneous and averaged flow structure", Journal of Fluids and Structures, Vol. 16, No. 8, 2002, pp. 1059-1071.

[8] Chia-Hung, L. and Jerry M. Ch., "Observation of hysteresis in flow around two square cylinders in a tandem arrangement", Journal of Wind Engineering and Industrial Aerodynamics, Vol. 90, 2002, pp. 1019-1050.

[9] Mahbub Alam, M. D.., Moriya, M., Takai, K. and Sakamoto, H., "Suppression of fluid forces acting on two square prisms in a tandem arrangement by passive control of flow", Journal of Fluids and Structures, Vol. 16, No. 8, 2002, pp. 1073-1092.

[10] Mahbub Alam, M. D., Moriya, M., and Sakamoto, H., "Aerodynamic characteristics of two side-by-side circular cylinders and application of wavelet analysis on the switching phenomenon", Journal of Fluids and Structures, Vol. 18, 2003, pp. 325-346.

[11] Mahbub Alam, M. D., Moriya, M., Takai, K. and Sakamoto, H., "Fluctuating fluid forces acting on two circular cylinders in a tandem arrangement at a subcritical reynolds number", Journal of Wind Engineering and Industrial Aerodynamics, Vol. 91, 2003, pp. 139-154.

[12] Mahbub Alam, M. D. and Sakamoto, H., "Investigation of strouhal frequencies of two staggered bluff bodies and detection of multistable flow by wavelets", Journal of Fluids and Structures, Vol. 20, 2005, pp. 425-449.

[13] Mahbub Alam, M. D., Sakamoto, H. and Zhou, Y., "Determination of flow configurations and fluid forces action on two staggered circular cylinders of equal diameter in cross-flow", Journal of Fluids and Structures, Vol. 21, 2005, pp. 363-394.

[14] Mahbub Alam, M. D., Sakamoto, H. and Zhou, Y., "Effect of T-shaped plate on reduction in fluid forces on two tandem cylinders in a cross-flow", Journal of Wind Engineering and Industrial Aerodynamics, Vol. 94, 2006, pp. 525-551.

[15] Etminan, A., "Numerical simulation of fluid flow and heat transfer over two square cylinders", MSc. Thesis, Mechanical Engineering Department, Yazd University, Yazd, Iran, 2007.

[16] Sohankar, A., Norberg, C. and Davidson, L. "Low-reynolds-number flow around a square cylinder at incidence: study of blockage, onset of vortex shedding and outlet boundary condition", Int. Journal for Numerical Methods in Fluids, Vol. 26, 1998, pp. 39-56.

[17] Sohankar, A. and Etminan, A., "Forced-convection heat transfer from tandem square cylinders in cross flow at low reynolds numbers", Int. Journal for Numerical Methods in Fluids, Vol. 60, 2009, pp. 733-751.

[18] Etminan, A. and Barzegar, A. "Instantaneous and time-averaged flow structure around two heated square cylinders in the laminar flow regime", Applied Mechanics and Materials Journal, Vol. 110-116, 2012, pp. 2898-2902.

[19] Etminan, A., Moosavi, M. and Ghaedsharafi, N. "Determination of flow configurations and fluid forces acting on two tandem square cylinders in cross-flow and its wake patterns", Int. Journal of Mechanics, Vol. 5, 2011, pp. 63-74.

[20] Thompson, J. F., Warsi, Z. U. A. and Mastin, C. W., "Numerical grid generation, foundations and applications", Elsevier Science, New York, 1985, pp. 305-310.