Convective Heat Transfer Analysis of Tandem Square Cylinders with Edge Modifications

Debasish Biswas
Assistant Professor, Department of Mechanical Engineering,
Kalyani Government Engineering College, Kalyani, India

Abstract: Mixed convection heat transfer has been gone through in the present study for two tandem square cylinders for different edge modifications. Three different edge modifications (square, square with square-cut edge, square with chamfer) have been performed numerically for confined tandem square cylinders with low Reynolds number (Re) for both the working fluid of air (Pr = 0.7) and water (Pr = 7.56) with fixed blockage ratio of 0.25. 2-D numerical analysis has been studied using Ansys Fluent software. Reynolds number has been varied from 50 to 300 with the step of 50 and variation of the same has been plotted with Nusselt number (Nu) for both the working fluid of all the three different kind of edges.

Keywords: Two square cylinders, Edge modification, Nusselt number, Prandtle number, Reynolds number;

I. INTRODUCTION

In the last few decades, it has been found a great interest for the researchers to the Newtonian fluid flow over cylinders with square and circular cross sections which have been oriented perpendicular with the flow directions. Such research have found the momentum from analytical contemplations, due to the extensive form the phenomena of fluid glide which is interrelated with the idealized structures, seeing that a dependable expertise of engineering parameters (drag coefficient, Nusselt number, wake size, etc.) is regularly wanted for the layout of different applications of interest, such as cooling towers, chimneys, etc. Consequently, over a long period of time researchers are gathered a lot of information. Most of them which are found in literature are basically for circular cylinders [1-6]. In the circular cylinder the main problem associated with that there is no fixed separation points which can be eliminated in the square cylinders. This can attribute a quite separate case and hence that is the primary focus of interest in the present study.

Obviously, currently not simplest the values of the engineering parameters, as well as drag coefficient, wake size, Nusselt number (*Nu*), etc., vary from one governance to another still those additionally swank distinctive dependence at the Reynolds number (*Re*) and Peclet number (*Pe*). Still, ahead of task associate degree comprehensive donation and dialogue of this drawback, its miles good to count in brief the modern fissionability of the applicable literature.

Two tandem square cylinders with an insulated vertical channel for lower side of the Reynolds number at the specific range of ($1 \le Re \le 30$) has been investigated numerically for the mixed convection heat transfer by Chatterjee [7]. Opposed convection for particularly air with *Pr* of 0.7 which is the case for air, has been investigated by him for Richardson number (*Ri*) in the ranges of ($-1 \le Ri \le 1$). The study of vertical duct with five in-line cooled/heated square cylinders has been carried out by Chatterjee and Raja [8]. They have been found out the rate of convective heat transfer by the numerical analysis. Different degrees of blockage ratios of 0%, 10%, 25% and 50% have been implemented by them to analyse heat transfer rate and investigate the variation of *Nu*. The thermal transport and the flow visualization have been interpreted for the presentation of the vorticity, representative streamlines and isothermal patterns. The sequel action of distinct blockage ratios (*Br*) ($Br = 0.1$, 0.3 and 0.5) and for the fixed value of *Re* of 100 at $-1 \le Ri \le 1$ for a fixed *Pr* of *0.7* which is air for square cylinders on an upward flow regime and the numerical method have been implemented to measure rate of heat transfer by Sharma and Eswaran [9]. The fluid flow characteristics and the rate of convective heat transfer have been numerically investigated for two equal square cylinders for the arrangement tandem in nature for the lower region of *Re* which has been considered in the ranges of 1 to 200 and the working fluid has been assumed as air with fixed value of *Pr* is 0.71. For two same size isothermal square cylinders flow characteristics and mixed convection heat transfer have been analyzed by Chatterjee and Amiroudine [11]. They have been chosen a geometrical arrangement as tandem cylinders which are placed within the channel for the lower values of *Re*. With *Re* in the ranges of 1 to 30 they have investigated for two different *Pr* of 0.7 and 100 for the *Ri* ranges from 0 to 1 at a fixed single *Br* of 10%. For this particular flow conditions and the geometrical configurations they have been analyzed the vortex shedding, representative stream lines and isotherm patterns. The consequences of the distance between two plates and the cylinder spacing and the effect of *Ri* on the fluid flow characteristics and the phenomena of the heat transfer has been investigated by Lacroix and Carrier [12] for vertically separated horizontal cylinders. Two square bars of equal in dimensions arranged by side by side in a horizontal channel have been studied for both steady and unsteady flow by Durga Prasad and Dhiman [13] in the *Re* ranges from 10 to 100 and *Pr* ranges from 0.7 to 50. They have been varied the gap ratio between two bars. The different gap ratios chosen by them are 1.5, 2, 2.5, 5 and 10 for the particular *Br* of 1/18. For the said conditions they have investigated and presented Strouhal number, total drag coefficient and average value of *Nu*. From their analysis it has been observed that Reynolds number has an adverse effect on overall drag coefficient. A row which comprises with five square cylinders that has been arranged in a side-by-side manner has been studied Chatterjee *et al.* [14] numerical analysis. The sequel of the distance between five cylinders has been investigated by them to analyze the mechanism of the wake structure and vortex shedding. A numerical analysis of square cylinders in a confined horizontal channel of two same size tandem arranged has been carried out by Huang *et al.* [15] for analyzing mixed convection heat transfer for the equal cylinder spacing of four different width of cylinders with a fixed *Br* of 0.1 and for a particular vale of *Pr* of 0.7 (air). They have been varied the values of *Re* from 80 to 150 and of *Ri* which is ranges from 0 to 1. The impact of thermal buoyancy which is a function of *Ri* has been carried out by them for analyzing the fluid flow and the rate of heat transfer. Sensitiveness of lift coefficient compared to drag coefficient is more which has been shown by them. Unsteady periodic flow regime has been investigated for square cylinder in a horizontal channel with the parameters of $60 \leq Re \leq 160$ and $0.7 \leq Pr \leq 50$ has been investigated by Sahu *et al.* [16] through the numerical analysis. For both the constant temperature model (CTM) and constant heat flux model (CHFM) they have been found out a simple heat transfer correlation.

Heat transfer that is mixed convective in nature has been gone through in this present study for two tandem square cylinders for different edge modification. Three different edge modifications (square, square with square cut edge, square with chamfer) have been performed numerically for confined tandem square cylinders with low Reynolds number for both the working fluid of air ($Pr = 0.7$) and water ($Pr = 7.56$) with fixed *Br* of 0.25. The main objective in this current investigation is to study the effect of edge modification on the fluid flow characteristics and the rate of heat transfer phenomena which is associated with *Nu*. 2-D numerical analysis has been studied using Ansys Fluent software. *Re* has been varied from 50 to 300 with the step of 50 and variation of the same has been plotted with *Nu* for both the working fluid of all the three different kind of edges.

II. PROBLEM DESCRIPTION

The gadget of interest right in the present work is the constrained waft of an incompressible fluid which is flowing through the channel with a cylinder that is heated and rectangular in shape positioned symmetrically at the centre-line which has been depicted in Fig. 1 in which the computational domain has been depicted with the boundary conditions. The various dimensions have been taken according to article [11]. After that two shapes will be placed in the place of square shaped bluff bodies in the same position. The magnified views of other two shapes such as chamfered corners and square cutting corners have been shown in Fig. 2 and Fig. 3 respectively. At the inlet the square cylinder is revealed to a velocity field which is parabolic in nature. Here, the maximum velocity has been considered as u_∞ at the channel inlet and a constant temperature has been considered as $T\infty$ at channel inlet.

 ISSN: 2582-8150

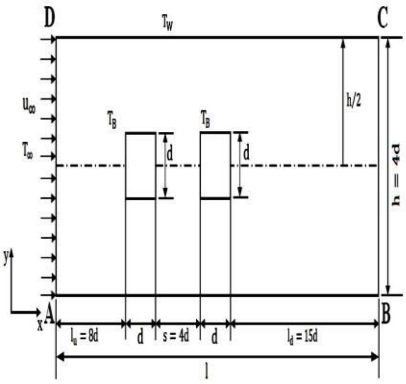

Figure1: Computational domain for square cylinder.

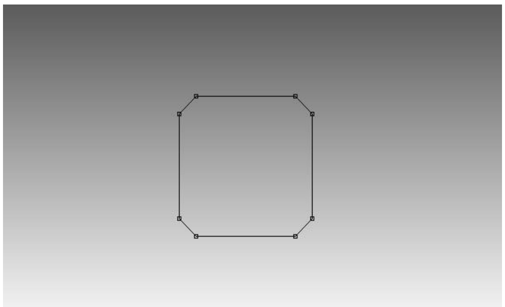

Figure 2: Square shaped with chamfered corners.

Figure 3: Square shaped square end corners.

III. NUMERICAL METHODOLOGY

The structure of the grid which has been incorporated here has been shown in Fig. 4. A non-uniform grid structure has been considered for throughout the computational domain. It includes five distinct zones which have both the uniform and non-uniform grid distribution with smaller grids in the region where the gradient is more or the focus of study lie down and the grids are coarser in size in the areas where gradients are less in nature. A clustered mesh with very less in size is used in the nearby region to cylinders for capturing the interactions between wake and wall acceptably in both the directions. Also near the upper and lower walls where the wake-wall interactivity is more a fine clustered grid is implemented. Here two distinct steps are executed for individual time level. In first condition using previous time-level pressure field a speculated velocity is acquired from the discretized momentum equation. Then in the second step pressure correction equation has been iterated for getting velocity corrections. By solving the equation of momentum which is Navier–Stokes equations a velocity field has been acquired which has been considered as the input of the equation of energy.

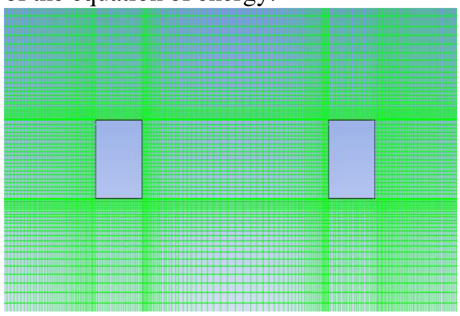

Figure 4: Grid Structure of Square shaped bluff bodies

For the calculation and analyzing of velocity distribution and temperature profile around the considered bodies the governing equations are solved. All the equations for mass momentum and the thermal energy are solved in dimensionless forms with minimal effect of dissipation and with the fixed thermo-physical properties which have been considered from article [11].

IV. RESULTS AND DISCUSSIONS

The temperature profiles by assuming the Reynolds number ranges from 50 to 300 have been evaluated with step size 50, Prandtl Number for air 0.71 and for water 7.56 and with the fixed blockage ratio of 0.25 for three different edge modifications of bluff body. Temperature profile for *Re=50* for both air and water with three different shapes have been given below.

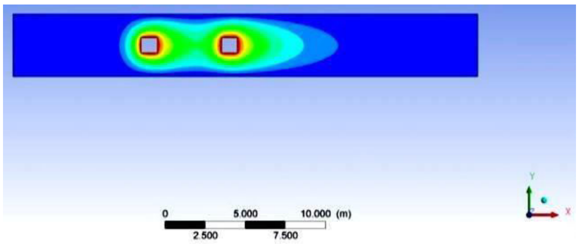

Figure 5: Temperature Profile for *Re=50* (Air-Square).

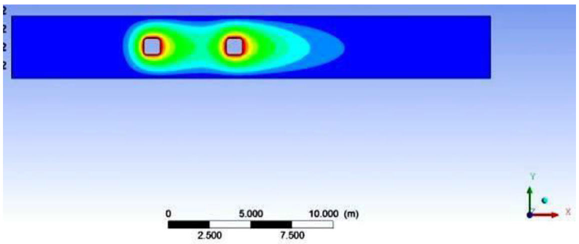

Figure 7: Temperature Profile for *Re=50* (Air-Square with chamfered end).

Figure 8: Temperature Profile for *Re=50* (Water- Square with chamfered end).

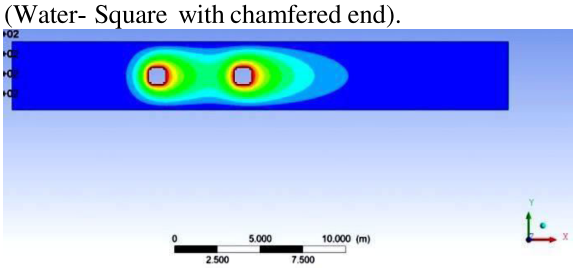

Figure 9: Temperature Profile for *Re=50* (Air-Square with square end).

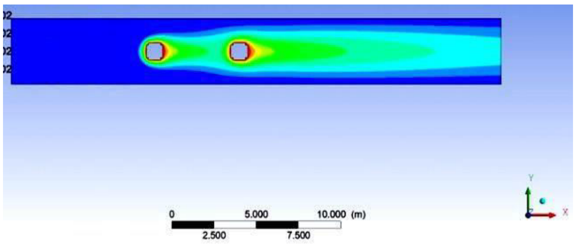

Figure 10: Temperature Profile for Re=50 (Water- Square with square end).

A. Average Nusselt Number (Nu)

For all the three different edge modified shapes (i.e. for square, square shape with corner chamfer and square shape with square corner body), the distinction of average *Nu* with the variation of *Re* have been plotted for both air and water in the Fig. 11 and Fig. 12 respectively.

Figure 6: Temperature Profile for *Re=50* (Water-Square).

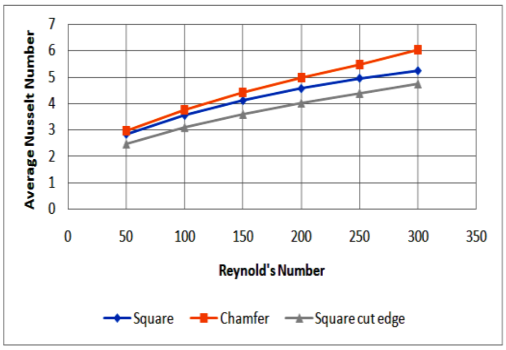

Figure 11: *Nu* vs *Re* for Air

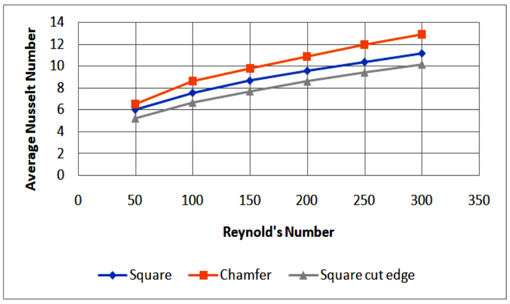

Figure 12: *Nu* vs *Re* for Water

From the above two figures it can be observed that the value of *Re* has proportional effect on average values of *Nu*. And it has been found that the maximum value of average *Nu* is for chamfer edge and minimum for square cut edge. Again, if a comparison is made between air and water it has been observed that water is in higher side in the value for the *Nu*. Hence, this implies the fact of the rate of heat transfer is more when water is flowing through the channel compared to air flowing through the channel. So, the variation in *Pr* and *Re* has a huge impact on the rate of heat transfer enhancement.

B. Skin Friction Coefficient

The main component of profile drag is skin friction drag that can be accomplished as a resistance force which is exerted on a moving object within a fluid domain. The major contribution of skin friction drag is due to the fluid viscosity and that can be measured a function of the *Re*. Fig. 13 depicts the deviation of skin friction coefficient with *Re*. From the Fig. 13 it is understandable that the Reynolds number has an adverse effect on skin friction coefficient.

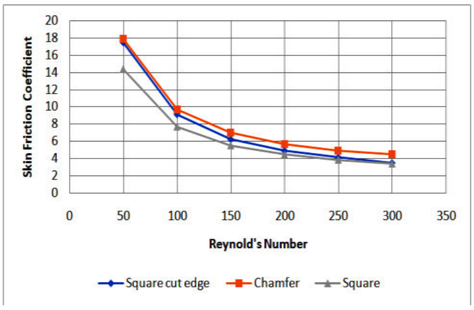

Figure 13: C_f vs *Re* (Same for air and water).

V. CONCLUSIONS

The conclusions that can be drawn from the current work are as follows

1. The effective nature of *Re* and *Pr* on the heat transfer phenomena around two bluff bodies placed adjacently and collinearly have been depicted. The value of *Pr* has favorable effects on the calculated values of the *Nu* for isothermal boundary conditions for a particular value of the *Re*. Also, the value of *Re* has a favorable effect on *Nu* which enhances the rate of heat transfer for the fixed value of *Pr*.
2. The *Re* has adverse effects on skin friction co-efficient where skin friction co-efficient remains constant with the variation of the value of *Pr*.

REFERENCES

[1] A. Venugopal, A. Agrawal and S.V. Prabhu, "Span wise correlations in the wake of a circular cylinder and a trapezoid placed inside a circular pipe", Journal of Fluids and Structures, Vol. 54, 2015, pp 536-547.

[2] A. Kondjoyan and J.D. Daudin, "Effects of free stream turbulence intensity on heat and mass transfers at the surface of a circular cylinder and an elliptical cylinder, axis ratio 4", International Journal Heat Mass Transfer, Vol. 38(10), 1995, pp. 1735-1749.

[3] S. Sanitjai and R.J. Goldstein, "Forced convection heat transfer from a circular cylinder in cross flow to air and liquids", International Journal Heat Mass Transfer, Vol. 12, 2004, pp. 4795-4805.

[4] A.C. Benim, E. Pasqualotto and S.H. Suh, "Modelling turbulent flow past a circular cylinder by RANS, URANS, LES and DES", Progress in Computational Fluid Dynamics, Vol. 8, 2008, pp. 299-307.

[5] H. Ding, C. Shu, K.S. Yeo and D. Xu, "Simulation of incompressible viscous flows past a circular cylinder by hybrid FD scheme and meshless least square-based finite difference method", Computer Methods in Applied Mechanics and Engineering, Vol. 193, 2004, pp. 727-744.

[6] R.D. Selvakumar S. Dhinakaran "Forced convective heat transfer of nanofluids around a circular bluff body with the effects of slip velocity using a multi-phase mixture model", International Journal of Heat and Mass Transfer, Vol. 106, March 2017, pp. 816-828.

[7] D. Chatterjee, "Mixed Convection Heat Transfer from Tandem Square Cylinders in a Vertical Channel at Low Reynolds Numbers", Numerical Heat Transfer, Vol. 58, 2010 pp 740-755.

[8] D. Chatterjee and Md. Raja, "Mixed Convection Heat Transfer Past In-Line Square Cylinders in a Vertical Duct", Thermal Science, Vol. 17, No. 2, 2013, pp 567-580.

[9] A. Sharma and V. Eswaran, "Effect of Channel-Confinement and Aiding/Opposing Buoyancy on the Two- Dimensional Laminar Flow and Heat Transfer across a Square Cylinder", International Journal of Heat Mass Transfer, Vol. 48, 2005, pp 5310–5322.

[10] A. Sohankar and A. Etminan, "Forced Convection Heat Transfer from Tandem Square Cylinders in Cross Flow at Low Reynolds Number", International Journal for Numerical Methods in Fluids, Vol. 60, 2008, pp 733-751.

[11] D. Chatterjee and S. Amiroudine, "Two-Dimensional Mixed Convection Heat Transfer from Confined Tandem Square Cylinders in Cross-Flow at Low Reynolds Numbers", International Communications in Heat and Mass Transfer, Vol. 37, 2010, pp 7-16.

[12] M. Lacroix and R. Carrier, "Mixed Convection Heat Transfer from Vertically Separated Horizontal Cylinders within Confining Walls", Numerical Heat Transfer, Part A, Vol. 27(4), 1995, pp 487-498.

[13] A.V.V.S. Durga Prasad and A.K. Dhiman, "CFD Analysis of Momentum and Heat Transfer around a Pair of Square Cylinders in Side-by-side Arrangement", Heat Transfer Engineering, Vol. 35, 2014, pp 398-411.

[14] D. Chatterjee, G. Biswas and S. Amiroudine, "Numerical Simulation of Flow Past Row of Square Cylinders for Various Separation Ratios", Computers & Fluids, Vol. 39, 2010, pp. 49-59.

[15] Z. Huang, G. Xi, W. Zhang and S. Wen, "Mixed Convection Heat Transfer From Confined Tandem Square Cylinders in a Horizontal Channel", International Journal of Heat and Mass Transfer, Vol. 66, 2013, pp 625-631.

[16] A.K. Sahu, R.P. Chhhabra and V. Eswaran, "Effects of Reynolds and Prandtl Number on Heat Transfer From a Square Cylinder in the Unsteady Flow Regime", International Journal of Heat and Mass Transfer, Vol. 52, 2009, pp 839-850.