P. Rajesh Kanna
Professor
Department of Mechanical Engineering
Velammal College of Engineering and Technology
India

M. Sivasubramanian
Assistant Professor
Department of Mechanical Engineering
Kalasalingam University
India

M. Uthayakumar
Professor
Department of Mechanical Engineering
Kalasalingam University
India

Numerical Investigation of Forced Convection Heat Transfer from Square Cylinders in a Channel Covered by Solid wall - Conjugate Situation

Flow over two isothermal offset square cylinders in a confined channel is simulated for different Reynolds number to reveal the forced convection heat transfer from the heated square cylinders to the ambient fluid for the fixed blockage ratio. The channel is covered by solid walls and conjugate heat transfer is considered. Heat transfer from the cylinders to the ambient fluid as well as conducted within solid wall through conjugate interface boundary is investigated for both steady and periodic flow. Simulation is carried out for Reynolds number which varies from 10 to 100 and Prandtl number equals 0.71. The onset of vortex begins when Reynolds number equals to 48. The conjugate interface temperature decreases when Reynolds number increases. The isotherm in the solid wall shows two dimensionality near cylinder region. The upstream cylinder results in higher Nusselt number during steady state and downstream cylinder results in higher Nusselt number for periodic flow.

Keywords: *Conjugate Heat Transfer, Laminar flow, Nusselt number, Offset cylinder*

1. INTRODUCTION

Conjugate heat transfer has many practical applications such as electronics cooling, turbine blade cooling, solar collectors and many more. When solid wall thickness is significant then the assumption as wall temperature as constant is invalid. In conjugate heat transfer mode the heat transfer in solid region is simultaneously solved along with energy equation in fluid region to find steady state results [1]. Flow over bluff body has received attention by many researchers for the past few decades because of its complex fluid dynamics and large industrial applications. Laminar flow over circular cylinder [2] was investigated to understand the vortex shedding, drag and lift coefficient and shedding frequency. Similar study was carried out by Okajima [3] for a square cylinder limited to laminar range. The influence of side walls was examined by Biswas et al [4]. Numerous studies were conducted to explore the angle of incidence in the bluff body [5, 6].

Sharma and Eswaran [7] presented flow and forced convection heat transfer results from square cylinder for blockage ratio (β = d/H) of 0.05 in a free stream. A closed form relation is arrived for Nusselt number as a function of Reynolds number. The effect of blockage ratio on the variation in Strouhal number with Re was numerically investigated by Biswas et al [4]. They documented that the Strouhal number undergoes a slight change with increasing Reynolds number. Sohankar and Etminan [8] numerically investigated flow physics from tandem square cylinders in a free stream condition and they reported that onset of vortex shedding occurs at Re = 40. In all the cases, the recirculation length of the upstream cylinder is larger than the corresponding value for the downstream cylinder. This length increases to a local maximum and further decreases for the upstream cylinder. Fluid present between the two cylinders causes steep uplift in the local Nusselt number in the front face of the downstream cylinder at critical Re.

Dipankar and Mondal [9] later confirmed their results and they revealed the influence of horizontal distance between these tandem cylinders. Recently, the effect of spacing ratio between identical square cylinders placed in an unconfined environment was reported by Chatterjee and Mondal [9]. They found that the flow starts deviating from its symmetric nature when space between tandem cylinders is greater than 4d. Rosales et al. [10] reported that when the Reynolds number increases, the flow shows an oscillatory nature at a relatively lower spacing ratio. Also, it is found that the average Nusselt number for the upstream cylinder approaches that of a single cylinder for spacing ratio beyond its critical value. One of the major findings from their study was the establishment of the critical spacing ratio for different Reynolds numbers above which vortices were seen to form in the gap between the cylinders. Below the critical spacing ratio, the flow within the gap may be either steady or unsteady oscillatory depending on the flow Reynolds number.

Galletti et al. [11] used proper orthogonal decomposition Galerkin model was successfully tested for flow over confined square cylinder. The model was effective in capturing the short- and long-term dynamics for appreciable variations of the Reynolds number.

Received: May 2015, Accepted: August, 2016
Correspondence to: M. Sivasubramanian
Assistant Professor, Department of Mechanical Engineering, Kalasalingam University, INDIA.
E-mail: m.sivasubramanian@klu.ac.in
doi:10.5937/fmet1701016K

Breuer et al. [12] tested lattice-Boltzmann automata (LBA) and finite volume method (FVM) for flow over square cylinder in a confined environment for the blockage ratio of 0.125. Excellent agreement between the LBA and FVM computations was found. The critical Reynolds number for the onset of vortex shedding is 60. Their predictions reveal that the flow separation from the leading edge at higher Reynolds number influences the frequency of vortex shedding.

Dutta et al. [6] performed measurements on a square cylinder to investigate the role of incidence angle and aspect ratio. The measurements reveal that the drag coefficient decreases with an increase in aspect ratio, while the Strouhal number is seen to increase with aspect ratio. Numerical experiments were performed by Valencia and Paredes [13] to understand the flow and heat transfer from square cylinder arranged side by side by varying Re and gap space in normal direction. The Reynolds number and space between square cylinder is varied for fixed blockage ratio equals to 0.125. Results presented for various Reynolds number for fixed gap value and various gap value for fixed Re. Steady flow was observed for larger gap value rather small gap for particular Re. Average Nusselt number declined for higher gap value. Numerical results show that for Re = 200, the flow is steady laminar with a recirculation zone attached to the rear side of the bars gap value 0.75d. Each cylinder shows different dominant frequency which appears for Re > 400 whereas for single cylinder corresponding Reynolds number is 60 [12] for the same blockage ratio.

Rosales et al. [10] carried out numerical experiment by arranging inline and offset tandem cylinders of different size for fixed Reynolds number equals to 500. The results show that the drag coefficient, Strouhal number and cylinder Nusselt number decrease as the heated cylinder approaches the wall. The highest Strouhal number value is observed in the channel-centered, inline-eddy configuration. The presence of small cylinder in the upstream results reduction in the overall heat transfer and influencing the vortex shedding frequency. The bluff body study extended for diamond shaped baffles by Sripattanapipat and Promvonge [14]. The study revealed that the reduction of the baffle angle leads to an increase in the Nusselt number and friction factor. The effects of blockage ratio on the combined free and forced convection from a long heated square obstacle confined in a horizontal channel are numerically investigated [15]. Flow symmetry is lost with the introduction of the cross buoyancy and as the value of the Richardson number gradually increases (ie., Ri > 0).

Recently, Durga Prasad and Dhiman [16] numerically studied confined square cylinder arranged side by side by using commercial software for blockage ratio of 0.055. It is evident that when spacing ratio was increased then the Strouhal number increases initially and further decreases when gap is enlarged between the cylinders. Beyond critical value of this gap each cylinder behaves independently. The maximum crowding of isotherms is found to be on the front surface, indicating the highest Nusselt number, as compared to other surfaces of the cylinders.

Numerous studies were dedicated to document the effect of blockage ratio, effect of confinement, angle of incidence, arrangement of tandem square cylinders, forced and mixed convection heat transfer. From the open literature it is found that the study on offset arrangement of square cylinder is very limited. This study aims to investigate the forced convection heat transfer from two isothermal square cylinders placed in a channel offset to each other. The offset arrangement enhances convec–tion heat transfer within the fluid. The channel top and bottom walls are covered by solid walls which also undergoe conduction mode of heat transfer. The conjugate heat transfer is simulated to investigate the steady and periodic behavior from two offset heated square cylinders placed in a confined channel covered by solid walls.

2. MATHEMATICAL FORMULATION AND NUMERICAL PROCEDURE

In the present study it is aimed to simulate flow over two identical square cylinders placed in a confined channel offset to each other. The offset distance (OD) between the cylinder A and B is fixed as 2d for the entire study. The top and bottom channel wall is covered by solid wall of thickness equal to cylinder thickness. The schematic of the present problem is shown in Fig. 1. The study is limited to two dimensional incompressible flow in laminar range. The fluid Prandtl number is fixed as 0.71. The governing Navier-Stokes equations are solved by well known stream function-vorticity formulation. The non-dimensional form of governing equation are:

Stream function equation

$$\nabla^2 \psi = -\omega \tag{1}$$

Vorticity equation

$$\frac{\partial \omega}{\partial t} + \frac{\partial (u\omega)}{\partial x} + \frac{\partial (v\omega)}{\partial y} = \frac{1}{\mathrm{Re}} \nabla^2 \omega \tag{2}$$

Energy equation in fluid region

$$\frac{\partial \theta}{\partial t} + \frac{\partial (u\theta)}{\partial x} + \frac{\partial (v\theta)}{\partial y} = \frac{1}{\mathrm{Re\,Pr}} \nabla^2 \theta_f \tag{3}$$

Energy equation in solid region

$$\frac{\partial \theta_s}{\partial t} = \left(\frac{\alpha_s}{\alpha_f} \right) * \frac{1}{\mathrm{Re\,Pr}} \nabla^2 \theta_s \tag{4}$$

where ψ is stream function and $u = \frac{\partial \psi}{\partial y}, v = -\frac{\partial \psi}{\partial x}$ and $\omega = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}$.

The variables are scaled as $x = \bar{x} / d$; $y = \bar{y} / d$; $u = \bar{u} / \bar{u}_0$; $v = \bar{v} / \bar{v}_0$; $t = \bar{t} / \left(d / \bar{u}_0 \right)$. The overbar indi–cates dimensional variable and temperature is non dimen–sionalised as $\theta = \left(T - T_{ambient} \right) / \left(T_{cylinder\,wall} - T_{ambient} \right)$; θ_s–Solid wall Temperature. Re is the Reynolds number defined as $\mathrm{Re} = \bar{u}_0 d / \nu$. Here $\bar{u}_0$, is the average horizontal fluid velocity at the channel inlet.

The boundary conditions applied in the present numerical simulation are discussed below. At the channel inlet, parabolic horizontal velocity profile is assumed with an average velocity of ($\bar{u}_0$), and the fully developed flow conditions are applied at the channel exit similar to Breuer et al. [12]. No slip and no penetration conditions are applied for velocity components along all the solid walls. The boundary conditions are as following:

At channel entrance (x = 0):

$$u(y) = y - y^2/6,\ \psi(y) = y^2/2 - y^3/18,$$
$$\omega(y) = y/3 - 1,\ \theta(y) = 0.0 \tag{5}$$

At exit (x = 55):

$$\frac{\partial^2 \psi}{\partial x^2} = 0,\ \frac{\partial \omega}{\partial x} = 0, \frac{\partial u}{\partial x} = 0, \frac{\partial \theta}{\partial x} = 0 \tag{6}$$

Along solid boundaries:

$$u = 0, v = 0 \tag{7}$$

At inlet fluid temperature is set as ambient, ie: $\theta = 0$ and solid walls which covered the channel at top and bottom are maintained at adiabatic. Along conjugate interface boundary heat flux from fluid region is equal to solid region, and $\theta_s = \theta_f$.

$$k_s = \frac{\partial \theta_s}{\partial y} = k_f \frac{\partial \theta_f}{\partial y} \tag{8}$$

The square cylinder walls are maintained at constant temperature as θ = 1. The details of the boundary conditions are shown in figure 1(b). Walls are the sources of vorticity and vorticity is evaluated from the cylinder solid walls as well as from channel walls. Based on inlet, prescribed constant stream function value is assigned along the cylinder walls. The energy equation for solid region Eq. 4 consists of RePr in the right hand side is due to the non dimensionalisation for time and reason was explained in Chiu et al [17] and also used by Kanna and Das [18]. Convergence criteria used for vorticity equation (2), energy equation in fluid (3) and solid (4) is,

$$\varepsilon = \sum_{i,j=1}^{i_{\max}, j_{\max}} \left(\Phi_{i,j}^{n+1} - \Phi_{i,j}^{n} \right) \tag{9}$$

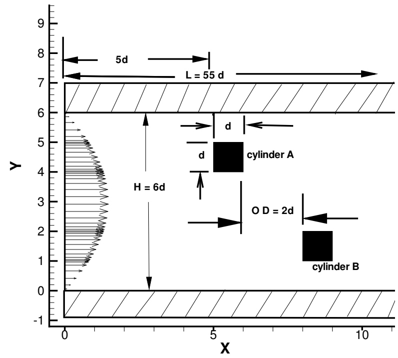
where ϕ will be $\phi(\omega, \theta, \theta_s)$.

The governing equations are discretised on non uniform rectangular grids. The central differencing scheme is adopted for both the convective as well as the diffusive terms [19]. Stream function equation, Eq. (1), is solved by Gauss-Seidel method. The unsteady vorticity transport equation, Eq. (2) and Eq. (3) are solved by Alternate Direction Implicit scheme (ADI), along with Eq. (1), until the steady state solutions are arrived. The details of the numerical procedure of ADI method are reported in a previous work of one of the authors [20]. At every time step, the residue of the discretised vorticity equation are monitored using equation 9 and the solution is accepted only when the sum of all residues become negligibly small as less than 0.0001 for vorticity and 10-6 for energy equation. In addition to this the numerical values of u and v velocities at every time step are monitored to see the changes with time marching. When there are no changes in the variables with further time steps, the solution is considered as the steady state solution. The steady state momentum results are used to solve energy equation for fluid region (3). Equation 9 is used to monitor the steady state results in the solid region. Periodic results have obtained after 10 periods of cycles.

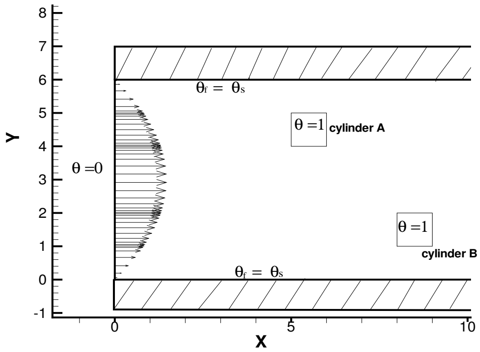
3. CODE VALIDATION AND GRID INDEPENDENCE STUDY

The channel wall is covered by solid material at top and bottom which is common in many industrial applications using heat exchangers. The outer surface of the solid wall is assumed to be adiabatic and inside is getting contact with channel wall. Thus conjugate situation arises. The schematic of the problem is shown in Fig. 1(a). From the open literature to the author's knowledge there is a lack of numerical as well as experimental study for the present geometry. Therefore to validate the present numerical procedure few benchmark problems of square cylinder are solved and compared with available results. Flow over single cylinder in a confined channel, two square cylinders arranged in a channel was solved. Results are presented for both steady and periodic behaviour of square cylinders. It is found that good agreement among the available results and variation is registered in few cases due to the usage of coarse grids for those problems during the present simulation. Flow over single square cylinder placed in a channel was simulated for blockage ratio of 0.125 (Breuer et al. [12]). The creep flow was observed for Re = 1. Flow separates at the rear side of the cylinder wall and two stable pair of vortex is appeared when Re = 30 (Fig. 2(a)). A discrepancy in the reattachment length is observed was due to the usage of coarse grids in the present simulation when compare to Breuer etal. [12] (ie., 8% grids of Breuer et al. [12] only used in the present computation). The Strouhal number (St) is compared with available results of Breuer et al. [12] and Mohammed Rahnama and Kakimeh Hadi-Mohaddam [21] for different Reynolds number (Fig. 2(b)). The St is increased with Reynolds number non linearly to a local maximum and further it is decreased. It is observed that the present numerical simulation shows close agreement with them. Further flow over pair of square cylinders (Sohankar and Etminan [22]; Allanaboyina [16]) also verified. Streamline contour and local Nusselt number were presented for two kinds of cylinder arrangement (Fig. 2(c)-2(f)). It is found that the present method could be able to produce acceptable results for two cylinders placed in a channel.

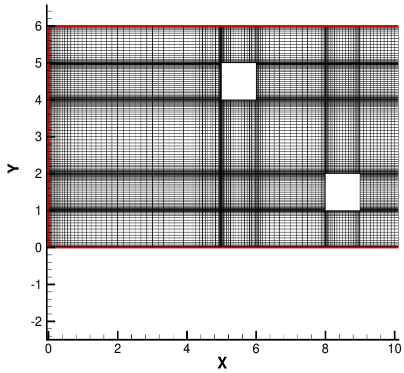
Computational grids 151 * 81(case 1); 201 * 101(case 2); 231 * 121(case 3); 251 * 141(case 4); 301 * 201(case 5) are tested to find the grid dependency on global parameters. Grids 251 * 141 shows less than 0.1% difference in downstream vortex length as well as cylinder average Nusselt number corresponding with grids 301*201. Therefore grids 251 * 141 are chosen to simulate the present problem and grids nearer to the cylinder are shown in figure (Fig. 1(c)-1(d)). Grids are clustered around the cylinder and wide spread in the downstream and hence minimum grid space occurred as 0.007d at the corner of the cylinder where velocity gradient are steep and grid space as 0.13d occurred at far downstream in the channel. It is shown that the present method could be able to produce acceptable results for simulation of offset square cylinder as conjugate case.



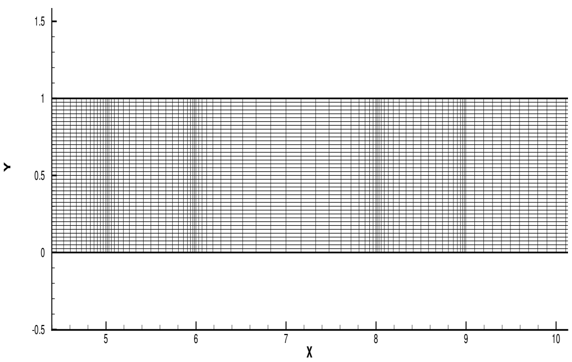
(a) Schematic of the problem



(b) Boundary condition

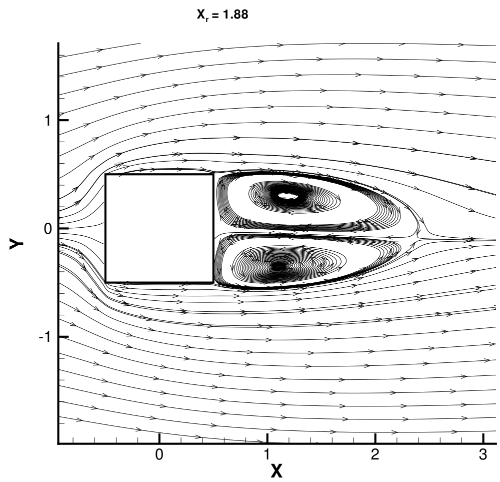


(c) computational grids: Fluid region

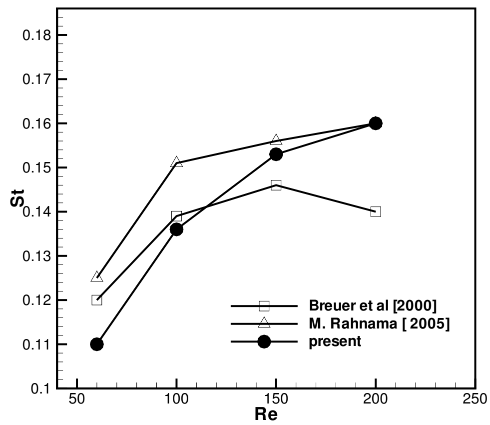


(d) computational grids: solid wall region

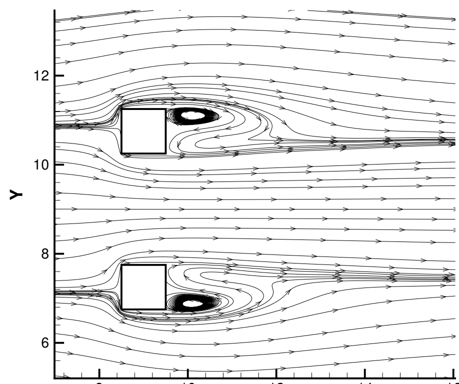
Figure 1. Schematic and boundary conditions of the problem



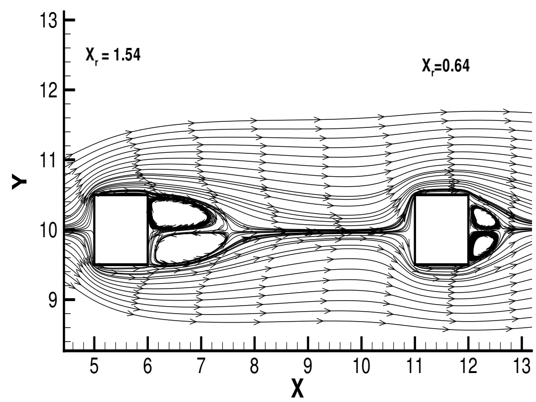
(a) Re = 30: X_r = 1.59 [12]



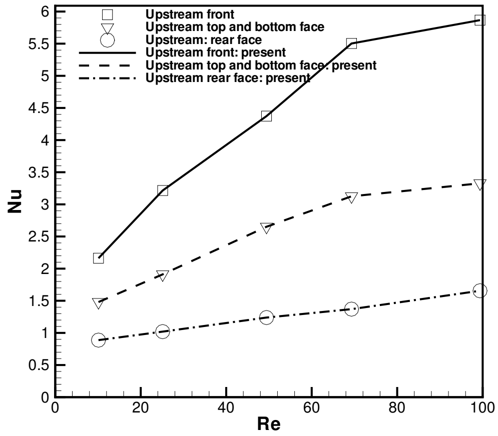
(b) Strouhal number: [12, 21]



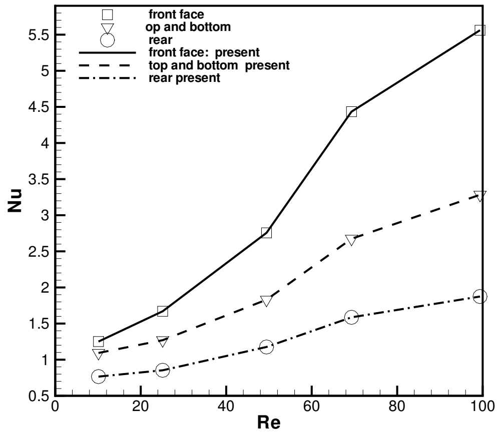
(c) streamline contour: Re = 50 [16]



(d) Re = 20 [22] Up cylinder X_r = 1.66, down cylinder X_r = 0.68



(e) Local Nu([22]) Upstream cylinder

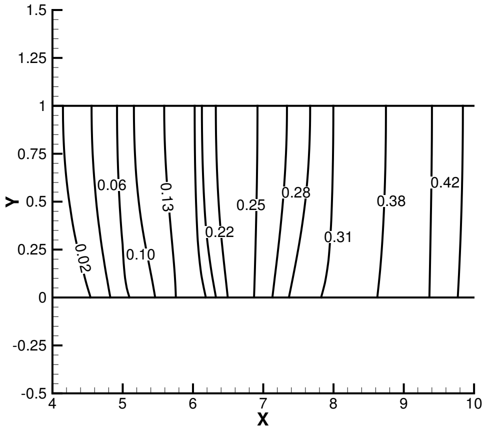


(f) Local Nu([22]) Downstream cylinder

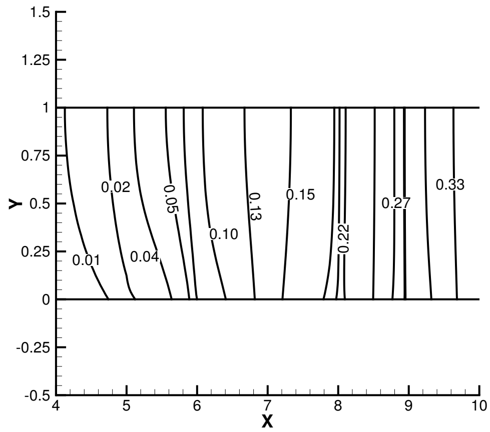
Figure 2. Simulation of square cylinders placed in a confined channel

4. RESULTS AND DISCUSSION

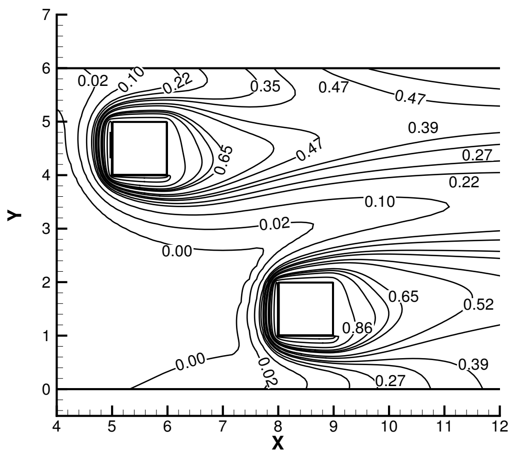
Flow over two offset isothermal square cylinders placed in a confined channel is simulated under conjugate forced convection heat transfer by varying Reynolds number. The results are reported for steady state as well as periodic conditions. The distance between the cylinders in normal direction is fixed as 2d. The blockage ratio (β = d/H) is constant and set as 0.167 and Prandtl number is chosen as 0.71. Along the solid wall interface, channel wall heat flux is the same in the fluid region as well as solid region (Eqn. 8). The solid to fluid conductivity ratio (k_s/k_f) is assumed default value as 1. The cylinders are maintained at constant wall temperature. Simulations are carried out for different Reynolds number and results presented in terms of streamline contour, temperature contour, conjugate interface temperature (θ_b), local and average Nusselt number. The present investigation is focused on finding results under steady state as well as periodic conditions. Results presented for Re = 20 to demonstrate the steady state behaviour and corresponding value for unsteady is Re = 80 and default K = 1.0 for all the case considered. The isotherm contour from solid wall and fluid region around square cylinders are reported in figure 3 for steady state condition. The hot cylinder A is placed at 5d from inlet and the cylinder B is placed at 8d distance from the inlet. It is noticed that around cylinder region the two dimensionality in the temperature contour is perceived (Fig. 3(a), 3(e)) in the solid wall and upstream and far downstream from the cylinder, temperature variation occurs mostly in the streamwise direction rather in the normal direction. This trend is common in both top and bottom solid wall. Figure 3(c) shows the temperature contours in the fluid region. The front face stagnation region isotherm are clustered and spread along the top and bottom face of the cylinder similar to a single cylinder encounter a free stream [7] or double cylinder placed in a confined channel [10]. Fluid exhibit a jet like structure from the gap between top cylinder and top wall due to no slip condition at the cylinder top face and channel wall.



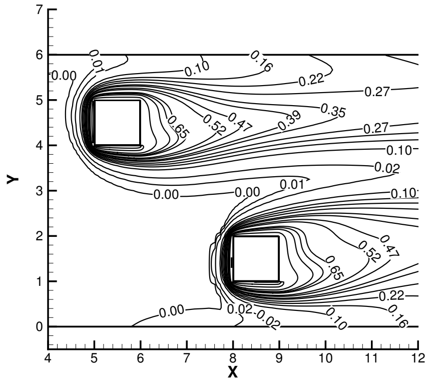
(a) Re = 20 Top solid wall



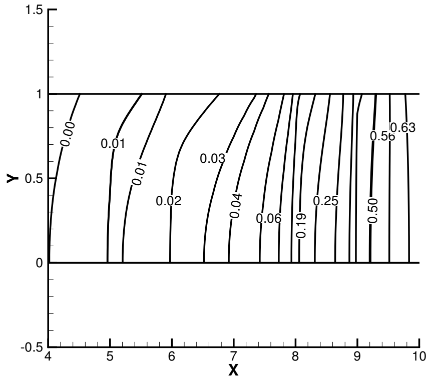
(b) Re = 40 Top solid wall



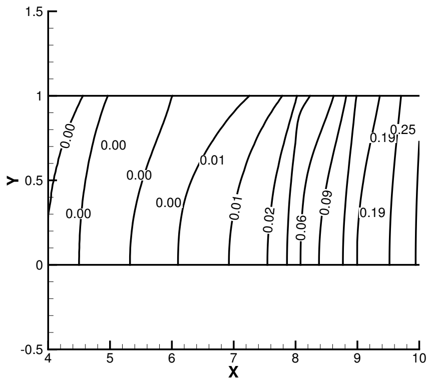
(c) Re = 20 Fluid region



(d) Re = 40 Fluid region



(e) Re = 20 Bottom solid wall

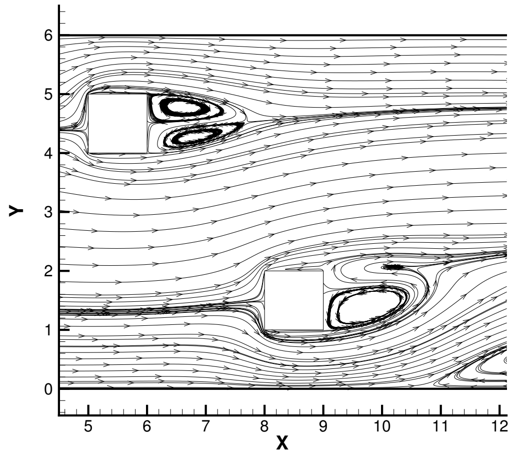


(f) Re = 40 Bottom solid wall

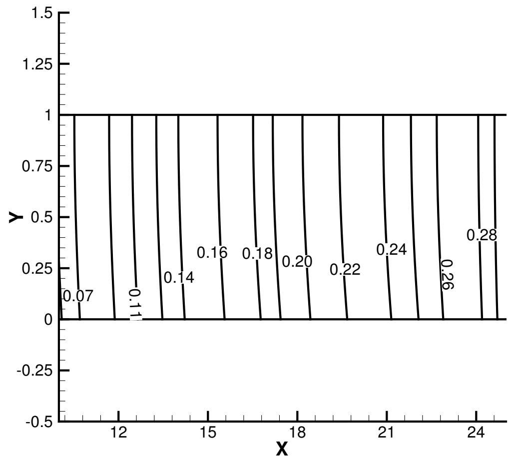
Figure 3. Temperature contour: Steady state condition

The temperature contours also reflect the same. Fluid expand through the gap between bottom cylinder face and channel wall. Therefore isotherm contours spread in the downstream. When Re is increased due to enhanced cooling effect the isotherm values are decreased both in the solid region as well as fluid region (Fig. 3(b), 3(d), 3(f)). Also it is noticed that more isotherm contours clustered in the front face of the cylinder at higher Re values. The two dimensionality is araised in the temperature contour in the bottom solid wall near interface region (Fig. 3(f)). Onset of vortex begins when Reynolds number is increased. To find the critical Reynolds number, the Reynolds number is increased and it is found that up to Re $\leq$ 40 both cylinders exhibit steady state results. The critical Reynolds number for the present offset distance is Recr = 48. Therefore for understanding the unsteady behavior of offset cylinder, simulations are reported for Re = 80.

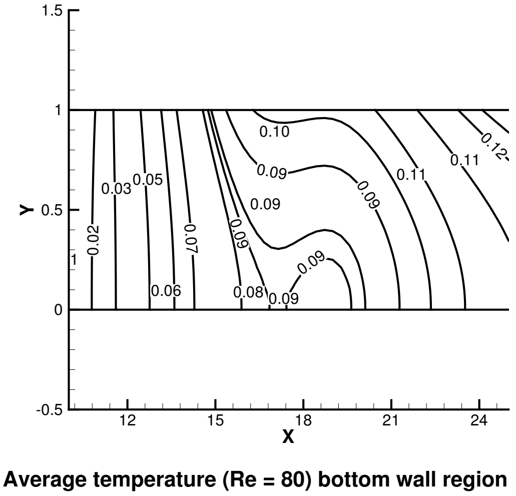
It is observed that from cylinder A flow separates from the rear wall and form a pair of stable vortices. This vortices less affected during a time period. From the bottom cylinder flow separates from the rear wall and forms counter rotating vortex which gradually increases its size and separate from the wall at the end of half cycle. Further this vortex detaches from the cylinder and move downwards. It is also noticed that from the bottom wall flow separates and detach from the wall during vortex shedding. This is identical with Rosales et al [10] for offset square cylinders in a confined channel.



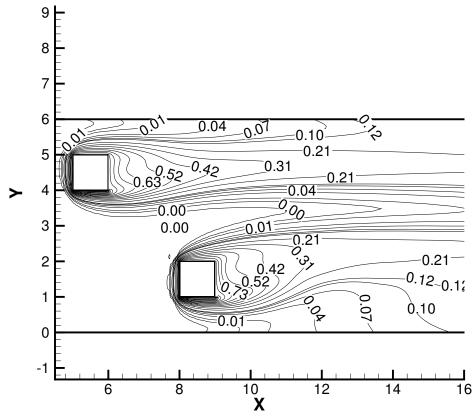
(a) Average streamline contour (Re = 80) fluid region



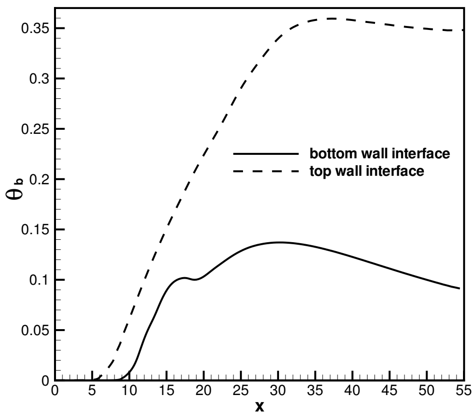
(b) Average temperature (Re = 80) top wall region



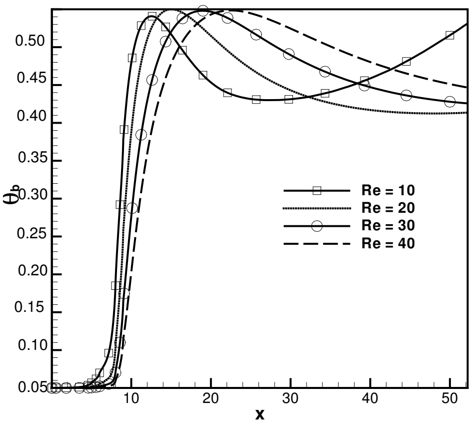
(c) Average temperature (Re = 80) bottom wall region



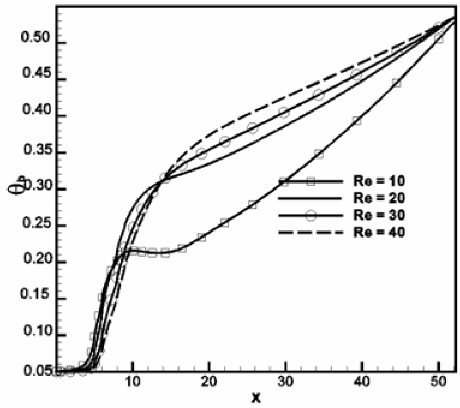
(d) Average temperature (Re = 80) fluid region



(e) Average interface temperature: Re = 80



(f) Bottom wall interface temperature



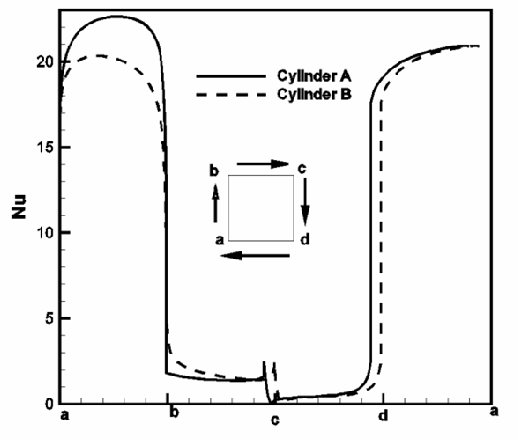
(g) Top wall interface temperature (θb)

Figure 4. Time average periodic temperature: K = 1

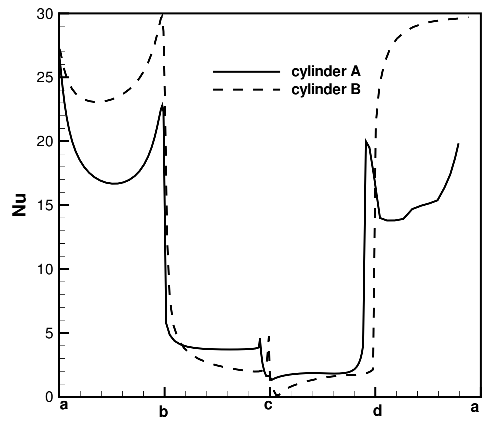
It is clearly evident that in the fluid region due to parabolic inlet profile and offset nature of square cylinder the temperature contours are asymmetrical and differ from square cylinder placed in free stream. The front face is stagnation region where thickening of isotherm contour and thinning in the downstream are occurred. In the downstream of the cylinder flow separates and eddy forms which dictate different temperature contour depending upon the strength of the vortex. Since the downstream cylinder receives not only ambient fluid, but also deflected fluid from the cylinder A, this results in complex flow structure which results the in same in the temperature contour, as well. The time, average streamline contours and temperature contours from the geometry is shown in figure 4. Due to the brevity of the time average temperature contours are shown for solid wall region. Figure 4(b) for isotherm contour for top wall and figure 4(c) shows the bottom wall region. During the entire cycle period almost the top cylinder shows one dimensional temperature dissipation in the x direction for the Reynolds number considered. Since the bottom solid wall region is closely associated with cylinder B, the temperature contours also exhibit two dimensionality in the temperature contours near cylinder B.

The time average interface temperature is shown in figure 4(e). The top wall interface temperature increases to a local maximum and reaches an asymptotic value. The interface temperature from the bottom wall is increased linearly up to few cylinder length distance in the downstream. There is a kink observed along bottom wall interface temperature, which is due to bottom wall flow separation in the downstream of the cylinder B. The effect of Re in the conjugate interface temperature (θ_b) along the bottom and top wall is shown in figure 4(f) and 4(g). The bottom wall interface temperature θ_b, value is very small till cylinder region. Further in the downstream it increases till wake region to reach a local maximum and further it decreases to reach an asymptotic value in the channel region. This is due to the presence of counter rotating wake which enhances the temperature rise in the bottom wall. For the same location when Re is increased the temperature value is decreased due to increased inertia force by ambient fluid. However, at low Reynolds number (Re = 10) fluid encounters difficult to overcome resistance to flow in the long channel conduction tends to influence the heat transfer in the fluid region. Along the top wall the interface temperature continuously increase linearly in the entire downstream location (Fig. 4(g)). However, presence of cylinder wake causes sharp increment in the temperature in the wake region.

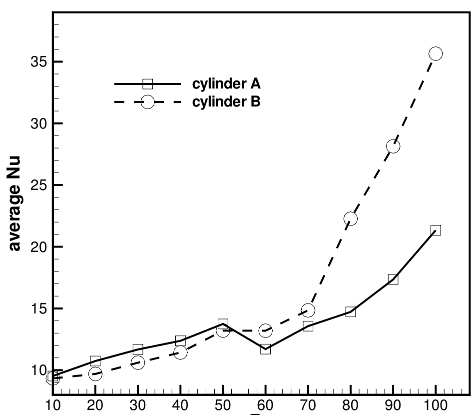
The local and average Nusselt number from the cylinder is reported in figure 5. Figure 5(a) shows the local Nusselt number along the cylinder faces a-b-c-d-a for Re=20 and figure 5(b) shows the corresponding results for Re =80. The stagnation face results in high Nusselt number similar to single cylinder [21] placed in a channel. The top and rear face of cylinders results in lower local Nusselt value. When Reynolds number increases it, is observed that downstream cylinder results higher Nusselt number than the upstream due to the periodic nature of the fluid flow at the rear side of the cylinder. It is worth noting here that this trend is identical with Sohankar and Etminan [8] for their inline two tandem square cylinder for gap ratio equals to 5. In the present case both cylinders encounter asymmetry fluid hence the cylinder bottom face results in higher Nusselt number than the cylinder top face. Similar trend is observed by Rosales et al. [10] for their offset square cylinder placed in a confined channel channel. The average Nusselt number is reported in the figure 5(c). Up to steady state the average Nusselt increases linearly. Beyond critical Reynolds number due to the presence of rear side vortex and bottom wall flow separation results in a steep increment in the average Nusselt number for downstream cylinder.



(a) Local Nu: steady state Re = 20



(b) Local Nu: periodic case: Re = 80



(c) Average Nu

Figure 5. Local and average Nusselt number: time average values for periodic results

5. CONCLUSION

Numerical investigation is carried out for flow over two heated offset square cylinders placed in a confined channel. The steady and periodic results reported for laminar range of Reynolds number. The onset of vortex shedding is at $45 \leq Re \leq 50$. The downstream cylinder is more pronounced to the variation in Reynolds number beyond critical Reynolds number value. Identical pair of vortex is formed from the rear side of upstream cylinder A which is not formed from downstream of cylinder B. Beyond critical Reynolds number the increment in Re enhances heat transfer from downstream cylinder significantly. Isotherm contours are crowded at the front wall of the cylinders and they spread widen in the downstream. Heat transfer shows two dimensionality in the top and bottom solid wall near a few cylinder diameter distances in the downstream. The local Nusselt numbers from top and bottom wall are not analogous to a single cylinder undergoing heat transfer due to the asymmetry nature of fluid flow. The cylinder average Nusselt number increases when Re increases. The bottom wall Nusselt number is greater than the top wall Nusselt number beyond critical Re.

ACKNOWLEDGMENT

The first author gratefully acknowledged the grant received from DST, Govt of India, under Fast Track Scheme SR/FTP/ETA – 0015/2010.

REFERENCES

[1] Kanna, P.R., Das, M.K.: Conjugate heat transfer study of two-dimensional laminar incompressible offset jet flows. Numerical Heat Transfer: Part A. 48, 671–691, 2005.

[2] Eaton, B.E.: Analysis of laminar vortex shedding behind a circular cylinder by computer aided flow visualization. Journal of Fluid Mechanics. 180, 117–145, 1987

[3] Okajima, A.: Strouhal numbers of rectangular cylinders. Journal of Fluid Mechanics. 123, 379–398, 1982..

[4] Mukhopadhyay, A., et al.: Numerical investigation of confined wakes behind a square cylinder in a channel. International Journal for Numerical Methods in Fluids. 14, 1473–1484, 1992.

[5] Sohankar, A., Norberg, C., Davidson, L.: Numerical simulation of unsteady low-reynolds number flow around rectangular cylinders at incidence. Journal of Wind Engineering and Industrial Aerodynamics. 69-71,189–201, 1997.

[6] Sushanta Dutta, Panigrahi, P. K., Muralidhar, K.: Experimental investigation of flow past a square cylinder at an angle of incidence. Journal of Engineering Mechanics. 134, 788–803, 2008.

[7] Atul Sharma, Eswaran, V.: Heat and fluid flow across a square cylinder in the two-dimensional laminar flow regime. Numerical Heat Transfer: Part A. 45, 247–269, 2004.

[8] Sohankar, A., Etminan, A.: Forced-convection heat transfer from tandem square cylinders in cross flow

at low reynolds numbers. International Journal for Numerical Methods in Fluids. 60, 733–751, 2009.

[9] Dipankar Chatterjee, Bittagopal Mondal.: Forced convection heat transfer from tandem square cylinders for various spacing ratios. Numerical Heat Transfer: Part A. 61, 381–400, 2012.

[10] Rosales, J.L. , Ortega, A., Humphrey, J.A.C.: A numerical simulation of the convective heat transfer in confined channel flow past square cylinders: comparison of inline and offset tandem pairs. International Journal of Heat and Mass Transfer. 44, 587–603, 2001.

[11] Galletti, B., Bruneau, C. H., Zannetti, L., Iollo, D A.: Low-order modelling of laminar flow regimes past a confined square cylinder. Journal of Fluid Mechanics. 503, 161–170, 2004.

[12] Breuer, M., Bernsdorf, J., Zeiser, T., Durst, F.: Accurate computations of the laminar pow past a square cylinder based on two dierent methods: lattice-boltzmann and rnite-volume. International Journal of Heat and Fluid Flow. 21,186–196, 2000.

[13] Alvaro Valencia, Ronald Paredes.: Laminar flow and heat transfer in confined channel flow past square bars arranged side by side. Heat and Mass Transfer. 39,721–728, 2003.

[14] S. Sripattanapipat, P. Promvonge.: Numerical analysis of laminar heat transfer in a channel with diamond-shaped baffles. International Communica–tions in Heat and Mass Transfer 36, 32–38, 2009.

[15] Dhiman, A. K., Sharma, N., Kumar. S.: Wall effects on the cross-buoyancy around a square cylinder in the steady regime. Brazilian Journal of Chemical Engineering. 29,253–264, 2012.

[16] Allanaboyina, Durga Prasad, V. V. S., Amit K. Dhiman.: Cfd analysis of momentum and heat transfer around a pair of square cylinders in side-by-side arrangement. Heat Transfer Engineering, 35,398–411, 2014.

[17] Chiu, W. K.S, et al.: Experimental and numerical study of conjugate heat transfer in a horizontal channel heated from below. ASME Transactions Journal of Heat Transfer. 123, 688–697, 2001.

[18] Kanna, P.R. , Das, M.K.: Conjugate heat transfer study of backward-facing step flow a benchmark problem. International Journal of Heat and Mass Transfer. 49,3929–3941, 2006.

[19] Kanna, P.R. Das, M.K.: Effect of geometry on the conjugate heat transfer of wall jet flow over backward-facing step. Journal of Heat Transfer. 131,114501–1–7. 2009.

[20] Das, M.K. Kanna, P.R.: Application of an adi scheme for steady and periodic solutions in a liddriven cavity problem. International Journal of Numerical Methods for Heat & Fluid Flow, 17, 799 – 822, 2007.

[21] Mohammad Rahnama and Kakimeh Hadi-Mohaddam.: Numerical investigation of convective heat transfer in unsteady laminar flow over a square cylinder in a channel. Heat Transfer Engineering, 26, 21–29, 2005.

[22] Sohankar , A., Etminan, A.: Forced-convection heat transfer from tandem square cylinders in cross flow at low reynolds numbers. International Journal for Numerical Methods in Fluids. 60, 733-751, 2009.

NOMENCLATURE

d	side of square cylinder, m
f	shedding frequency, Hz
H	height of the channel, m
k	thermal conductivity *W/mK*
Nu	Nusselt number
Pr	Prandtl number $= \upsilon / \alpha$
Re	Reynolds number = $\bar{u}_0 d / v$
t	dimensional time, s
t	nondimensional time
$\bar{u}, \bar{v}$	dimensional velocity components along *x, y* axes, m/s
u, v	nondimensional velocity components along *x; y* axes
$\bar{u}_0$	average horizontal velocity at the channel inlet, m/s
$\bar{x}, \bar{y}$	dimensional Cartesian co-ordinates, m
x, y	nondimensional Cartesian co-ordinates

Greek symbols

∞	ambient condition
α	thermal diffusivity, m^2/s
β	blockage ratio, d/H
ε	convergence criteria
μ	dynamic viscosity, kg/m-s
υ	kinematic viscosity, m^2/s
ω	nondimensional vorticity
ψ	nondimensional stream function
θ	nondimensional temperature, $= \dfrac{T - T_\infty}{T_w - T_\infty}$

Superscripts

avg	average
f	fluid
n	normal direction
s	solid
w	wall

НУМЕРИЧКО ИСТРАЖИВАЊЕ ПРЕНОСА ТОПЛОТЕ ПРИНУДНИМ СТРУЈАЊЕМ СА ЧЕТВРТАСТИХ ЦИЛИНДАРА У КАНАЛУ ПОКРИВЕНОМ ЧВРСТИМ ЗИДОМ – КОМБИНОВАНО СТАЊЕ

П. Рајеш Кана, М. Сивасубраманиан, М. Улхајакумар

Извршена је симулација струјања преко два изотермна размакнута четвртаста цилиндра у затвореном каналу за различите вредности Рејнолдсовог броја да би се одредио пренос топлоте принудним струјањем са четвртастих цилиндара на амбијентални флуид код фиксног процента запречавања. Канал је покривен чврстим зидом и разматра се комбиновани пренос топлоте. Пренос

топлоте од цилиндара до амбијенталног флуида као и провођење кроз чврсти зид преко комбинованог граничног интерфејса истражује се код стационарног и нестационарног струјања. Симулација варира од 10 до 100 за Рејнолдсов број а за Прандтлов број износи 0,71. Почетак вртложног кретања настаје када вредност Рејнолдсовог броја износи 48. Температура комбинованог интерфејса опада са порастом Рејнолдсовог броја. Изотерма код чврстог зида показује дводимензионалност у близини цилиндра. Код узводног струјања долази до пораста Нуселтовог броја при стационарном стању, док код низводног струјања Нуселтов број опада при нестационарном стању.