Procedings of COBEM 2007
Copyright © 2007 by ABCM

19th International Congress of Mechanical Engineering
November 5-9, 2007, Brasília, DF

NUMERICAL SIMULATION OF FLOWS OVER CYLINDERS UNDER THE INFLUENCE OF BUOYANCY USING THE IMMERSED BOUNDARY METHOD

Ana Lúcia Fernandes de Lima e Silva, alfsilva@unifei.edu.br
Sandro Metrevelle Marcondes de Lima e Silva, metrevel@unifei.edu.br
Federal University of Itajubá - UNIFEI, Institute of Mechanical Engineering - IEM, Prof. José Rodrigues Seabra Campus, 1303 BPS Av., Pinheirinho District, ZIP code 37500-903, Box-Postal 50, Itajubá, Minas Gerais, Brazil.

Abstract. *In the present work the numerical simulation of two-dimensional laminar flows inside a plane channel, over a isothermal square cylinders is carried out using the Immersed Boundary method with the Virtual Physical Model. A correction pressure method with central differences for the spatial discretization and the second-order Runge-Kutta time integration scheme are employed. The effect of adding/opposing buoyancy for Richardson number of* -1 *to* 1 *and the channel-confinement are studied for Prandtl number of 0.71 and Reynolds* $= 100$. *The Strouhal* (St) *and Nusselt* (Nu) *numbers and the mean drag* (C_d) *coefficient are obtained as a function of the Richardson number (Ri) and the blocked ratio (B/H %). The Kármán vortex street is also presented for different conditions of buoyancy and channel confinement conditions.*

Keywords: *immersed boundary method, finite difference, square cylinder, blockage ratio, buoyancy effect*

1. INTRODUCTION

The heat transfer process is important in many types of external or confined flows. The presence of an immersed body can generate the formation of vortices and the vortex shedding, that can influence the flow dynamics. All the parameters which influence the structure of the Von Kármán street are been studied. Both phenomenons are present, for example, in the anemometry field, where the sensor in a constant temperature can influence the flow dynamics and the quality results. With the objective of minimizing this influence, the project of these instruments is done with the previously known of the heat transfer and aerodynamic coeficientes. In many other applications there is interest in this class of problems, as the heat exchangers, the risers of offshore structures and the cooling of equipments. Several experimental, numerical e theoretical works have been realized.

Concerning about the numerical works, there are many studies presented in the literature about isothermal flows with cylinders and this is a benchmark problem to study the dynamic of this flows. Some authors use classical methodologies with the objective of analyzing the results and better to understand the flow. Other studies are done to propose new numerical methodologies, to improve the quality, the efficiency and the velocity in the results obtaining. Some classical works can be mentioned as Williamson (1996) that presents a revision from the literature about flows over circular cylinders, Saiki and Biringen (1996) used the Virtual Boundary Method to simulate staggered and rotating cylinders, Meneghini and Bearman (1997) and Meneghini et al. (2001) presented the results of the simulations with oscillating cylinder, cylinders side by side and in Tandem. Actually Sumner et al. (2005) realized experiments to determine the aerodynamic forces and the vortex shedding frequency of two cylinders of same diameter diagonally disposed, for Reynolds number of 3.2 x 10^4 a 7.4 x 10^4.

Flows with heat transfer are being very studied due to the great occurrence in real situations. Patnaik et al. (1999) simulated flows over a circular cylinder under the buoyancy effect in the downstream and upstream directions. The authors showed the aerodynamic coefficients, the vortex shedding frequency and the Nusselt number as a function of Reynolds and Richardson. In their work, Patnaik et al. (2000), the authors analyzed flows over a single circular cylinder and cylinders in Tandem, for Richardson numbers of -1 to 1. They showed results of Strouhal (St), drag coefficient (C_d) and root mean square of lift coefficient (C_{lrms}) as function of Ri and the vorticity and temperature fields. There are many other works involving mixed and forced convection with cylinders as in Sharma and Eswaran (2005a) and (2005b), that analyzed the confinement effect of the flow over square cylinders and the heat transfer. Studies based on the Immersed Boundary Method with heat transfer are also been presented in literature, because the advantages of the methodologies in relation to the use of Cartesian grids to simulate any kind of geometric form. It can be mentioned the studies of Giacomello et al. (2000), Pacheco et al. (2005) and Pacheco-Vega (2006).

For the present work an analysis of mixed and forced convection of an isothermal square cylinders is presented using the Immersed Boundary Method with the Physical Virtual Model, IBM/PVM, (Lima e Silva, 2002). The code was developed by the authors with the main objective of to study the effect of aiding and opposing buoyancy, as well as the the effect of the proximity channel walls on the flow and the heat transfer across a heated square cylinder using the proposed methodology IBM/VPM. The great advantage of this methodology is to study stagnant or moving bodies of any geometrical shape, as presented by the authors for example in Oliveira et al., 2006. For the present work the heating of the immersed body is considered and the results of square cylinder are compared with other numerical results from the literature. The effect of adding/opposing buoyancy is considered by the Richardson number and the results of drag coefficient, Nusselt and Strouhal numbers and the wake structure are presented. This study also attempts to understand the influence of the parameters (Ri and B/H) on the Kármán vortex street for the square cylinder. The governing equations used to obtain the discretized formulation are briefly described in next section.

2. GOVERNING EQUATIONS, BOUNDARY CONDITIONS AND NUMERICAL APPROACH

In the Immersed Boundary Method a mixed Eulerian-Lagrangian formulation is used to represent the flow and the immersed boundary. At the present work a Cartesian grid describes the flow using a Finite Difference method and a Lagrangian grid, composed by a finite number of points, describes the immersed body. The Eulerian and the Lagrangian grids are coupled by a force field calculated at the Lagrangian points and then distributed across the Eulerian nodes in the body neighborhood. The two-dimensional, time dependent Navier Stokes, continuity and energy equations are solved for an incompressible, Newtonian, viscous flow. The flow is assumed to be Boussinesq approximated. The non-dimensional form of theses equations can be written as:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{1}$$

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{\partial p}{\partial x} + \frac{1}{Re}\nabla^2 u + Ri\theta + f_x \tag{2}$$

$$\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} = -\frac{\partial p}{\partial y} + \frac{1}{Re}\nabla^2 v + f_y \tag{3}$$

$$\frac{\partial \theta}{\partial t} + u\frac{\partial \theta}{\partial x} + v\frac{\partial \theta}{\partial y} = \frac{1}{Re.Pr}\nabla^2 \theta + q \tag{4}$$

where u and v are the velocity components, p is the pressure, f_x and f_y are the Eulerian force components and $\theta = (T - T_o)/(T_c - T_o)$. T_c and T_o are the temperatures of the cylinder and at the inlet, respectively.

The Immersed Boundary Method is used to model the cylinder inside the channel. The boundary conditions of velocity and temperature are imposed through the force $\mathbf{f} = (f_x, f_y)$ and energy (q) terms added to the governing equations. These terms have the function of to model the immersed body with the :

$$\mathbf{f}(\mathbf{x}, t) = \sum \mathbf{F}(\mathbf{x}_k, t) D(\mathbf{x}_k)(\mathbf{x} - \mathbf{x}_k)\Delta s^2, \tag{5}$$

$$q(\mathbf{x}, t) = \sum Q(\mathbf{x}_k, t) D(\mathbf{x}_k)(\mathbf{x} - \mathbf{x}_k)\Delta s^2, \tag{6}$$

where **F** and Q are calculated through the temperature, velocity and pressure values, interpolated from the Eulerian grid. D is the distribution function suggested by Juric (1996) and Δs is the distance between two lagrangian points. The Virtual Physical Model proposed by Lima e Silva (2202) is used to obtain the Lagrangian quantities, over the immersed boundary, through the following equations:

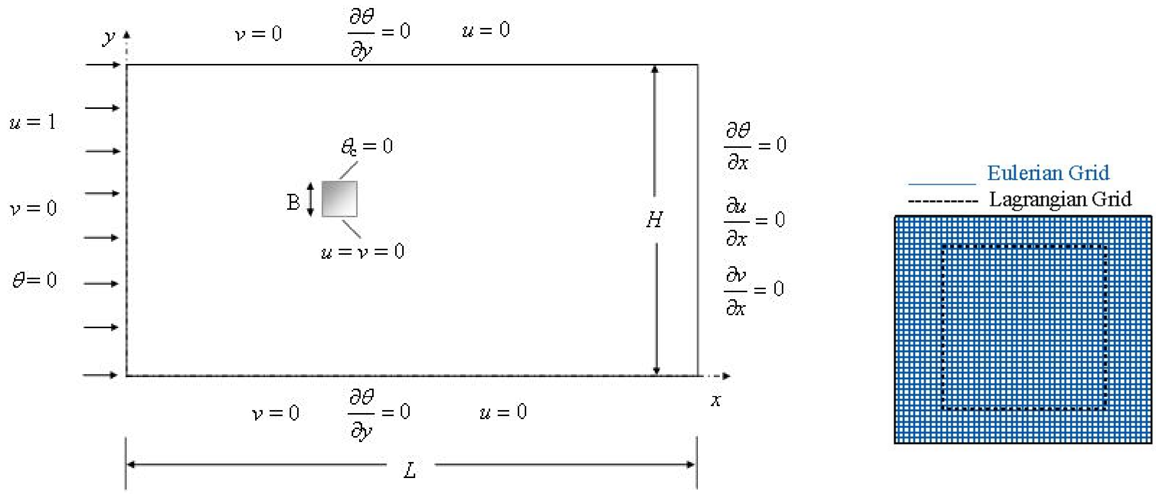
$$\mathbf{F}(\mathbf{x}_k, t) = \frac{\partial \mathbf{V}}{\partial t} + (\mathbf{V}.\nabla)\mathbf{V} - \frac{1}{Re}\nabla^2 \mathbf{V} + \nabla p, \tag{7}$$

$$Q(\mathbf{x}_k, t) = \frac{\partial \theta}{\partial t} + \mathbf{V}.\nabla\theta - \frac{1}{RePr}\nabla^2\theta. \tag{8}$$

All terms of Eqs. (7) and (8) are obtained on the Lagrangian grid and then are used in Eqs. (5) and (6) to get the Eulerian variables.

Equations are discretized with a second order finite differences in space and the second order Runge-Kutta method for time. The coupling between pressure and velocity is done with a fractional step method based on a pressure correction scheme. More details can be seen in Lima e Silva (2002) and in Lima e Silva et. al (2003).

The physical model used in this study with the boundary conditions are shown in Fig. 1(a). A two-dimensional channel with height H and length L is used to simulate this problem. An isothermal square cylinder of B side with constant temperature is set within the channel. The distance of the cylinder from the inlet is $16.5B$ and $L = 55.0$ for all studied cases. These values were chosen in preliminary studies. At the inlet, the velocity and temperature of the fluid are uniform and the flow develops from left to right. The gravity acts through the Richardson number in the flow direction (positive values of Richardson) and opposite to the flow (negative values of Richardson). The no-slip boundary conditions on the cylinder and the constant temperature ($u = v = 0$ and $\Theta_c = 0$) are not directly imposed. They are set through the



(a) (b)

Figure 1. Physical model with boundary conditions (a) and closer view of the cylinder with both grids (b).

force $\mathbf{f} = (f_x, f_y)$ and energy (q) terms added to the governing equations as explained previously. The channel walls are adiabatic for all simulations.

A non-uniform Cartesian Eulerian grid was used and the number of points is varied in order to guarantee 40 points inside the cylinder for all simulations. This value was set after refinement studies from previous works. The grid is uniform in a square region evolving the cylinder, as shown if Fig. 1(b).

3. RESULTS

The simulations were carried out for different values of the blockage ratio ($B/H = 4, 10, 20, 30, 50\%$) and Richardson numbers ($Ri = Gr/Re^2$) of -1.0 to 1.0, where $Gr = g\beta(T_c - T_o)B^3/\nu^2$ is the Grashof number and Re is based on the square width B. The results for Reynolds 100 and a Prandtl number of 0.71 are presented in this paper in order to be compared with other numerical results from the literature.

3.1 Flow aspects

Figures 2, 3 and 4 show the instantaneous vorticity contours for different values of Ri and for $B/H = 10$, 30 and 50%, respectively. Solid and dashed lines represent positive and negative values of voticity, respectively and the legend presented in Fig. 2 is also valid for Figs. 3 and 4. These figures are presented in the vertical position just for simplicity but the flow develops from the left to the right as shown in Fig. 1.

The sinuous motion of the vortex shedding is suppressed in all cases for $Ri = 0.5$, except for $B/H = 20\%$ (not shown here) and 30% where the suppression occurred for $Ri = 0.25$. The oscillation motion increases with cooling ($Ri < 0$) and decreases with heating ($Ri > 0$). The vortex shedding ceases approximately at $Ri = 0.5$, $Ri = 0.25$ and $Ri = 0.5$ for $B/H = 10$, 30 and 50%, respectively. The values presented by Sharma and Eswaran (2005) were $Ri = 0.4$, $Ri = 0.25$ and $Ri = 0.5$ for the same blockage ratios which demonstrate the same flow behavior. It is known that for this Reynolds ($Re = 100$) the vortex shedding is also presented for the cases where buoyancy is absence, however for $Ri > 0$ vortex shedding can be suppressed for some blocked ratios. For lower Reynolds, the pair of vortices behind the cylinder can totally vanish when the flow and buoyancy are in the same direction. It is also observed that the structure of the wake and the size of the vortices are dependent on the Richardson number. Chang and Sa (1990) also studied the effect of buoyancy with circular cylinders. These authors mentioned that the strength of of shear layer increases and the roll-up process is more activated increasing cooling because the velocity in the wake region is reduced by the negative buoyancy force of the cold fluid. It was also observed here with the square cylinder. In opposite, heating gives stability to the vortex street with an increase in the velocity in the wake region. Increasing the blockage ratio, the streamlines near to the cylinder are confined and the velocity in the wake is also increased. As the blockage ratio increases less heating is required to stabilize the flow but this behavior has a limit. This limit was observed looking at Figs. 3 and 4. More heating was necessary to stabilize the flow in $B/H = 50\%$ ($Ri = 0.5$) than in $B/H = 30\%$ ($Ri = 0.25$) because the interaction between the fluid from the cylinder surfaces and from the channel walls increases and probably contributes with the breakdown of the vortex street. Sharma and Eswaran (2005a) also discussed the same phenomenon.

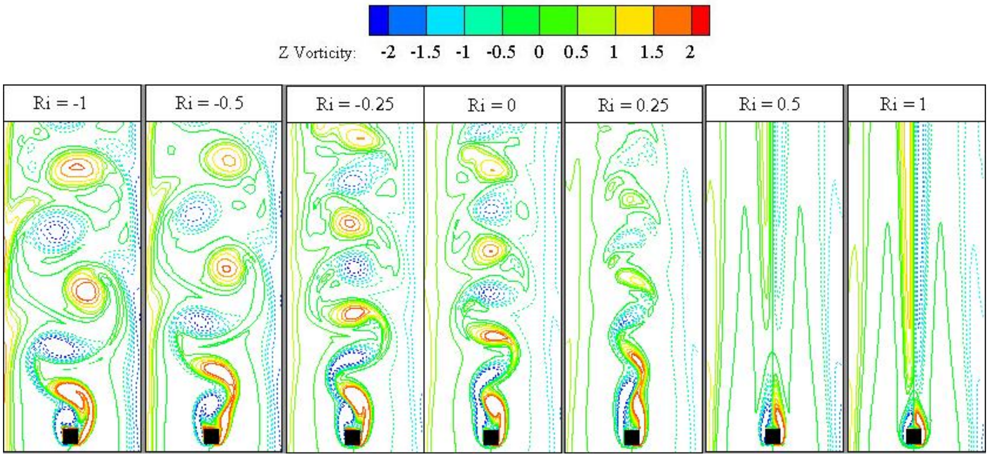


Figure 2. Vorticity contours under the influence of buoyancy for $B/H = 10\%$.

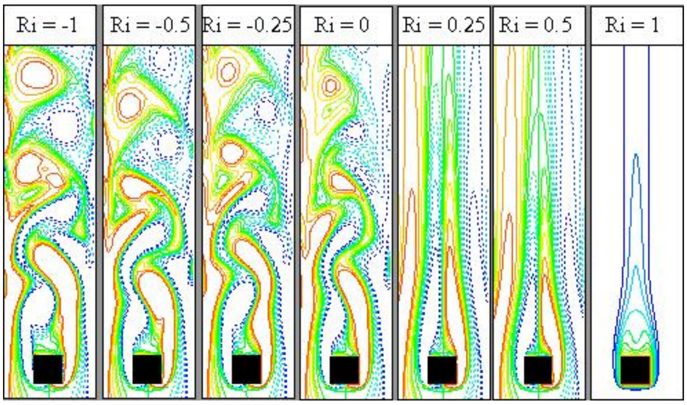


Figure 3. Vorticity contours under the influence of buoyancy for $B/H = 30\%$.

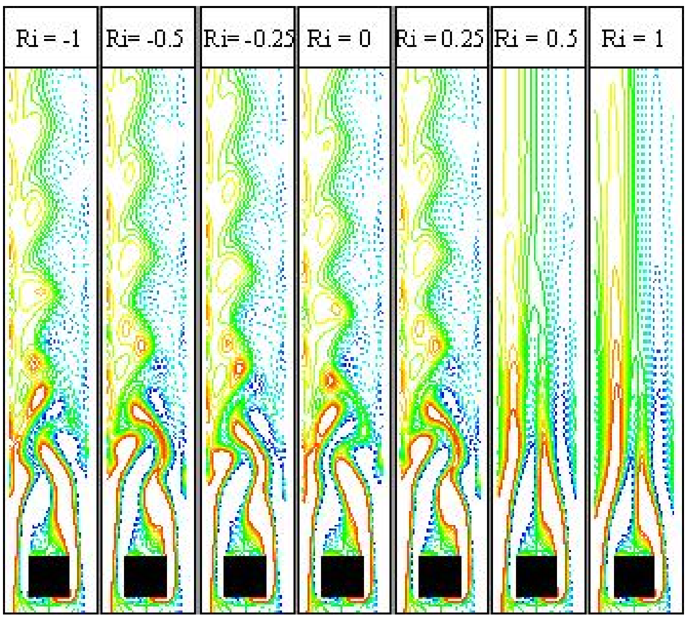


Figure 4. Vorticity contours under the influence of buoyancy for $B/H = 50\%$.

3.2 Drag Coefficient and Strouhal Number

In Figure 5 it is presented the drag coefficient variation with the non-dimensional time for different blockage ratios. The drag increases with the blockage ratio, as expected, for all Ri numbers. The amplitude of oscillation increases with the buoyancy opposite to the flow ($Ri < 0$).

Table 1 presents the mean drag coefficients obtained from an average over many cycles of vortex shedding for a channel-confinement of $B/H = 30\%$. The results are very well compared with the results of Sharma and Eswaran (2005a).

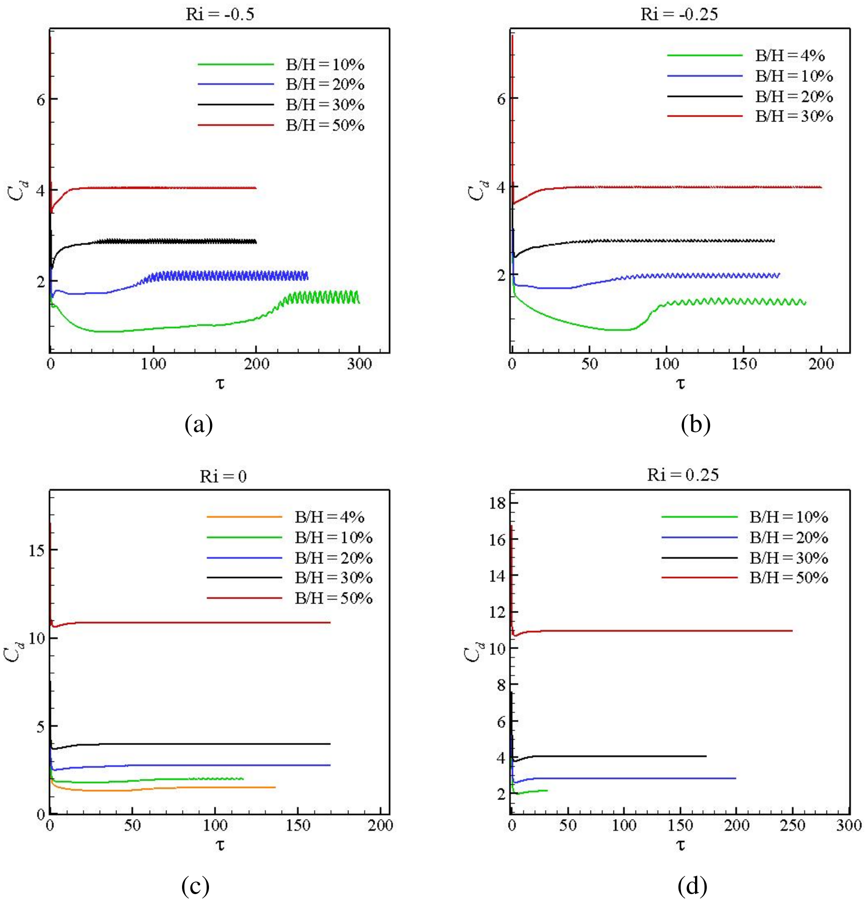


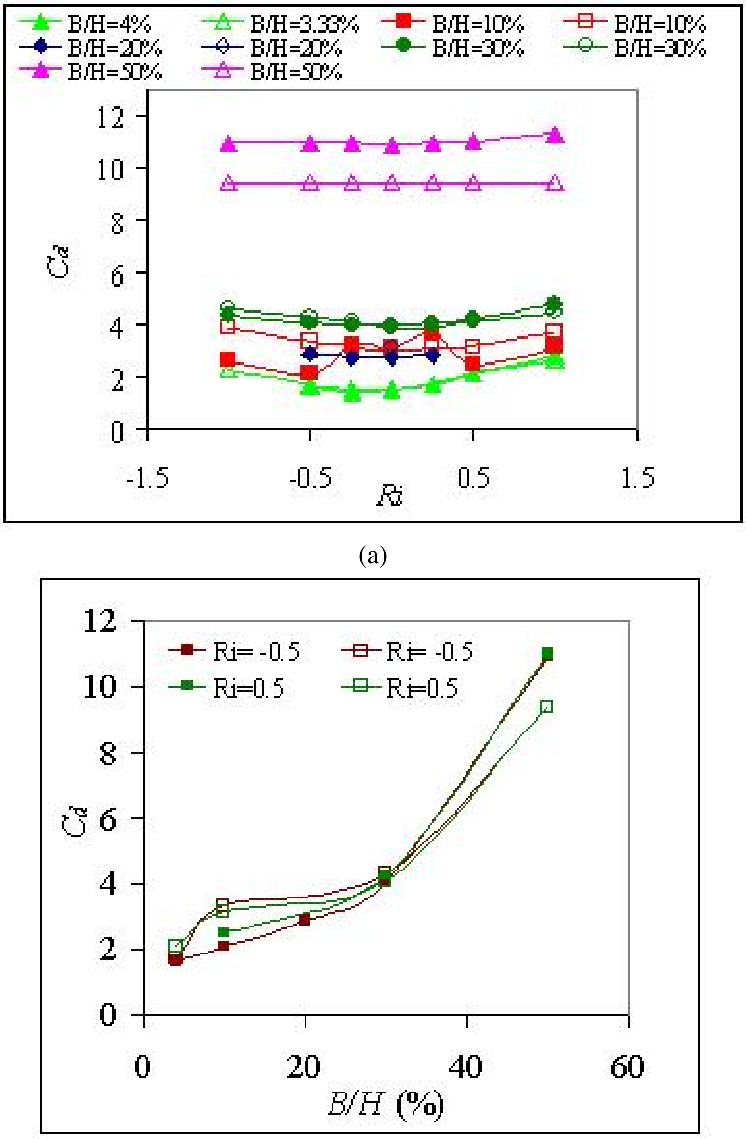
Figure 5. Temporal variation of Drag Coefficient for different values of $B/H\%$ under the influence of buoyancy. $Ri = -0.5$ (a), $Ri = -0.25$ (b), $Ri = 0$ (c) and $Ri = 0.25$ (d).

Table 1. Present results for the mean drag coefficient under the influence of buoyancy for $B/H = 30\%$ and $Re = 100$.

Ri	-1	-0.5	-0.25	0	0.25	0.5	1
Author	C_d						
Present results	4.37	4.05	3.98	3.98	4.05	4.26	4.77
Sharma and Eswaran (2005a)	4.64	4.3	4.1	3.9	3.9	4.1	4.45

Another way to analyze these results is presented in Fig. 6 where the drag coefficient is shown as a function of the Ri and $B/H(\%)$. The results are presented for different blockage ratios and compared with the results of Sharma and Eswaran (2005b). The present results are little higher than the literature results only for $B/H = 50\%$. For $B/H = 10\%$ and $0.5 \leq Ri \leq -0.5$ the results were smaller and for this blocked ratio the results presented a weird behavior that must be better investigated. For the other blockage ratios the results were well compared. The authors attribute this behavior to the use of different methodologies and grids. The drag increases with Ri and B/H and presents a minimum at $Ri = 0$ in $B/H = 10\%$. As the blocked ratio increases, the increase in C_d is more pronounced ($B/H > 20\%$).

The non-dimensional vortex shedding frequency or Strouhal was obtained from the Fast Fourier Transform of the lift force signal. Strouhal results as a function of Ri are shown in Fig. 7(a) where it is possible to see a good accordance

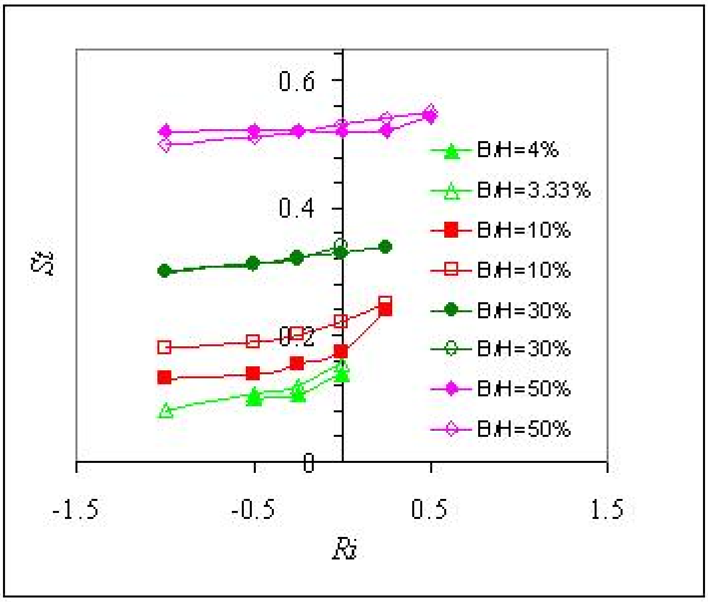


(a)

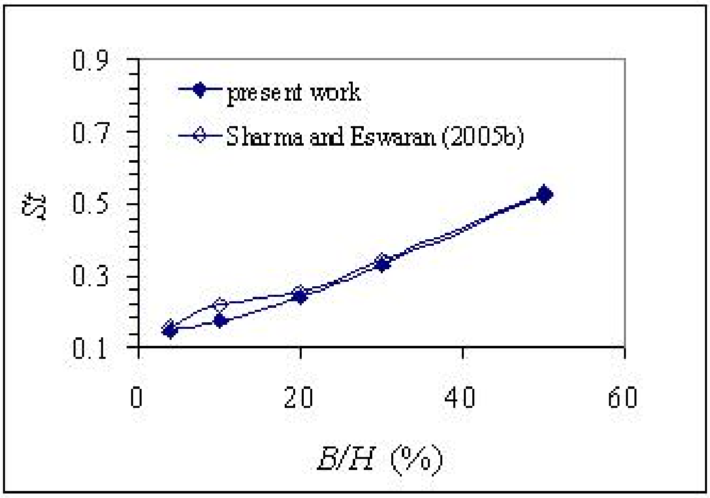
(b)

Figure 6. Drag coefficient as function of Ri (a) and as function of $B/H(\%)$ (b). Filled symbols correspond to the present results and open symbols to the Sharma and Eswaran (2005a) results.

with the literature results. For $B/H = 10\%$ the difference with the Sharma and Eswaran (2005a) results was higher. The Strouhal increases with Ri, however beyond a certain range of Richardson numbers, only the presence of twin vortices are seen instead of a vortex street. This process was called of "breakdown of Kàrmàn vortex street" by Noto and Matsumoto (1991). The heating ($0 < Ri < 1$) increases the velocity in the wake region and causes the shear layer to be weakened and it results in a faster vortex shedding while with the cooling ($-1 < Ri < 0$) the strength of the shear layer increases, the roll-up process is more activated and the velocity in the wake region is reduced. Hence the separation takes place earlier. Figure 7(b) shows that St increases with the blocked ratio for the forced convection case ($Ri = 0$) due to the acceleration of the cylinder-wall boundary layer and this increase is more accentuated for $B/H > 20$.



(a)



(b)

Figure 7. Strouhal number as function of Ri for different values of $B/H\%$(a) and as function of $B/H\%$ for $Ri = 0$ (forced convection case) (b). Filled symbols correspond to present results and open symbols to the Sharma and Eswaran (2005a) and (2005b) results.

3.3 Temperature Distribution and Nusselt Number

Figure 8 shows the instantaneous isotherms for $Ri = 0$ and the results of Sharma e Eswaran (2005b). There is a great similarity on the positions and magnitude of the isotherms. It was also noted that for small blocked ratio the contours of value 0.1 envelops the cylinder, but is opened in the channel walls direction for $B/H = 50\%$. For intermediate blocked ratios there is a single closed contour row of 0.1, however for $B/H = 20\%$ and $B/H = 30\%$ there are two rows in a staggered arrangement. This same behavior was observed by Sharma e Eswaran (2005b).

The local Nusselt is calculated for each Lagrangian point by $-\frac{\partial \theta}{\partial n}$, where n is the direction normal to the the cylinder surface. The average Nusselt is obtained by averaging the local Nusselt over all the surfaces. The final value is time-averaged over at least the last 20 cycles of vortex shedding. In the following figure, (Fig. 9), the mean Nu is presented for different values of B/H, for all simulated Ri numbers. It is clear that Nu increase with B/H, but is almost constant between $B/H = 10\%$ and $B/H = 20\%$. The interaction of the cold fluid next to the channel walls and the hot fluid next to the cylinder increases with the blocked ratio, resulting in a higher value of the Nusselt number.

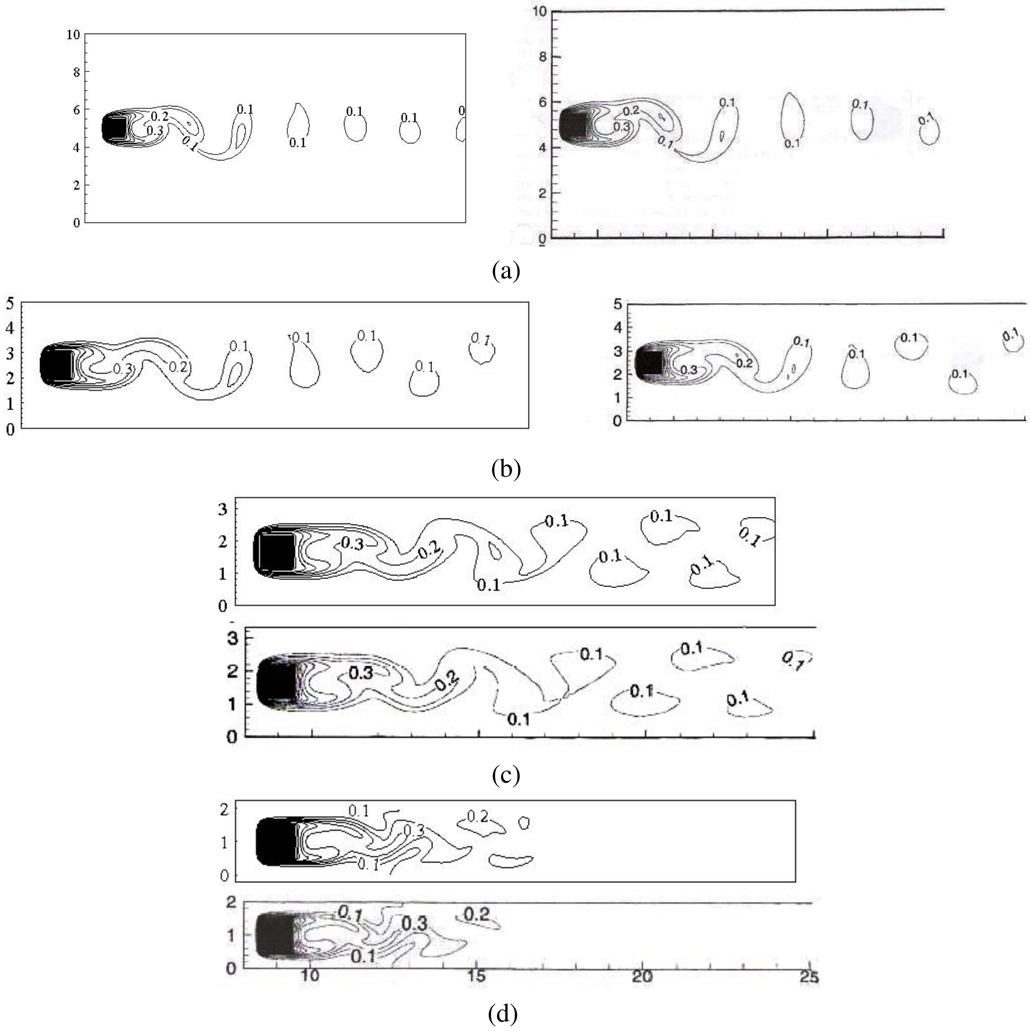


Figure 8. Instantaneous Isotherms for blocked ratios of: $B/H = 10\%$ (a), $B/H = 20\%$ (b), $B/H = 30\%$ (c) and $B/H = 50\%$ (d). Top Figure is of the present work and the bottom figure is the Sharma and Eswaran (2005a) and (2005b) results.

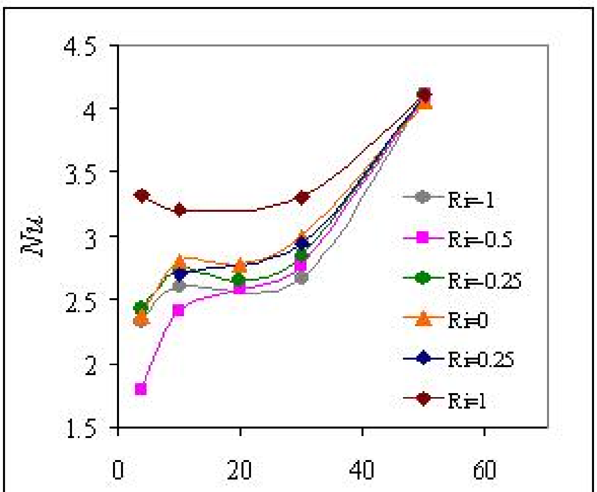


Figure 9. Mean value of the Nusselt as a function of $B/H\%$ for all simulated Ri.

4. CONCLUSIONS

The Virtual physical Model was used in the present work to indirectly impose the boundary conditions of no-slip and constant temperature on an immersed square cylinder. The simulations were realized for a square cylinder due to the availability of results on the literature but with the Immersed Boundary Method is possible to simulate any geometric shape with no change of the Eulerian grid generation. That is the great advantage of the methodology. The influence of the buoyancy, adding ($0 < Ri < 1$) or opposing ($-1 < Ri < 0$) to the flow direction, was analyzed through the drag, Strouhal, Nusselt and structure of the wake for different blocked ratios of the channel walls ($B/H = 4, 10, 20, 30$ and 50%). At the simulated Reynolds ($Re = 100$) it was observed that up to $B/H = 30\%$ the minimum heating, required for the vortex shedding suppression, decreases with the increase of the blocked ratio. This behavior was observed up to $B/H = 30\%$. For $B/H = 50\%$ the critical Ri increases. The drag coefficient increases with the blockage ratio and at the same blocked ratio, there is a minimum value at $Ri = 0$ for $B/H = 4\%$, $B/H = 20\%$ and $B/H = 30\%$. For $B/H = 10\%$ the curve oscillates and for $B/H = 50\%$, C_d does not change significantly with Ri. The Strouhal and the Nusselt numbers increase with Ri and B/H. The present results demonstrate the capability of the methodology, Immersed Boundary method with Virtual Physical Model, to simulate flows with heat transfer. It must be also emphasized that any other geometry of the immersed body should be easily simulated with this methodology.

5. ACKNOWLEDGEMENTS

The authors would like to thank FAPEMIG for the financial support (TEC 0084/06), CNpq and the Institute of Mechanical Engineering of the Federal University of Itajubá.

6. REFERENCES

Chang, K.S. and Sa, J.Y., 1990, "The Efect of Buoyancy on Vortex Shedding in the Near Wake of a Circular Cylinder", Journal of Fluid Mechanics, Vol. 220, pp. 253-266.

Giacomello, M.V., Rocha, L.A.O., Schettini, E.B.C. and Silvestrini, J.H., 2000, "Estudos Numéricos de Escoamentos ao Redor de Três Cilindros com Transferência de Calor", Proceedings of the 11th Brazilian Congress of Thermal Sciences and Engineering, pp. 1-12.

Juric, D., 1996, "Computational of Phase Change", Ph. D. Thesis, Mechanical Engineering University of Michigan, USA.

Lima e Silva, A.L.F., 2002, "Desenvolvimento e Implementação de Um Nova metodolgia para Modelagem de Escoamentos sobre geometrias complexas: Método da Fronteira Imersa com o Modelo Físico Virtual", Tese de doutorado, Universidade Federal de Uberlândia.

Lima e Silva, A.L.F., Silveira Neto, A. and Damasceno, J.J.R., 2003, "Numerical Simulation of Two Dimensional Flows over a Circular Cylinder using the Immersed Boundary Method", Journal of Computational Physics, Vol. 189, pp. 351-370.

Meneghini, J.R. and Bearman, P.W.,1997, "An Investigation of the Effect on the Vortex Shedding of a Sudden Transverse Disturbance Applied to a Circular Cylinder", Journal of Wind and Engineering and Industrial Aerodynamics, Vols. 67-71, pp. 229-238.

Meneghini, J.R., Saltara, F., Siqueira, C.L.R. and Ferrari Jr., J.A.,2001, "Numerical Simulation of Flow Interference Between Two Circular Cylinders in Tandem and Side-by-Side Arrangements", Journal of Fluids and Structures, Vol.15, pp. 327-350.

Noto, K. and Matsumoto, M.,1991, "Generation and Suppresion of the Kàrman Vortex Street unpon Controlling Surface Temperat of a Cylinder" , Proceedings of Numerical Methods Laminar and Turbulent Flows, Vol. 7, pp. 671-678.

Oliveira, J.E.S., Lima e Silva, A.L.F. and Silveira Neto, A., 2006, "Simulação Numérica de Escoamentos sobre Aerofólios Móveis usando o Método de Fronteira Imersa", Proceedings of the 11th Brazilian Congress of Thermal Sciences and Engineering- ENCIT 2006.

Pacheco, J.R., Pacheco-Vega, A., Rodic, T. and Peck, R.E.,2005, "Numerical Simulation of Heat Transfer and Fluid Flow Problems using an Immersed-Boundary Finite-Volume Method on Nonstaggered Grids" , Numerical Heat Transfer B , Vol. 48, pp. 1-24.

Pacheco-Vega, A., Pacheco, J.R., Rodic, T. and Peck, R.E., 2006, "A General Scheme for the Boundary Conditions in Convective and Diffusive Heat Transfer with Immersed Boundary Methods", submited to the Journal of Heat Transfer.

Patnaik, B.S.V., Narayana, P.A.A. and Seetharamu, K.N., 1999, "Numerical Simulation of Vortex Shedding Past a Circular Cylinder under the Influence of Buoyancy", International Journal of Heat and Mass Transfer,Vol.42, pp. 3495-3507.

Patnaik, B.S.V., Narayana, P.A.A. and Seetharamu, K.N., 2000, "Finite Element Simulation of Transient Laminar Flow Past Circular Cylinder and Two Cylinders in Tandem", International Journal of Numerical Methods for Heat and Fluid Flow, Vol. 10, pp. 560-580.

Saiki, E.M. and Biringen S.,1996, "Numerical Simulation of a Cylinder in Uniform Flow: Application of a Virtual Boundary Method", Journal of Computational Physics, Vol.123, pp. 450-465.

Sharma, A. and Eswaran, V., 2005a, "Effect of Channel-confinement and Adding/opposing Buoyancy on the Two-dimensional Laminar Flow and Heat Transfer Across a Square Cylinder", International Journal of Heat and Mass Tranfer, Vol. 48, pp. 5310-5322.

Sharma, A. and Eswaran, V., 2005b, "Effect of Channel-confinement on the Two-dimensional Laminar Flow and Heat Transfer Across a Square Cylinder", Numerical Heat Transfer, Vol. 47, pp. 79-107.

Sumner, D., Richards, M.D. and Akosile, O.O.,2005, "Two Staggered Circular Cylinders of Equal Diameter in Cross-Flow", Journal of Fluids and Structures, Vol.20, pp. 255-276.

Williamson, C.H.K., 1996, "Vortex Dynamics in the Cylinder Wake", Annual Reviews of fluid mechanics, Vol.28, pp. 447-539.

7. RESPONSIBILITY NOTICE

The author(s) is (are) the only responsible for the printed material included in this paper