CONVECTION HEAT TRANSFER AROUND A BANK OF TUBES

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. *The convection heat transfer around circular cylinders has been studied to be applied in heat exchangers, heaters, electronic circuits and other thermal equipment. In these applications, it is necessary to analyze the influence of the cylinders in the fluid flow and vice versa, on the heat transfer phenomenon. This paper presents a study about the Newtonian incompressible two-dimensional flow on isothermal cylinders using the Immersed Boundary Method (IBM) with the Physical Virtual Model (PVM). The heat transfer effect in the flow around a group of three cylinders was studied in forced and mixed convection. Different numbers of cylinders were also arranged in a single row configuration with different distances between them. Navier-Stokes and energy equations were discretized using the Finite Difference method for space and a second order Runge-Kutta method for time, and solved in a two-dimensional domain. Streamlines, isotherms, drag and lift coefficients and Nusselt number are presented. The results showed a great influence of the distances among the cylinders over the aerodynamic coefficients and over the Nusselt number.*

Keywords: *numerical simulation, convection, immersed boundary method, bank of tubes*

1. INTRODUCTION

The heat transfer phenomenon is presented in several kinds of flows like external or confined flows. Besides, an immersed body can generate the formation and detachment of vortices, changing the flow dynamics. The classical Von Kàrmam street has been extensively studied. These phenomena are simultaneously present, for example, in the anemometry field, where the sensor can affect the flow and interfere in the results. To minimize this influence, the project of this instruments is done with the previous knowledge of the aerodynamics and heat transfer coefficients. This class of flows is also present in heat exchangers, risers of petroleum, cooling of electronics components and others. For this reason many experimental, numerical and theoretical studies have been accomplished.

A vast number of numerical work may be found in literature for the benchmark problem of isothermal flows with a single cylinder. Some papers are based on classical methodologies and have the main objective of generating data to better understand the flow dynamics. Others proposed new numerical methodologies with the aim of improving the quality of the results, the efficiency and the speed to obtain the results. Some classical papers can be cited like Williamson (1996) who presented a bibliographical revision about flows around circular cylinders, Saiki and Biringen (1996) used the Immersed Boundary Method to simulate steady and rotating cylinders. Meneghini and Bearman (1997) and Meneghini *et al.* (2001) showed the results of the simulations with oscillating and side by side cylinders and the configuration in Tandem. Recently, Sumner *et al.* (2005) presented experimental results of the aerodynamics forces and vortex shedding frequency for two circular cylinders with equal diameter disposed diagonally, for Reynolds numbers of 3.2 x 10^4 a 7.4 x 10^4. Due to its great practical applications, this class of flows is still being studied.

Some papers deal with the flow around a bank of geometries like cylinders. Cylinder-like structures can be found both alone and in groups, in heat exchangers, offshore structures, cables and others. Ishigai and Nishikawa (1975) presented an experiment for a single array of cylinder with different gap sizes between the cylinders. The authors observed two types of flows. In the first, the vortex shedding from two adjacent cylinders is 180^o out of phase and in the second behavior, the cylinders near the smaller gap from the wall shed vortices in phase whereas cylinders farther from this side shed vortices out of phase. Huang *et al.* (2006) used the commercial code Fluent to study flows around a bank of cylinders in order to understand the dynamics of the forming section of a paper machine. The authors simulated laminar flow around two rows of unequal-sized cylinders at low Reynolds number and presented the vorticity field and velocity profiles.

When the buoyancy effect is present, the vortex shedding becomes more complicated. Patnaik *et al.* (1999b) simulated flows over single cylinders with the buoyancy effect. The aerodynamic coefficients, the vortex shedding frequency and the Mean Nusselt number as function of Reynolds and Richardson were presented. Also in the work of Patnaik *et al.* (1999a) the authors reported their results of flows with single cylinders and cylinders in tandem, for Richardson numbers from -1 to 1. They presented Strouhal number, drag and lift as function of Richardson number and vorticity and temperature fields. Many other papers concerning forced and mixed convection with cylinders have been published (Sharma and Eswaran, 2005a,b). Some studies based on the Immersed Boundary method with Heat Transfer have been outstanding in literature, since they take advantage of the Cartesian grid generation to simulate complex geometries (Giacomello *et al.*, 2006; Pacheco *et al.*, 2005; Pacheco-Vega *et al.*, 2006).

In the present work, the numerical investigations were carried out to simulate flows over three heated cylinders to analyze the effect of flow interaction among them. The cylinders were arranged in an triangular configuration in a horizontal channel. The flow is assumed to be two-dimensional and laminar (Re = 200) since the same configuration can exists in very compact heat exchangers. The simulations were carried out in forced and mixed convection in order to compare the effect of buoyancy which is orthogonal to the direction of the fluid. The vertical and horizontal distances among the cylinders were varied and the influence on the flow dynamics was analyzed. Drag coefficients and Nusselt numbers were computed from the velocity, pressure and temperature fields. A single row simulation was done with the objective of analyzing the influence of the gap between the cylinders and their distances to the walls. The purpose of this study was to clarify the heat transfer and flow characteristics around a single row of cylinders. This configuration occurs, for example, in a forming section of a paper machine where the dynamic of the flow upstream must be well known. These studies also contributed with the validation of the PVM applied for the analysis of the convection heat transfer phenomenon.

2. METHODOLOGY

Navier-Stokes equations for non-isothermal fluids and the equations for the Immersed Boundary Method with the Physical Virtual Model (PVM) are written here considering the following assumptions:

- the fluid is Newtonian and the flow is incompressible;
- the immersed body (i.e. the cylinder) is described by the PVM instead of by the direct imposition of the boundary conditions of velocity and temperature;
- the effect of the density variation due to temperatures changes is considered only in the gravitational force (Boussinesq hypothesis);
- the energy generation term was neglected, because the effects of internal heat were not considered (for example absorption or emission radiation) neither the humidity which could be responsible for the latent heat exchange;
- there are no Coriolis force effects nor rotation effects of the coordinate system.

The equations of mass conservation, momentum and energy (temperature) can be written in a non-dimensional form as:

$$\boldsymbol{\nabla}.\boldsymbol{V} = 0 \tag{1}$$

$$\frac{\partial \boldsymbol{V}}{\partial t} + \boldsymbol{\nabla}.(\boldsymbol{V}\boldsymbol{V}) = -\frac{1}{\rho}\boldsymbol{\nabla}P + \boldsymbol{\nabla}.\left(\frac{\nu_{ef}}{Re}\boldsymbol{\nabla}\boldsymbol{V}\right) + \frac{1}{Re}(\boldsymbol{\nabla}\nu_{ef})\boldsymbol{\nabla}^T\boldsymbol{V} + Ri\theta + \boldsymbol{f} \tag{2}$$

$$\frac{\partial \theta}{\partial t} + \boldsymbol{\nabla}.(\boldsymbol{V}\theta) = \boldsymbol{\nabla}.\left[\left(\frac{\nu}{RePr} + \frac{\nu_t}{RePr_t}\right)\boldsymbol{\nabla}\theta\right] + q \tag{3}$$

where $\boldsymbol{V}$, P and θ are the non-dimensional velocity fields, pressure and temperature, respectively. $\nu_{ef} = \nu + \nu_t$ is the effective cinematic viscosity and ν_t the turbulent cinematic viscosity which is obtained by the Smagorinsky model for the turbulent simulations. The relevant non-dimensional parameters are: Reynolds (Re), Prandtl (Pr) and Richardson ($Ri = \frac{Gr}{Re^2}$), where Pr_t is the turbulent Prandtl number. $\boldsymbol{f}$ is the Eulerian force, added to the momentum equations (Immersed Boundary Method, Lima E Silva (2002)), given by:

$$\boldsymbol{f}(\boldsymbol{x}, t) = \sum \boldsymbol{F}(\boldsymbol{x_k}, t) D_{ij}(\boldsymbol{x_k})(\boldsymbol{x} - \boldsymbol{x_k}) \Delta s^2. \tag{4}$$

$D_{ij}(\boldsymbol{x_k})$ is the interpolation function, $\boldsymbol{x_k} = (X_k, Y_k)$ are the Lagrangian points over the interface, Δs is the distance between two Lagrangian points and $\boldsymbol{x} = (X_i, Y_j)$ are the Eulerian points. $\boldsymbol{F}(\boldsymbol{x_k}, t)$ is the Lagrangian force density, modeled by the PVM presented below. $\boldsymbol{f}(\boldsymbol{x}, t)$ is the Eulerian force which is different from zero only over the interface. Equation (4) represents the fluid/solid interaction.

Analogously, q has the function to model the temperature of the cylinder and it was given by:

$$q(\boldsymbol{x}, t) = \sum Q(\boldsymbol{x_k}, t) D_{ij}(\boldsymbol{x_k})(\boldsymbol{x} - \boldsymbol{x_k}) \Delta s^2, \tag{5}$$

where Q is calculated based on the temperature values interpolated from the Eulerian grid. In the present work the distribution function suggested by Peskin and McQueen (1994) is used:

$$D_{ij}(\boldsymbol{x_k}) = \frac{g[(X_k - X_i)/h][(Y_k - Y_j)/h]}{h^2}, \tag{6}$$

$$g(r) = \begin{cases} g_1(r) & if \quad | r | < 1, \\ \frac{1}{2} - f_1(2- | r |) & if \quad 1 < | r | < 2, \\ 0 & if \quad | r | > 2, \end{cases} \tag{7}$$

where:

$$g_1(r) = \frac{3 - 2 | r | + \sqrt{1 + 4 | r | - 4 | r |^2}}{8}. \tag{8}$$

r is equal to $(X_k - X_i)/h$ or $(Y_k - Y_j)/h$ and h is Eulerian grid length. The PVM proposed by Lima E Silva *et al.* (2003) is used to calculate the Lagrangian force, $\boldsymbol{F}(\boldsymbol{x_k}, t)$:

$$\boldsymbol{F}(\boldsymbol{x_k}, t) = \frac{\partial \boldsymbol{V}}{\partial t}(\boldsymbol{x_k}, t) + (\boldsymbol{V}.\boldsymbol{\nabla})\boldsymbol{V}(\boldsymbol{x_k}, t) + -\boldsymbol{\nabla}[\nu_{ef}(\boldsymbol{\nabla V} + \boldsymbol{\nabla}^T \boldsymbol{V})] + \frac{1}{\rho}\boldsymbol{\nabla} p(\boldsymbol{x_k}, t). \tag{9}$$

The terms on the right hand side of Eq. (9) are obtained from Eulerian field interpolation. The same procedure is done for Q, calculated by:

$$Q(\boldsymbol{x_k}, t) = \frac{\partial \theta}{\partial t}(\boldsymbol{x_k}, t) + \boldsymbol{\nabla}.(\boldsymbol{V}(\boldsymbol{x_k}, t)\theta(\boldsymbol{x_k}, t)) - \boldsymbol{\nabla}. \left[\left(\frac{\nu}{RePr} + \frac{\nu_t}{RePr_t} \right) \boldsymbol{\nabla}\theta(\boldsymbol{x_k}, t) \right] \tag{10}$$

After calculating $Q(\boldsymbol{x_k}, t)$ over the Lagrangian grid, the Eulerian variable $q(\boldsymbol{x}, t)$, Eq. (5), is obtained to be used in Eq. (3) for the new temperature field calculation. The distribution and interpolation procedures are detailed in Lima E Silva *et al.* (2003).

3. Numerical Results

The discretized forms of the transport equations, Eqs. (1), (2) and (3) are derivated by using the finite difference method with second order Runge-Kutta for the momentum equations and the second order Adams Basforth for the temperature equation. The fractional step method, presented by Armfield and Street (1999) was applied as a pressure correction.

Two different configurations were studied with a number of geometries (NG) varying from three to ten. In case 1 the dynamic and the thermal field were analyzed in a triangular configuration in forced and mixed convection. In case 2 a bank of cylinders disposed side by side was simulated in a confined domain only in forced convection. The results of drag and lift coefficients and Nusselt and Strouhal numbers are presented for both cases.

3.1 Case 1 - Triangular Configuration

Figure 1 shows the two-dimensional domain and the boundary conditions for the triangular configuration. For these simulations the following values were chosen, $L_x = 20d$, $L_y = 6d$, $S_h = 2.5d, 3d$ and $S_v = 1.25d, 1.5d$ where d is the cylinders diameter. The horizontal channel has solid walls on upper and lower faces and the inner flow has a uniform profile. A non-uniform grid of 274 x 171 points was used in a uniform region near the cylinders as shown in the closer view of Fig. 1. The simulation was carried out for Reynolds at 200, Prandtl at 0,7 and for the cases of forced convection $Ri = 0.0$ and mixed convection $Ri = 2.0$ and $Ri = 5.0$.

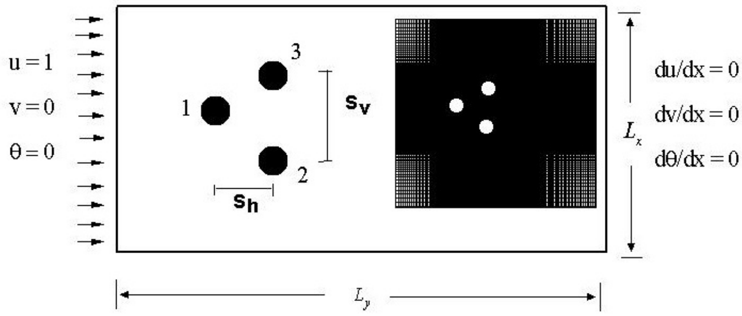


Figure 1. Computational domain and boundary conditions for case 1.

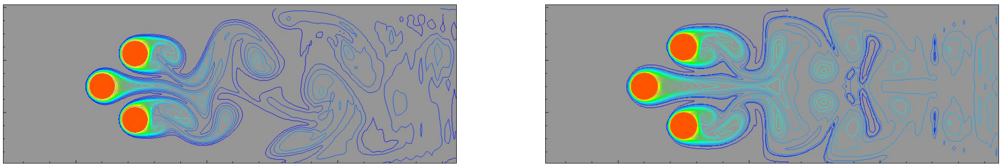
Table 1 presents the results of drag and lift coefficients, Strouhal and Nusselt numbers for each cylinder in each simulation.

Table 1. Drag and lift coefficients, Strouhal and Nusselt numbers for the three Cylinders

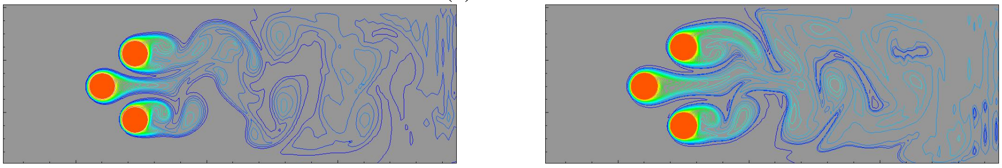
$S_h = 1.25$ and $S_v = 2.5$						$Sh = 1.5$ and $S_v = 3.0$					
Cylinder	Ri	C_d	C_l	St	Nu	Cylinder	Ri	C_d	C_l	St	Nu
1	0	2.6	10^{-4}	0.3	8.3	1	0	2.4	10^{-6}	0.3	5.6
	2	2.5	0.01	0.4	8.3		2	2.3	0.3	0.4	7.1
	5	6.1	5.0	0.5	9.9		5	5.8	5.4	0.5	10.3
2	0	2.7	0.3	0.3	8.9	2	0	2.9	0.3	0.3	6.2
	2	4.2	2.9	0.4	10.1		2	4.5	2.3	0.4	8.4
	5	8.3	6.6	0.5	11.4		5	7.3	5.7	0.5	11.5
3	0	2.7	-0.3	0.3	6.1	3	0	2.9	-0.3	0.3	6.2
	2	1.9	1.5	0.3	8.3		2	1.9	1.6	0.3	7.1
	5	1.5	5.4	0.5	7.4		5	1.9	4.9	0.5	7.1

In the simulation with forced convection ($Ri = 0$) the flow is symmetric when the distance between the cylinders is higher and each cylinder has its own vortex street. Cylinders 2 and 3 present the same values of drag, lift, Strouhal and Nusselt, as expected. For the other two simulations with mixed convection, the flow is not symmetric on the front stagnation point, because the direction of the buoyancy effect is ortogonal to the flow direction. Nusselt number is higher for mixed convection than for forced convection. It is explained from that upward buoyancy effect can make the flow patterns behind the cylinders toward the upper-right surfaces of cylinders, Hu and Jue (2006). The values of drag coefficient for cylinder 1 are smaller than for cylinders 2 and 3 because they are closer to the wall and have obvious vortex shedding. The exception is when the flow is reversal for $Ri = 5$ and a decrease of the drag coefficient occurs.

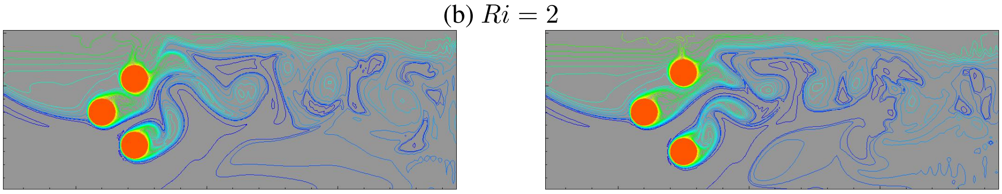
Figure 2 (a), (b) and (c) shows the isotherms for $Ri = 0$ (or null Grashof number), $Ri = 2$ and $Ri = 5$, respectively. The buoyancy effect can change the flow pattern as shown in Figures 2 (b) and (c). Therefore, the non-symmetric flow patterns on the front stagnation point of cylinder 1 exist, and there is a reversal that includes two vortices along the downstream location near the lower channel-wall as can be seen in Fig. 3 with the streamlines.



(a) $Ri = 0$

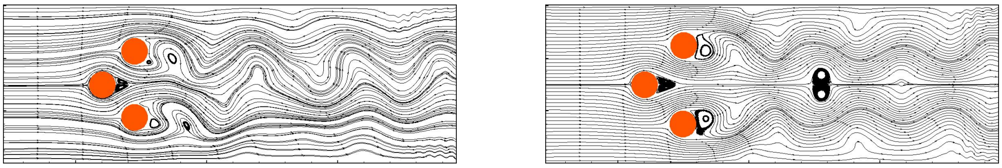


(b) $Ri = 2$

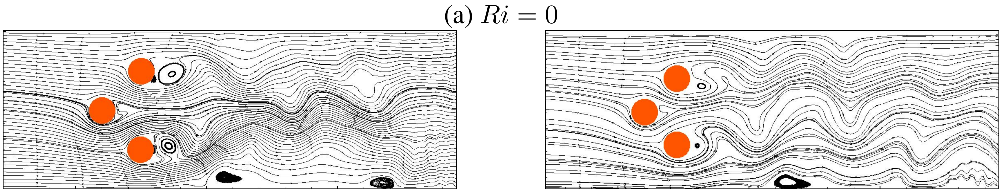


(c) $Ri = 5$

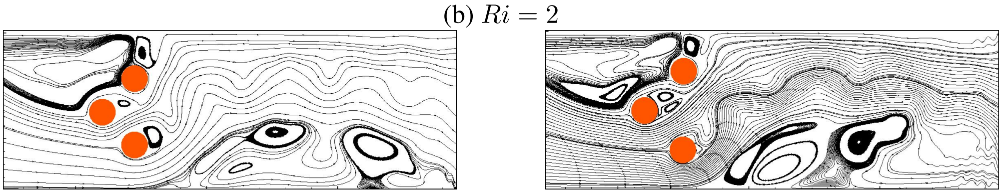
Figure 2. Instantaneous Isotherms for (a) $Ri = 0$, (b) $Ri = 2$ and (c) $Ri = 5$. Left column $S_h = 2.5d$ and $S_v = 1.25d$. Right column $S_h = 3d$ and $S_v = 1.5d$.



(a) $Ri = 0$



(b) $Ri = 2$



(c) $Ri = 5$

Figure 3. Streamlines for (a) $Ri = 0$, (b) $Ri = 2$ and (c) $Ri = 5$. Left column $S_h = 2.5d$ and $S_v = 1.25d$. Right column $S_h = 3d$ and $S_v = 1.5d$

Other simulations with forced convection and different values os S_h and S_v were also done and the results showed the high influence of these parameters on the temperature fields and aerodynamic coefficients. It was observed that as the distance between the cylinders increases, the vortex shedding starts sooner and the flow symmetry is lost. For small values of S_h and S_v ($S_h = 2$ and $S_v = 2$), the vortices behind cylinders 2 and 3 are bigger and the flow develops as though the cylinders were a single structure. There is a great influence of the vortices from cylinders 1 on cylinders 2 and 3 when S_v is increased.

3.2 Case 2 - Single Row of Cylinders

This case consists of a bank of cylinders disposed side by side and confined in a horizontal two-dimensional channel, as presented in Fig. 4. The domain has dimensions of $L_x = 40d$, $L_y = 21d$. The simulations were carried out by varying the number of cylinders, the distances between them (g) and distances to the wall (h). For this configuration only the forced convection was considered.

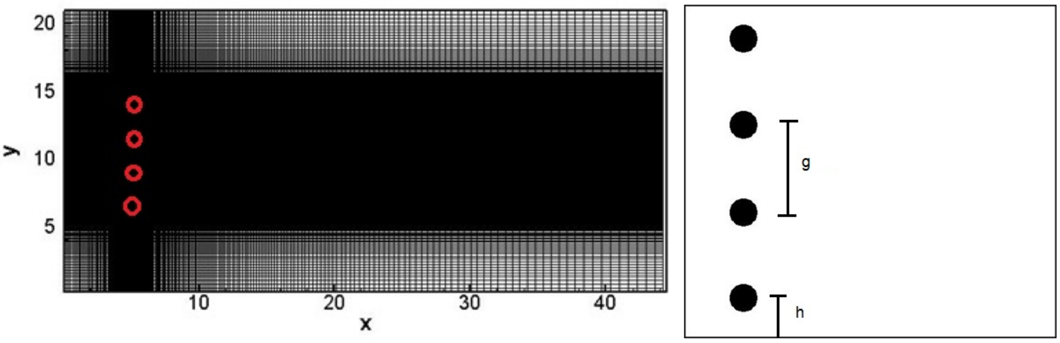


Figure 4. Computational domain and boundary conditions for case 2.

For a single row of cylinders of equal diameter, different regimes were observed when the gap between cylinders and their distances to the wall were changed. Reynolds number was 150 for all simulations in case 2 and the grid was also refined near the cylinders. Table 2 shows the drag coefficient and Nusselt number for the first cylinder near the bottom. n is the number of geometries, g the distance between the centers of the two adjacent cylinders and h the distance of the

Table 2. Drag Coefficient and Nusselt number for the First Cylinder on the Bottom

n	g	h	C_d	Nu
6	2.0	5.0	2.7	4.7
6	2.5	3.75	2.3	4.5
6	3.0	2.5	2.1	4.4
6	3.5	1.25	2.1	4.4
6	3.7	0.75	2.1	4.4
7	1.2	6.4	2.9	3.7
7	2.0	4.0	3.0	4.7
7	2.5	2.5	2.5	4.6
7	3.0	1.0	2.4	4.6
7	3.2	0.4	2.3	4.4
8	1.5	4.75	3.6	5.1
8	2.0	3.0	3.1	4.8
8	2.5	1.25	2.7	4.7
8	2.7	0.55	2.7	4.7
9	1.2	5.2	4.2	4.2
9	1.5	4.0	3.7	5.2
9	2.0	2.0	3.3	4.9
9	2.1	1.6	3.3	4.9
9	2.2	1.2	3.2	4.8
10	1.2	4.6	5.0	4.4
10	1.5	3.25	4.4	5.4
10	1.7	2.35	4.1	5.1
10	2.0	1.0	3.9	5.0

first and last cylinders centers to the wall. It is possible to note the influence of the wall distance from cylinders and the gap in these values. The drag decreases as g increases for a number of cylinders . For the same distance between two cylinders, the drag coefficient increases by increasing n and decreasing h. Nusselt number presents the same behavior for all simulations, except for $n = 10$ and $g = 1.2$. In this case, the group of cylinders is similar to a single body. Nusselt number increases when the gap between the cylinders decrease up to 1.5. Below this value, Nusselt value decreases for all cylinders. This behavior is expected because the convective heat transfer coefficient decrease as the bank forms a single set.

Figure 5 shows Nusselt number for a central cylinder as a function of gap g. It is possible to observe a maximum value close to $g = 1.5$ for all simulations. This is probably the optimum configuration to maximize the heat transfer.

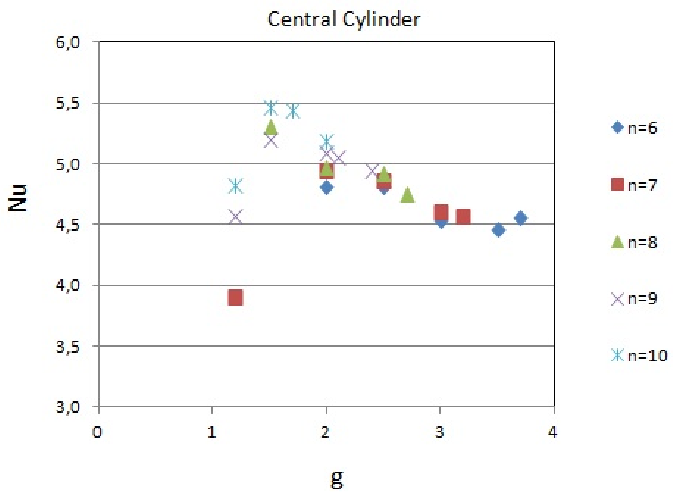


Figure 5. Nusselt number as a function of gap for all simulations in case 2.

4. CONCLUSIONS

The Immersed Boundary Method with the Virtual Physical Model was used to simulate flows on a bank of heated cylinders. The heat transfer analysis is simplified with this methodology because the relevant parameters are obtained directly from the Eulerian field even if a complex geometry is presented. It is very simple to calculate drag and lift coefficients, for example, over any kind of geometry. In mixed convection, the flow interactions among circular cylinders and channel walls are quite different from those in forced convection. Therefore, the average time Nusselt number, drag and lift coefficients are also different. The flow dynamics and isotherms are symmetric on the front stagnation point only in forced convection. In mixed convection, the effect of upward buoyancy generates an inclined flow and some vortices along the lower wall of the channel. For the simulations with a row of cylinders, the results showed the great influence of the walls as the number of cylinders increased.

5. ACKNOWLEDGEMENTS

The authors would like to acknowledge the financial support from FAPEMIG, CNPq and Federal University of Itajubá-MG.

6. REFERENCES

Giacomello, M.V., Rocha, L.A.O., Schettini, E.B.C. and Silvestrini, J.H., 2006. "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.

Hu, H-W., P.S.W.H.S.Y. and Jue, T.C., 2006. "Transient mixed convective heat transfer predictions around three heated cylinders in a horizontal channel". *International Journal of Numerical Methods for Heat and Fluid Flow*, Vol. 6, pp. 674–692.

Huang, Z., Olson, J.A., Kerekes, R.J. and Green, S.I., 2006. "Numerical simulation of the flow around rows of cylinders". *Computers and Fluids*, Vol. 35, pp. 485–491.

Ishigai, S. and Nishikawa, E., 1975. "Experimental study of structure of gas flow in tube banks with tube axes normal to flow (part ii: On the structure of gas flow in single-column, single-row, and doublerows tube banks)". *Bull JSME*, Vol. 18, No. 119, pp. 528–535.

Lima E Silva, A.L.F., 2002. *Desenvolvimento e Implementação de uma Nova Metodologia para Modelagem de Escoamentos sobre Geometrias Complexas: Método da Fronteira Imersa com Modelo Físico Virtual*. Ph.D. thesis, Universidade Federal de Uberlândia, Uberlândia. MG, Brasil.

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*, Vol. 69-71, 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.

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". *submetido ao Journal of Heat Transfer*.

Patnaik, B.S.V., Narayana, P.A.A. and Seetharamu, K.N., 1999a. "Numerical simulation of laminar flow past a transversely vibrating circular cylinder". *Journal of Sound and Vibration*, Vol. 228, No. 3.

Patnaik, B.S.V., Narayana, P.A.A. and Seetharamu, K.N., 1999b. "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.

Peskin, C.S. and McQueen, D.M., 1994. "A general method for the computer simulation of biological systems interacting with fluids". *SEB Symposium on Biological Fluid Dynamics, Leeds, England*, Vol. 49, pp. 265–276.

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, No. 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. 477–539.

7. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.