A LAGRANGIAN PARTICLE METHOD FOR THE HEAT TRANSFER PROBLEM WITH BUOYANCY FORCE

Marcos Vinícius Roselli, marcosvroselli@yahoo.com.br
Luiz Antonio Alcântara Pereira, luizantp@unifei.edu.br
Mechanical Engineering Institute, Federal University of Itajubá (UNIFEI), Itajubá, Brazil

Alex Mendonça Bimbato, alexbimbato@feg.unesp.br
Department of Energy, São Paulo State University (UNESP), Guaratinguetá, Brazil

Abstract. *A Lagrangian vortex method is extended to take into account mixed heat transfer. The energy equation is linked to the Navier-Stokes equations by using the Boussinesq approximation; therefore, the buoyancy force is computed. Nascent vortex elements and nascent temperature particles are introduced into the flow field close to a body surface. The present model simulates the interaction between vorticity and temperature; the main contribution of paper is the creation of vorticity from heat. The two-dimensional strategy proposes the creation of vorticity from heat at each time step of the simulation by changing the strength of cluster of vortex elements due to temperature gradients. The numerical results are encouraging and show the potentialities of the present method to simulate combined forced and natural convection in the separated flow region behind the circular cylinder.*

Keywords: *vortex and temperature elements; panels method, aerodynamics of bluff body, buoyancy force*.

1. INTRODUCTION

The incompressible fluid flow and thermal fields created by the interaction between vorticity and heat are important in both fundamental research and practical applications. The Lagrangian particles method including both vortex and temperature elements has been employed in the literature to a variety of problems from the view point of engineering applications, such as shear layer, external flow, internal flow, flow with multiple bodies, etc. The main advantages of a typical temperature particles method are that both vortex and temperature elements are placed only where vorticity is non-zero and the boundary conditions in the far field are automatically satisfied. In general, the Lagrangian methods are especially useful for flows which are dominated by localized vorticity distributions, e.g., shear flows, wakes, and jets (Kamemoto, 2004).

Ghoniem and Sherman (1985) investigated one-dimensional heat diffusion using random walk scheme (Chorin, 1973, 1978). They presented a complete analysis of temperature particles with different properties and the vorticity generation due to the heat transfer process. Kamemoto and Miyasaka (1999) employed the core spreading model (Leonard, 1980) to simulate the forced convection heat transfer around a heated circular cylinder at high Reynolds numbers. Discrete temperature elements with thermal core were introduced in the thin thermal layer along the body surface. Although they made an approximation that the temperature in the thermal layer was constant along the normal direction, the time-averaged Nusselt number distribution showed reasonable agreement with that of experiment (Igarashi, 1984).

Ogami (2001) introduced two models for creating vortices from temperature particles to simulate buoyancy force. The model 1 changed the strength of each vortex element during each time step of the simulation through a direct interpretation of the vorticity transport equation and consequently was a natural extension of the method of Ghoniem and Sherman (1985) to two-dimensional problems. This model is suitable for the regions where both vortex and temperature elements are overlapped. The model 2 indicated that a vortex pair (one positive and one negative) could be generated from one temperature particle. This kind of idea was based on the fact that the slope of a temperature element is precisely approximated by the density distribution of two Gaussian particles with opposite strength. In addition to these two models, the diffusion velocity method (Ogami and Akamatsu, 1991) was used in order to handle heat and vorticity diffusion. In the one-dimensional field, the models were compared with analytic solution, and the accuracy and validity were clarified. In the two-dimensional field, the application sample with the natural convection and the interaction between heat and vortex was shown.

Alcântara Pereira and Hirata (2003) analyzed the mechanism of heat transfer in the separated flow region behind a heated circular cylinder surface considering the similarity between the energy equation and the vorticity transport equation for a two-dimensional and incompressible flow (Kamemoto and Miyasaka (1999). Vortex elements and temperature particles were introduced into the flow field close to a wall surface to ensure boundary conditions of problem, however, no attempt to include interaction between vorticity and heat was made.

The purpose of the present study is to extend the Lagrangian vortex code developed by Alcântara Pereira and Hirata (2003) including the buoyancy force effect. The model 1 presented by Ogami (2001) is numerically more appropriated (lower computational cost) because no additional vortex elements generation is necessary into the fluid domain. The Boussinesq approximation can be applied to the incompressible Navier-Stokes equations, with constant physical properties except for the buoyancy effect due to the moderated density variations, changing the vorticity field. This seems that then the flow field temperature is not high, the thermally driven velocity is small relative to sonic speed.

2. PROBLEM FORMULATION

The equations governing a typical two-dimensional and incompressible flow of a Newtonian fluid with constant properties around a heated body surface can be written as:

$$\nabla \cdot \mathbf{u} = 0 \ , \tag{1}$$

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u} = \frac{Gr}{Re^2}\,\theta\,\mathbf{j} - \nabla p + \frac{1}{Re}\nabla^2\mathbf{u}\text{, where } \theta = \frac{T - T_\infty}{T_w - T_\infty}\,, \tag{2}$$

$$\frac{\partial \theta}{\partial t} + (\mathbf{u} \cdot \nabla)\theta = \frac{1}{Re\,Pr}\nabla^2\theta \ . \tag{3}$$

In the above equations $\mathbf{u}$ is the velocity field, p is the pressure, θ is the dimensionless temperature (being respectively T_w the constant body surface temperature and T_∞ the far away fluid temperature) and $t = d/U$ is the time scale (being respectively U and d the velocity scale and the length scale of the physical problem). The dimensionless numbers, respectively, the Reynolds number, the Prandtl number and the Richardson number, are defined as:

$$Re = \frac{Ud}{\nu}\,, \tag{4}$$

$$Pr = \frac{\nu}{\alpha}\,, \tag{5}$$

$$Ri = \frac{Gr}{Re^2}\,, \tag{6}$$

where ν is the kinematic viscosity and α is the thermal diffusivity.

The governing equations (1) and (2) are described with the vorticity transport equation by taking the rotational of the Navier-Stokes equations, such as:

$$\frac{\partial \omega}{\partial t} + (\mathbf{u} \cdot \nabla)\,\omega = Ri\,\frac{\partial \theta}{\partial x} + \frac{1}{Re}\nabla^2\omega\,. \tag{7}$$

The pressure field is described with the pressure Poisson equation by taking the divergence of the Navier-Stokes equations (Shintani and Akamatsu, 1994), such as:

$$H\overline{Y_i} - \int_{S_1} \overline{Y}\nabla G_i \cdot \mathbf{e}_n dS = \iint_\Omega \nabla G_i \cdot (\mathbf{u} \times \boldsymbol{\omega}) d\Omega - \upsilon \int_{S_1} (\nabla G_i \times \boldsymbol{\omega}) \cdot \mathbf{e}_n dS \ \text{ and } Y = p + \rho\frac{u^2}{2}\,,\ u = |\mathbf{u}|\,, \tag{8}$$

where H is 1.0 inside the flow domain and is 0.5 on the surface body. Here, $G_i = (1/2\pi)\log R^{-1}$ is the fundamental solution of Laplace equation, R being the distance from i^{th} vortex element to the field point. It is worth to observe that this formulation is specially suited for a Lagrangian scheme because it utilizes the velocity and vorticity field defined at the position of the vortices in the cloud. Therefore it does not require any additional calculation at mesh points. Numerically, Eq. (8) is solved by mean of a set of simultaneous equations for pressure Y_i.

The impermeability condition on the body surface is imposed by using source flat panels (Katz and Plotkin, 1991). Each nascent Lamb vortex (Kundu, 1990) must be positioned at a shedding point placed near the pivotal point of each flat panel. The strength of the nascent vortex element $\Delta\Gamma$ is obtained imposing the no-slip condition on the all pivotal points. Therefore, each panel has a pivotal point to ensure simultaneously the impermeability and the no-slip conditions.

The strength of each nascent temperature element is obtained through an interpretation for the Fourier law close to the heated body surface (Nakamura et al., 2001):

$$\frac{\partial q}{\partial t} = -\alpha \frac{\partial T}{\partial n}$$, where n denotes the normal direction to the surface. (9)

The pivotal points are used to ensure the value of Tw as constant during all numerical simulation.

3. NUMERICAL METHOD

The foundation of a temperature particles method rests on the use of Eq. (3) and Eq. (7) to track a fluid based on the evolution of vorticity and temperature fields. The vorticity and energy equations are free of the computational instabilities associated with the convective term. Computational simulation requires the discretization in space and time of Eq. (3) and Eq. (7). The two-dimensional, incompressible unsteady flow around a heated body surface uses an algorithm based on Eq (3) and Eq. (7) that splits the advective-diffusive operators in the following forms (Chorin, 1973):

$$\frac{D\theta}{Dt} = \frac{\partial \theta}{\partial t} + (\mathbf{u} \cdot \nabla)\theta = 0 \ , \tag{10}$$

$$\frac{D\omega}{Dt} = \frac{\partial \omega}{\partial t} + (\mathbf{u} \cdot \nabla)\omega = 0 \ , \tag{11}$$

$$\frac{\partial \theta}{\partial t} = \frac{1}{\mathrm{Re}\,\mathrm{Pr}} \nabla^2 \theta \ , \tag{12}$$

$$\frac{\partial \omega}{\partial t} = \frac{1}{\mathrm{Re}} \nabla^2 \omega \ . \tag{13}$$

The temperature particles method represents the heat by using temperature elements, whose transport by advection and diffusion is carried out in a sequence within the same time increment of Δt. First, a Lagrangian approach is used to simulate the heat advective process, governed by Eq. (10). The advective motion of each temperature element is determined by integration of each temperature path equation, which here is solved using a first-order Euler scheme (Ferziger, 1981). The total velocity in the temperature element point is calculated considering the uniform flow, the source panels and the vortex cloud (the Biot-Savart law).

The strength of each nascent temperature element is computed due to the thermal flux such as (see Eq. (9)):

$$\Delta q_k = \frac{1}{\mathrm{RePr}} \frac{T_k - T_\infty}{T_w - T_\infty} \frac{\Delta s_k \Delta t}{\varepsilon} , \tag{14}$$

where Δs_k is the panel length, ε is the normal distance between the pivotal point k and the shedding point and Δt is the time increment. The temperature induced by each temperature element is given by a Gaussian distribution:

$$T(r) = \frac{\Delta q}{\pi \sigma_T^2} \exp\left(-\frac{r^2}{\sigma_T^2} \right)$$, σ_T is the temperature core radius. (15)

The process of heat diffusion, governed by Eq. (12), is simulated using the random walk method (Lewis, 1999):

$$\varsigma_k(t) = \sqrt{\frac{4\Delta t}{\mathrm{RePr}} \ln\left(\frac{1}{P}\right)} \left[\cos(2\pi Q)_x + \sin(2\pi Q)_y \right], \tag{16}$$

where P and Q are random numbers between 0 and 1.

The vorticity advective process is governed by Eq. (11). The advective motion of each vortex element is determined by integration of each vortex path equation, which here is solved using a first-order Euler scheme. The total velocity in the vortex element point is calculated considering the uniform flow, the source panels and the vortex cloud (the Biot-Savart law).

The process of vorticity diffusion, governed by Eq. (13), is simulated using the random walk method (Lewis, 1999):

$$\varsigma_k(t) = \sqrt{\frac{4\Delta t}{Re}\ln\left(\frac{1}{P}\right)}\left[\cos(2\pi Q)_x + \sin(2\pi Q)_y\right]. \tag{17}$$

where P and Q are random numbers between 0 and 1.

Finally, the intensity of each vortex element is increased by the mount $\Delta\Gamma^*$ due to direct interpretation from Eq. (7) in the following manner (Ogami, 2001):

$$\frac{\partial \omega}{\partial t} = Ri\frac{\partial \theta}{\partial x}. \tag{18}$$

Using the circulation definition, $\Gamma = \iint_A \boldsymbol{\omega}\cdot\mathbf{n}dA$, the Eq. (18) is discretized such as:

$$\Delta\Gamma^* = Ri\frac{\partial\theta}{\partial x}\Delta t\,\Delta A \text{, } \Delta A \text{ is the area occupied by the vortex element with core radius of } \sigma. \tag{19}$$

The derivative on Eq. (19) is solved by derivative centered difference in the form:

$$\left.\frac{d\theta}{dx}\right|_i = \frac{\theta_{i+1}-\theta_{i-1}}{2\Delta x} - \frac{\Delta x^2}{6}\left.\frac{d^3\theta}{dx^3}\right|_i - \frac{\Delta x^4}{120}\left.\frac{d^5\theta}{dx^5}\right|_i - \ldots \tag{20}$$

The above equation is truncated in the first term, where Δx the i[th] vortex core radius. Therefore, the cluster of temperature elements is used to compute the dimensionless temperatures θ_{i+1} and θ_{i-1} in the neighborhood of the i[th] vortex element.

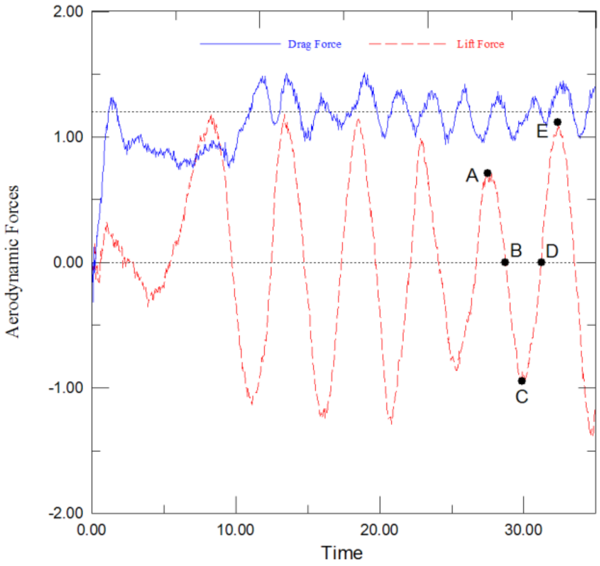
4. RESULTS, CONCLUSIONS AND FUTURE WORKS

In order to test the implementation of the vortex and temperature elements scheme, it was chosen the first test case successfully investigated by Bimbato et al. (2011) in the past assuming (without heat transfer): number of source flat panels used to represent the cylinder surface of $NP = 300$, time increment of $\Delta t = 0.05$ and Lamb vortex core of $\sigma_l = 0.001d$. For this simulation (so referred as case I) were also introduced temperature elements with core radius of $\sigma_t = 0.001d$. Therefore, at each pivotal point were shedding $NP = 300$ vortex elements, $NP = 300$ temperature elements during each time increment, body surface temperature of T_W = $303K$ and fluid temperature of T_∞ = $283K$. Table 1 presents all cases investigated in this present paper. The Prandtl number was fixed as $Pr = 1.0$ and the Richardson number, when assumed as $Ri = 0.0$, indicated forced convection. All the aerodynamic loads computations bellow described were evaluated between $15 \le t \le 35$.

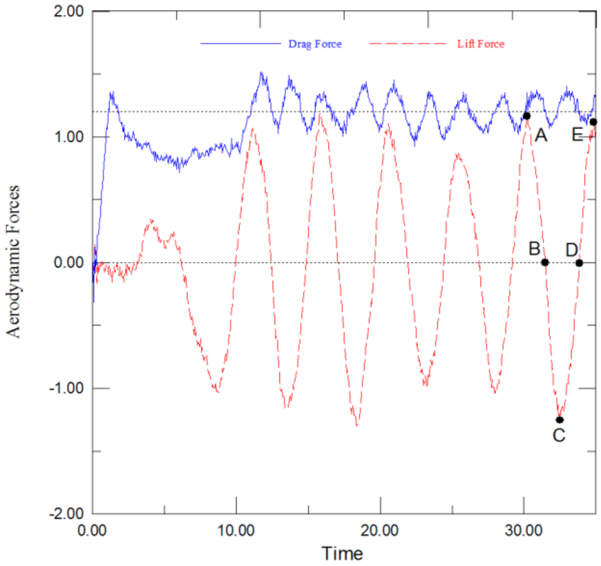
Table 1. Time-averaged values of drag and lift coefficients and Strouhal number for heated surface of circular cylinder ($Re = 10^5$ and $Pr = 1.0$)

Case	Ri	$\overline{C_D}$	$\overline{C_L}$	$\overline{St}$	$\overline{\theta_{sep}}$
Experimental result (Blevins,1984)	–	1.20	–	0.19	82°
Present simulation (case I)	0.00	1.200	-0.128	0.200	84°
Present simulation (case II)	0.01	1.167	0.015	0.200	86°
Present simulation (case III)	0.10	1.202	0.009	0.200	88°
Present simulation (case IV)	1.00	1.205	-0.088	0.200	82°

Two computed values of time histories of drag and lift coefficients for the heated surface case of circular cylinder are plotted in Fig. 1. Figure 1(a) shows the case with $Ri = 0.0$, in which the vortex shedding effect can be seen in oscillations of the lift and drag coefficients. As soon as the numerical transient is over and the periodic steady state regime is reached (from t=12 on, approximately) the lift coefficient shows a mean variation between – 1.40 and 1.11, approximately, with a dimensionless frequency about twice the frequency of the drag coefficient, in accordance to the physics involved in the flow (Gerrard, 1966). The frequency of this detachment of vortices is measured by the Strouhal number defined as $St = fd/U$, being f the detachment frequency of vortical structures (see in the Tab. 1).



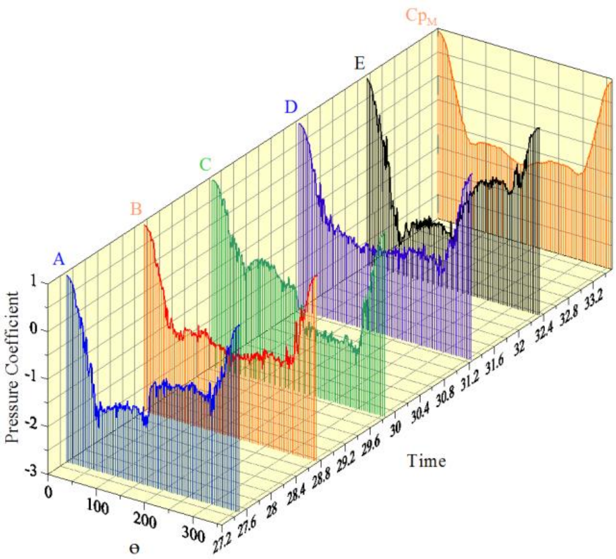
(a) with forced convection (case I: $Ri = 0.0$)



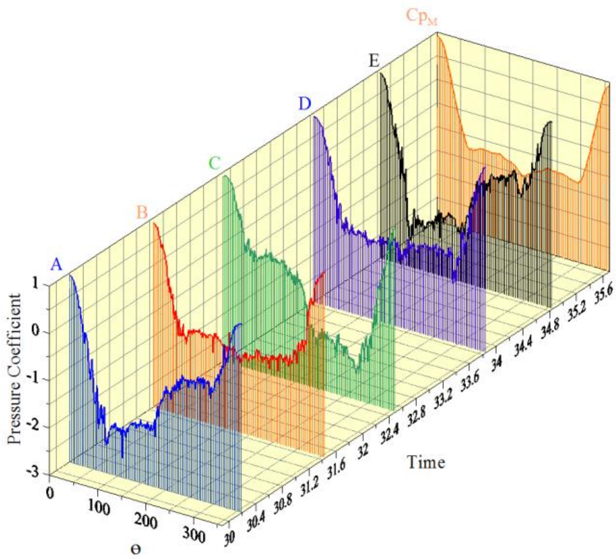
(b) with mixed convection (case III: $Ri = 0.1$)

Figure 1. Time histories of drag and lift coefficients for the heated cylinder with $Re = 10^5$ and $Pr = 1.0$.

The separation point obtained by the present simulation with forced convection (case I) is about $\theta \cong 84°$ while Blevins (1984) experimentally measured $\theta = 82°$. Another experimental investigation made by Son and Hanratty (1969) determined a value of $\theta = 78°$ for the separation angle. A very interesting observation was presented by Achenbach (1968) for $Re = 10^5$ (the subcritical flow regime): "it was found that the laminar boundary layer separates at $\theta = 78°$". Just before transition into the critical region at $Re = 2.6 \times 10^5$, the boundary layer is still laminar and separates at an angle $\theta = 94°$. Hence, separation takes place in the laminar mode as experimentally expected for a subcritical Reynolds number forming free shear layers.



(a) with forced convection (case I: $Ri = 0.0$)

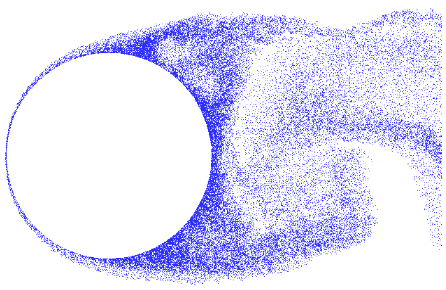


(b) with mixed convection (case III: $Ri = 0.1$)

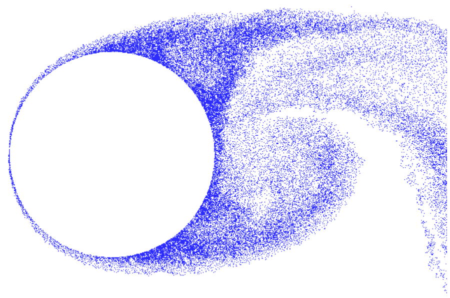
Figure 2. Instantaneous pressure distributions on the heated cylinder surface with $Re = 10^5$ and $Pr = 1.0$.

Because the mixed convection simulated in this research, it was possible to identify a moderate variation of the separation point when $Ri = 0.1$ (case III) from $\theta \cong 84°$ to $\theta \cong 88°$. The sequence of events already described by Bimbato et al. (2011) is here referred to explain the mechanics of the formation region of vortices behind bluff bodies.

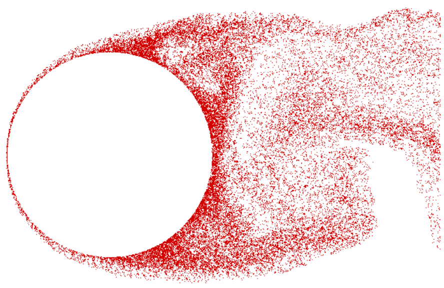
Starting of the Fig. 3(a) a new anti-clockwise vortex structure surges at the low side of the cylinder surface toward the viscous wake (case I: $Ri = 0.0$). This event can be identified through of the Point C in the Fig. 1(a), where the lift coefficient assumes its lower value. Note that the same Point C in the Fig. 2(a) seems lower pressure distributions at the low side of the body surface. At this very moment the clockwise vortex structure detaches from the rear part of the cylinder surface and is incorporated to the viscous wake at the instant corresponding to the Point D; this process creates a low pressure region at the rear part of the cylinder which is associated to the high drag value, see the Point D in the Fig. 2(a). Right after the event a new clockwise vortex structure starts at the high side of the cylinder surface (Point E) and the above described sequence of events repeats all over again.



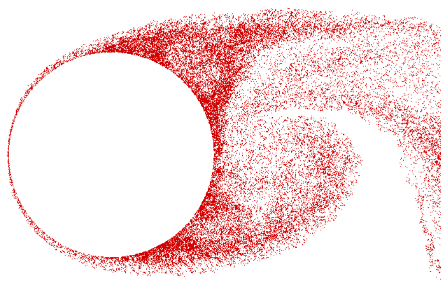
(a) vortex elements (case I: $Ri = 0.0$)



(b) vortex elements (case III: $Ri = 0.1$)



(c) temperature elements (case I: $Ri = 0.0$)



(d) temperature elements (case III: $Ri = 0.1$)

Figure 3. Near wakes behavior for heated cylinder surface with $Re = 10^5$ and $Pr = 1.0$: Point C.

Figure 3(b) shows the same anti-clockwise vortex structure identified by Point C when $Ri = 0.1$ (case III). The mixed convection did not affect the anti-clockwise vortex structure, however it was identified that moderate variation of the separation point (see Tab. 1). The mean pressure distributions (see C_{P_M} in the Fig. (2)) and the mean separation point values are evaluated between the instants identified from Point A until Point E in the Fig. 2(a) and Fig. 2(b). Figure 4 summarizes the time-averaged pressure coefficient distributions for all numerical cases reported in the Tab. 1.

Despite the differences presented in this preliminary investigation, the results are promising, that encourages performing additional tests in order to explore the phenomena in more details.

In order to clarify the mechanisms of the unsteady mixed-convective heat transfer in the separated flow region, some providence will take into account for future work. A typical simulation of this paper spent around 240 hours using the following configuration: INTEL CORE I7 - 2.8GHZ (BOX) 8MB CACHE (i7-860), MB INTEL DH55TC, 8GB RAM DDR3 1333 MHZ. The vortex-vortex and temperature-vortex interactions were subroutines that spend higher computational cost, because they involved the Biot-Savart law. Another one was the subroutine to compute the buoyancy force effect (Eq. (19)). From the higher computational cost identified here it is possible to conclude that simulations with large numbers of particles require great compute power, thus involve large amounts of time. Therefore, it is imperative to make use of a parallel implementation so that results can be obtained in a feasible time. As first providence it will be used OpenMP parallel programming in the most CPU intensive parts of the present code to run higher numerical simulation times. OpenMP is a standard for shared-memory systems, which are the majority of modern computers. All the created threads have a private memory section, along with another portion that is shared among others threads. Since all the data is stored in a single machine, the data exchange between cores is faster than in distributed memory systems. Moreover, OpenMP's ease of use and short system setup time allows for a simple implementation.

The temperature particles method developed in this paper is based on a previously validated Lagrangian vortex method developed by Bimbato et al. (2011). It was used the algorithm developed by Alcântara Pereira & Hirata (2003) including the novel model that simulates the interaction between vorticity and temperature through of the creation of vorticity from heat.

The main objective of the work with the implementation of a temperature particles method for the analysis of unsteady and mixed-convective heat transfer in a flow around a stationary body has been achieved. The new methodology, therefore, is able to provide good estimates for Strouhal number, lift and drag coefficients, pressure distribution and time-averaged Nusselt number (Alcântara Pereira and Hirata, 2003), and is able to predict the flow correctly in a physical sense.

All the numerical simulations were carried out with a high value of the Reynolds number; no attempted to use a turbulence modeling was made. The sub-grid turbulence modeling (Alcântara Pereira et al., 2002) is of significant importance for the numerical simulation. The local turbulence effects can be taking into account through a second order velocity structure function model adapted to a Lagrangian vortex method (Alcântara Pereira et al., 2002). The results of this analysis, taking into account the sub-grid turbulence modeling, are also being generated and will be presented in due time, elsewhere. It is very important to observe that a two-dimensional surface roughness model is more effective than a single two-dimensional turbulence model (Bimbato, 2012).

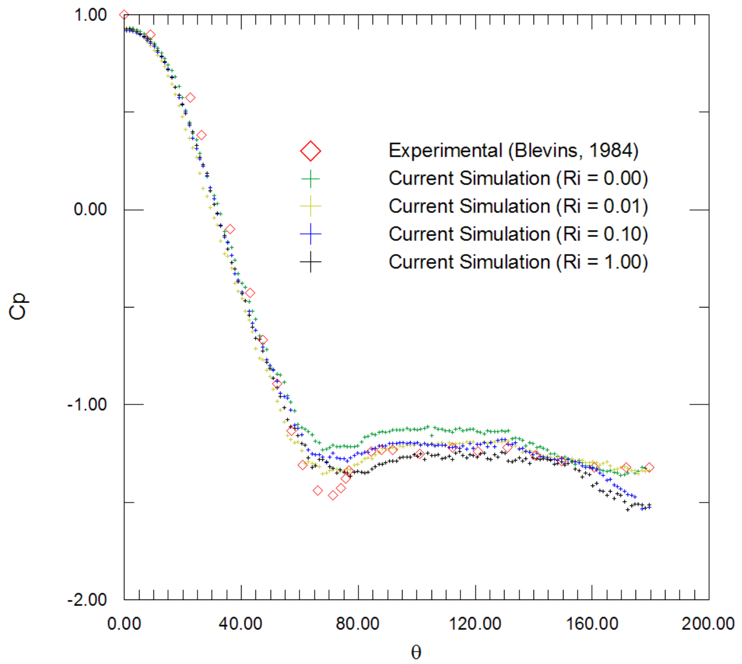


Figure 4. Time-averaged pressure coefficients for heated cylinder surface with $Re = 10^5$ and $Pr = 1.0$.

Future works will investigate the sensitivity of the methodology to the various simplifications used in creating an efficient scheme, and attempt to demonstrate a close connection to experimental predictions over a range of different flows. The focus will to analyze problems involving both Vortex Induced Vibrations (VIV) and mechanisms of ground effect with mixed-convective heat transfer. The transport of heat associated with aerodynamics of bluff bodies is of great importance in engineering. Many areas of fluid mechanics and heat transfer are involved in understanding this kind of flows. The experience gained with the present research added to the ones from previous one allows one to analyze complex situations where relative motions between bodies are present. These extend the applicability of the present Lagrangian-Lagrangian temperature particles method. The use of global as well local quantities combined to the near field flow pattern observations are useful to understand the complex mechanisms that lead the origin and the time evolution of the aerodynamic loads. The methodology developed in this paper is greatly simplified by the utilization of the Lagrangian manner.

5. ACKNOWLEDGEMENTS

This work was supported by FAPEMIG (Proc. APQ-02175-14).

6. REFERENCES

Achenbach, E., 1968, "Distribution of Local Pressure and Skin Friction around a Circular Cylinder in Cross-Flow up to $\mathrm{Re} = 5.0\times10^{6}$", J. Fluid Mech., 34(4): 625-639.

Alcântara Pereira, L.A., Ricci, J.E.R., Hirata, M.H. and Silveira-Neto, A., 2002, "Simulation of Vortex-Shedding Flow about a Circular Cylinder with Turbulence Modeling", Intern'l Society of CFD, Vol. 11, No. 3, October, pp. 315-322.

Alcântara Pereira, L.A. and Hirata, M.H. 2003. "Heat Transfer in the Wake Behind a Body Using A Particle Method". 17th International Congress of Mechanical Engineering, Proceedings of COBEM 2003, November 10-14, São Paulo, SP, Brazil.

Bimbato, A. M., Alcântara Pereira, L. A., Hirata, M. H., 2011. "Study of the vortex shedding flow around a body near a moving ground". Journal of Wind Engineering and Industrial Aerodynamics 99, 7-17.

Bimbato, A. M., 2012. "Study of Turbulent Flows around a Smooth or a Rough Bluff Body Using the Discrete Vortex Method". D.Sc. Thesis, Institute of Mechanical Engineering, UNIFEI, Itajubá, MG, Brazil (in Portuguese).

Chorin, A.J., 1973. "Numerical Study of Slightly Viscous Flow". Journal of Fluid Mechanics, Vol. 57, pp. 785-796.

Chorin, A.J., 1978. "Vortex Sheet Approximation of Boundary Layers". J. Comp. Physics, Vol. 27, pp. 428-442.

Ferziger, J.H., 1981, "Numerical Methods for Engineering Application", John Wiley & Sons, Inc.

Gerrard, J.H., 1966, "The Mechanics of the Formation Region of Vortices behind Bluff Bodies", J. Fluid Mech., 25: 401-413.

Ghoniem, A.F. and Sherman, F.S., 1985. "Grid-Free Simulation of Diffusion Using Randon Walk Methods". J. Comp. Physics, Vol. 61, pp. 1-37.

Igarashi, T., 1984. "Flow and Heat Transfer in the Separated Region around a Circular Cylinder" (In Japanese). Trans. JSME B 50-416, pp. 3008, 3014.

Kamemoto, K., 2004. "On Contribution of Advanced Vortex Element Methods Toward Virtual Reality of Unsteady Vortical Flows in the New Generation of CFD". Proceedings of the 10th Brazilian Congress of Thermal Sciences and Engineering-ENCIT 2004, Rio de Janeiro, Brazil, Nov. 29 - Dec. 03, Invited Lecture-CIT04-IL04.

Kamemoto, K. and Miyasaka, T., 1999. "Development of a Vortex and Heat Elements Method and Its Application to Analysis of Unstead Heat Transfer around a Circular Cylinder in a Uniform Flow". Proc. of 1st Int. Conf. on Vortex Metods, Kobe, Nov. 4-5, pp. 191-203.

Katz, J. and Plotkin, A., 1991. "Low Speed Aerodynamics: From Wing Theory to Panel Methods". McGraw Hill, Inc.

Kundu, P. K., 1990. "Fluid Mechanics". Academic Press.

Leonard, A., 1980. "Vortex Methods for Flow Simulations". J. Comp. Physics, Vol. 37, pp. 289-335.

Lewis, R.I., 1999. "Vortex Element Methods, the Most Natural Approach to Flow Simulation - A Review of Methodology with Applications". Proceedings of 1st Int. Conference on Vortex Methods, Kobe, Nov. 4-5, pp. 1-15.

Nakamura, H., Kamemoto, K. And Igarashi, T., 2001. "Analysis of Unsteady Heat Transfer in the Wake Behind a Circular Cylinder in a Uniform Flow by a Vortex and Heat Element Mehod". Proceedings of The Second International Conference on Vortex Methods, Sept. 26-28, Istanbul, Turkey.

Ogami, Y., 2001. "Simulation of Heat-Fluid Motion by the Vortex Method". JSME International Journal, Series B, Vol. 44, No. 4, pp. 513-519.

Ogami,Y. and Akamatsu,T., 1991. "Viscous Flow Simulation Using the Discrete Vortex Model -The Diffusion Velocity Method". Computers & Fluids, Vol 19, pp. 433-441.

Shintani, M. and Akamatsu, T, 1994. "Investigation of Two Dimensional Discrete Vortex Method with Viscous Diffusion Model". Computational Fluid Dynamics Journal, Vol. 3, No. 2, pp. 237-254.

Son, J.S. and Hanratty, T.J., 1969, "Velocity Gradients at the Wall for Flow around a Cylinder at Reynolds Number from $5{,}0\times10^{3}$ to $1{,}0\times10^{5}$", J. Fluid Mech., 35 (2): 353-368.

7. RESPONSIBILITY NOTICE

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