HEAT TRANSFER ANALYSIS AROUND AN OSCILLATING CIRCULAR CYLINDER IN THE PRESENCE OF A GROUND PLANE

Washington Humberto de Moura, whmoura@unifei.edu.br
Luiz Antonio Alcântara Pereira, luizantp@unifei.edu.br
Universidade Federal de Itajubá, Instituto de Engenharia Mecânica, CP 50, Av. BPS 1303, Itajubá, MG, CEP. 37500-903

Miguel Hiroo Hirata, hirata@fat.uerj.br
FAT/UERJ, Campus Regional de Resende, Estrada Resende–Riachuelo, Resende, RJ

Abstract. *In this paper the vortex cloud method was extended to take into account the convection and diffusion of heat transfer. Discrete heat particles were generated close to a circular cylinder surface oscillating with small amplitude in addition to nascent vortex elements in the presence of a ground plane. The influence of the frequency and amplitude oscillation on the aerodynamics loads and on the heat transfer are discussed and the preliminary numerical results are presented.*

Keywords*: vortex and heat element method, heat transfer, ground effect, small oscillation*

1. INTRODUCTION

The transference of heat from the surface of a bluff body to the surrounding fluid is very important in the design of a device such as a cooling system for an electronic component or a heat engine. This is the process which cools or heats the body. There is a second process by which the fluid is mixed downstream of the body – this is the convective transport of heat in the wake of a bluff body – being the most important consideration in the design of a device such as a flame holder or a mixed.

Viscous flow around circular cylinders has been studied by several authors during the last years not only to understand the fundamentals of general bluff body flows but also because this flow configuration itself is of direct relevance to many practical applications (e.g., tall buildings, bridge piers, chimneys, periscopes, heat exchangers tubes, cables, wires, and so on). In fact, the flow around cylinders includes a variety of fluid dynamics phenomena, such as separation, vortex-shedding and the transition to turbulence. The most essential parameter which characterizes the wake of a smooth circular cylinder is the Reynolds number, Re.

Thus by varying the Reynolds number, a rich set of flow is observed, but even more wake patterns are observed by forced oscillation of a cylinder. Under forced transverse oscillations, various vortex shedding patterns are possible by varying the oscillation frequency and amplitude (Williamson and Roshko, 1998).

On the other hand, the influence of the boundary layer formed on the ground is much more complicated and is still unclear despite several intensive studies reported so far. Roshko *et al.* (1975) measured the time-averaged drag and lift coefficients, C_D and C_L, for a circular cylinder placed near a fixed wall in a wind tunnel at $Re=2.0\times10^4$, which lies in the upper-subcritical flow regime, and showed that the C_D rapidly decreased and C_L increased as the cylinder came close to the wall. Zdravkovich (1985) observed, in his force measurements performed at $4.8\times10^4 < Re < 3.0\times10^5$, that the rapid decrease in drag occurred as the gap was reduced to less than the tickness of the boundary layer δ/d on the ground, and concluded that the variation of C_D was dominated by h/δ rather than by the conventional gap ratio h/d. He also noted that the C_L could be significantly affected by the state of the boundary layer, although it was insensitive to the tickness of the boundary layer.

Recently, Nishino *et al.* (2007) presented experimental results of a circular cylinder with an aspect ratio of 8.33, with and without end-plates, placed near and parallel to a moving ground, on which substantially no boundary layer developed to interface with the cylinder. The main purpose of experiments was to elucidate the fundamental mechanisms of ground effect in more details by using a moving ground running at the same speed as the freestream and thereby eliminating the confusing effects of boundary layer formed on the ground. Measurements were carried out at two upper-subcritical Reynolds numbers of 0.4 and 1.0×10^5. The results produced new insights into the physics of ground effect, and could serve as a database for both experimental and computational studies on the ground effect in the future. According to Nishino *et al.* (2007) experiments, for the cylinder with end-plates, on which the oil flow patterns were observed to be essentially two-dimensional, the drag rapidly decrease as h/d decrease to less than 1.0 but became constant for h/d of less than 0.85, unlike that usually observed near a fixed ground

The aim of this paper is to investigate numerically the mechanisms of the heat transport from an oscillating heated cylinder in ground effect using vortex method.

The vortex methods have been developed and applied for analysis of complex, unsteady and vortical flows in relation to problems in a wide range of industries, because they consist of simple algorithm based on physics of flow (Kamemoto, 2004). The essentials of vortex methods are presented by Chorin (1973), Leonard (1980), Sarpkaya (1989), Lewis (1999), Alcântara Pereira *et al.* (2002) and Stock (2007). The vortex methods offer a number of advantages over the more traditional Eulerian schemes: (i) computational elements only where vorticity is non-zero; (ii) no grid in the field; (iii) only 2D grid on vehicle surface; (iv) boundary conditions in the far field automatically satisfied.

In order to handle fluid flow and heat transfer by the vortex method, the following models are required (Ogami, 2001): (i) the modeling of discretization of heat distribution into the heat particles; (ii) the modeling of the process in which the vortex is generated by the effect of a heat; (iii) the modeling of the diffusion process of heat and vortex. Alcântara Pereira and Hirata (2003) extended the vortex method to take into account the convection and diffusion heat transfer. Discrete heat particles were generated close to a circular cylinder surface in addition to nascent vortex elements. The unsteady flow and heat transfer were simulated around a circular cylinder in a uniform flow. The result of the surface and time-averaged Nusselt number at constant surface temperature showed reasonable agreement with that of experiment. The generation of vortex due the heat was not analyzed.

Recently, Moura *et al.* (2007) analyzed the flow around an oscillating cylinder in ground effect, which moved with constant velocity. The amplitude of the oscillatory motion was considered to be small when compared with the body length, as the first approximation one is allowed to transfer the body boundary condition from the actual position to a mean position of the body surface. The impermeability condition was imposed through the application of a source panel's method. Lamb vortices were generated along the surfaces, whose strengths are determined to ensure that the no-slip condition is satisfied and that the circulation is conserved. The aerodynamics loads were computed using an integral formulation derived from the pressure Poisson equation. The analysis of oscillation effect on the mechanism of circular cylinder lift generation was presented.

Heat transfer from transversely oscillating cylinder in cross-flow was first studied by Sreenivassan and Ramachandram (1961) over the range 2500 < Re < 15000 in air. This is likely because the oscillation frequencies used in their experiments were much lower than the natural shedding frequency. Though they considered amplitude ratios up to 1.8, the largest value of f* in their study was 0.07, which is only about 1/3 of the natural shedding frequency. Later experiments have found that heat transfer from a cylinder is enhanced by transverse oscillations near the Strouhal frequency (Kezios and Prasanna, 1966).

In this paper, the vortex method (Moura *et al.*, 2006) is employed to simulate numerically the unsteady flow and heat transfer (Alcântara Pereira and Hirata, 2003) around and oscillating circular cylinder in ground effect which moves with constant velocity in a quiescent Newtonian fluid. The two-dimensional aerodynamics characteristics are investigated at a Reynolds number of 1.0×10^5. The main purpose of present numerical study is the investigation of the relationship between near-wake structure and heat transfer for small amplitude (Moura *et al.*, 2006).

The authors of this paper has developed the present vortex method to simulate the macro scale phenomena, therefore the smaller scale ones are taken into account through the use of a second order velocity function (Alcântara Pereira et *al*., 2002). In the present paper, the effects of small scale are not still considered.

2. GENERAL FORMULATION AND NUMERICAL METHOD

"Figure 1" describes a standard layout of the circular cylinder placed near the ground. Herein we consider the incompressible flow of a Newtonian fluid around a moving circular cylinder in an unbounded two-dimensional region. Also for all flows considered temperature variation has not a negligible impact on the flow field. The incident flow IS defined by free stream speed U with constant temperature T_∞ and the domain Ω with boundary $S = S_1 \cup S_2 \cup S_3$, S_1 being the body surface at constant temperature T_w, S_2 the plane surface and S_3 the far away boundary. The cylinder moves to the left with constant velocity; an oscillatory motion with small amplitude A and constant angular velocity λ is added to body motion. In this figure the (x,o,y) is the inertial frame of reference and the (η,O,ξ) is the coordinate system fixed to the cylinder; this coordinate system oscillates around the x-axis as $y_0=A\cos(\lambda t)$.

The evolution of such a fluid is governed by the following relations for conservation of mass, momentum and energy respectively

$$\mathrm{div}\mathbf{u} = 0 \tag{1}$$

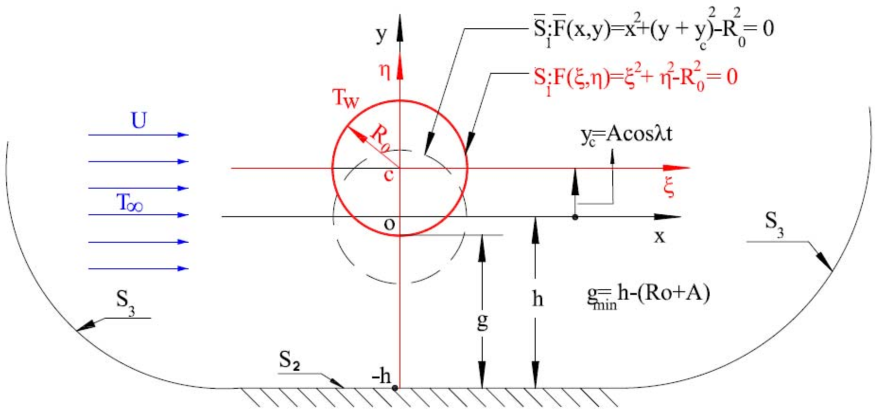
$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u}\cdot\nabla)\mathbf{u} + \nabla p = \upsilon\nabla^2\mathbf{u} + \mathbf{f} \tag{2}$$

$$\frac{\partial T}{\partial t} + (\mathbf{u}\cdot\nabla)T = \frac{k}{\rho c_p}\nabla^2 T\ . \tag{3}$$

In the equations above $\mathbf{u}$ is the velocity vector field, p is the pressure, $\mathbf{f}$ is body force, υ is the fluid kinematics viscosity coefficient, k is the thermal conductivity and ρc_p is the volumetric heat capacity ($k/\rho c_p$ is the thermal diffusivity). "Equation (3)" is according Boussinesq's approximation.

In this formulation it is necessary to solve for both the pressure, temperature and two velocity components in order to evolve the flow. This can complicate computational approaches and it proves favorable to consider the evolution equation for the curl of the velocity, the vorticity, instead

$$\boldsymbol{\omega} = \nabla \times \mathbf{u} \,. \tag{4}$$



Figue1. Circular cylinder in ground effect and the coordinate systems

For two-dimensional flow, the vorticity vector has only one component and thus can be expressed as a scalar field ω(x,y).

Taking the curl of "Eq. (2)" and applying "Eq. (1)" and the two-dimensionality constraint yields

$$\frac{\partial \omega}{\partial t} + (\mathbf{u} \cdot \nabla)\omega = \upsilon \nabla^2 \omega + \nabla \times \mathbf{f} \tag{5}$$

which is the evolution of the scalar vorticity field.

In order to determine a flow from the vorticity equation, "Eq. (5)", it is necessary to find a velocity field in terms of the vorticity field. To do so, consider decomposing the velocity vector into two fields

$$\mathbf{u} = \nabla \times \boldsymbol{\psi} + \nabla \phi \tag{6}$$

which $\boldsymbol{\psi}$ is the vector streamfunction and ϕ is the velocity potential. Now consider the curl of "Eq. (6)"

$$\boldsymbol{\omega} = \nabla \times \nabla \times \boldsymbol{\psi} + \nabla \times \nabla \phi = -\nabla^2 \boldsymbol{\psi} + \nabla(\nabla \cdot \boldsymbol{\psi}) \,. \tag{7}$$

Note that the velocity potential serves to include components of the velocity field which cannot be represented in the vorticity field. For two-dimensional flow, "Eq. (7)" simplifies to

$$\omega = -\nabla^2 \psi \,. \tag{8}$$

The Green's function for the two-dimensional Laplacian can be now convolved with the stream-function to give a vorticity-based representation of the streamfunction as follows

$$\psi(\mathbf{x}) = -\frac{1}{2\pi} \iint \ln\left|\mathbf{x} - \mathbf{x}'\right| \omega(\mathbf{x}')d\mathbf{x}' \,. \tag{9}$$

Now "Eq. (9)" can be used in "Eq. (6)" to yield the relation commonly known as the Biot-Savart law

$$\mathbf{u}(\mathbf{x}) = -\frac{1}{2\pi} \iint \frac{(\mathbf{x} - \mathbf{x}') \times \omega(\mathbf{x}')\hat{\mathbf{z}}}{\left|\mathbf{x} - \mathbf{x}'\right|^2} d\mathbf{x}' + \nabla \phi \,. \tag{10}$$

The foundation of vortex methods rests on the use of "Eq. (5)" and "Eq. (10)" to track a fluid based on the evolution of this vorticity field. The vorticity equation is free of the computational instabilities associated with the convective term. In the inviscid case without body forces and frame of reference acceleration

$$\frac{D\omega}{Dt} = 0 . \tag{11}$$

Computational simulation requires the discretization in space and time of "Eq. (5)" and "Eq. (10)". Particles strengths remain constant to satisfy "Eq. (11)", so

$$\frac{dx_i}{dt} = u(x_i) \text{ and } \frac{d\Gamma}{dt} = 0 , \tag{12}$$

where the amount of vorticity carried by a given particle is termed its circulation and represented by Γ.

The vorticity convection is governed by "Eq. (11)" and the velocity field is given by (Recicar *et al.*, 2006)

$$\mathbf{u}(\mathbf{x}, t) = \mathbf{ui}(\mathbf{x}, t) + \mathbf{ub}(\mathbf{x}, t) + \mathbf{uv}(\mathbf{x}, t) \tag{13}$$

where **ui** is the velocity vector of the incident flow, **ub** is the velocity vector induced by the body and plane surfaces and **uv** is the velocity vector induced due to the vortex cloud.

Note that "Eq. (3)" gives the law that the temperature distribution, T, moves both with the convection velocity. Vortex elements and discrete heat elements distributed in the flow field are followed during numerical simulation according to the first order Euler scheme. It is clear that the energy equation, "Eq. (3)", has the similar form to the vorticity transport equation, "Eq. (5)". This suggests that the energy equation can be solved in an analogous way using the random walk method to the motion of the heat elements to account for diffusion (Chorin, 1973).

In this paper, the temperature T_w is considered constant around the body surface, see "Fig. 1". The heat transport from the body surface to the fluid nearby the body surface is determined by the temperature gradient at the surface. The effect of buoyancy is not considered here because our study is focused on the forced convection heat transfer (Alcântara Pereira and Hirata, 2003). The surface heat flux is determined by Fourier's Law

$$\dot{q} = -\lambda \frac{dT}{dn} \tag{14}$$

where n denotes the normal direction to the surface and λ is the thermal conductivity of fluid. The heat quantity transfered from the surface (j-th panel with length ΔS_j) to the k-th nascent heat element is given by

$$\Delta Q_j = \alpha \Delta t \frac{\left(T_W - T_j\right)}{\varepsilon} \Delta S_j \tag{15}$$

in which $\alpha = \upsilon / Pr$ (Pr is Prandtl number) and ε is the displacement normal to the straight-line panel.

The temperature distribution $T(z)$ results from the contribution of all the heat particles in the field

$$T(z) = \sum_j \frac{\Delta Q_j}{\pi \sigma_T^2} \exp\left[-\frac{\left(z - z_j\right)^2}{\sigma_T^2} \right] \tag{16}$$

where σ_T is the core radius of the heat particles.

The pressure calculation starts with the Bernoulli function, defined by Uhlman (1992) as

$$Y = p + \frac{u^2}{2} , \; u = |\mathbf{u}| . \tag{17}$$

Kamemoto (1993) used the same function and starting from the Navier-Stokes equations was able to write a Poisson equation for the pressure. This equation was solved using a finite difference scheme. Here the same Poisson equation was derived and its solution was obtained through the following integral formulation (Shintani and Akamatsu, 1994)

$$H\overline{Y_i} - \int_{S_1} \overline{Y} \nabla G_i \cdot \mathbf{e}_n dS = \iint_{\Omega} \nabla G_i \cdot (\mathbf{u} \times \boldsymbol{\omega}) d\Omega - \upsilon \int_{S_1} (\nabla G_i \times \boldsymbol{\omega}) \cdot \mathbf{e}_n dS \tag{18}$$

where H is 1.0 inside the flow (at domain Ω) and is 0.5 on the boundary S_1. $G_i = (1/2\pi)\log R^{-1}$ is the fundamental solution of Laplace equation, R being the distance from ith 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. (18)" is solved by mean of a set of simultaneous equations for pressure Y_i.

4. RESULTS AND DISCUSSIONS

When the Vortex Method is applied to heat-fluid motion, it becomes evident that the procedure is very sensitive to the numerical parameters involved. The main influences are: the non-dimensional time step, $t^*=tU/d$ (where d is the cylinder diameter); distance of release particles from surface; particles blob radius and time increment.

The numerical simulations were restricted to the simple situation of a circular cylinder in ground effect. In the calculations, each boundary S_1 and S_2 of "Fig. 1" was represented by fifty (M=50) source panels, and the time step size and Reynolds number were taken as $\Delta t=0.05$ and $Re=1.0\times10^5$ respectively. In each time step the nascent vortex and heat elements were placed into the cloud through displacements $\varepsilon=\sigma_0 = 0.0009d$ and $\varepsilon=\sigma_T= 0.09d$, respectively, normal to the panels. The aerodynamics forces computations starts at t=10.

Using the solution technique described in the previous section, two representative cases will be analyzed. First, we remove the oscillation of the circular cylinder to the ground h/d=1000. The result of the first case is presented in "Table 1". In this table one can find also experimental, Blevins (1984) (with 10% uncertainty) and numerical results Mustto *et al.* (1998). The numerical results of Mustto *et al.* (1998) were also obtained using the Vortex Method with the Circle Theorem (Milne-Thompson, 1955), while the Panels Method, using straight-line vortex panels with constant density, was used in the results of Alcântara Pereira *et al.* (2002). The agreement between the two numerical methods is very good for the Strouhal number, and both results are close to the experimental values. One should observe, however, that three-dimensional effects are non-negligible for the Reynolds number used in the simulations. Therefore one can expect that a two-dimensional computation of such a flow must produce higher values for the drag coefficient. On the other hand, the Strouhal number is insensitive to these three-dimensional effects.

Table 1: Strouhal number, lift and drag coefficients for a circular cylinder with h/d=1000.

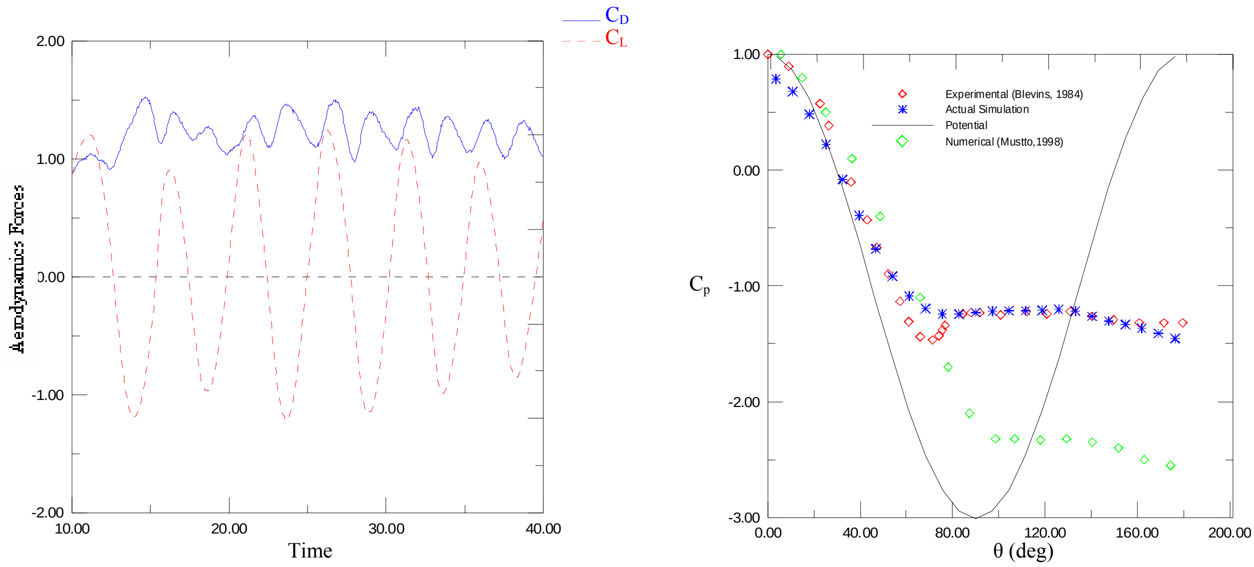
$Re = 10^5$, A = 0 and $\lambda = 0$	$\overline{C}_D$	$\overline{C}_L$	$\overline{S}_t$
Blevins (1984)	1.20	-	0.19
Mustto *et al.* (1998)	1.22	-	0.22
Present simulation	1.20	0.06	0.19

"Figure 2(a)" shows time histories of drag coefficient, C_D and lift coefficient, C_L, of the circular cylinder. Both values of drag and lift are defined by the sum of pressure forces. "Figure 2(a)" indicate that the fluctuation of C_D have double frequency that of C_L, because it fluctuate once for each of upper and lower shedding. "Figure 2(a)" shows that the lift coefficient oscillates around zero. The differences encountered in the comparison of the numerical result with the expected result are attributed mainly the inherent three-dimensionality of the real flow for such a value of the Reynolds number, which is not modeled in the present simulation.

"Figure 2(b)" shows the mean value of pressure coefficient around the discretized circular cylinder surface. The present result is compared with others results available in the literature and current simulation agree very well with the experimental ones. From the simulation the predict separation points occur around 86°, while the experimental value is around 82°. In other experimental investigation by Son and Hanratty (1969); determined a value of 78° for the separation angle.

Some discrepancies observed in the determination of the aerodynamics loads may be attributed to errors in the treatment of vortex element moving away from a solid surface. Because every vortex element has different strength of vorticity, it will diffuse to different location in the flow field. It seems impossible that every vortex element will move to same ε-layer normal to the solid surface (Yang and Huang, 1999). In the present method all nascent vortices were placed into the cloud through a displacement $\varepsilon=\sigma_0=0.0009d$ normal to the panels.

Numerical simulation for flow and heat transfer was performed around a circular cylinder at $Re= 1.0\times10^5$, Pr= 0.71 and h/d=0.7. A constant temperature $T_w= 363$ K on the body surface as a boundary condition, and the freestream temperature was set to $T_\infty= 293$ K.



(a) C_D and C_L (b) time averaged pressure coefficient

Figure 2: Time histories of aerodynamics loads, for A = 0, ω = 0, h/d=1000 and Re=10^5.

"Figure 3" shows distribution of vortex elements and heat elements at t= 19.9. A vortex sheet is formed behind the circular cylinder (blue points). There are several situations where this type of vortex shedding may cease, and one of them is that a cylinder is located near a plane boundary or ground. The red points in "Fig. 3" represent boundary layer developing on the ground. Nishino *et al.* (2007) reported that the mechanism of the ground effect are still far from being fully understood due to a variety of other influence factors, in particular the confusing influence of the boundary layer formed on the ground. In this paper we introduce two factors: (i) forced convection heat transfer and (ii) small oscillation.

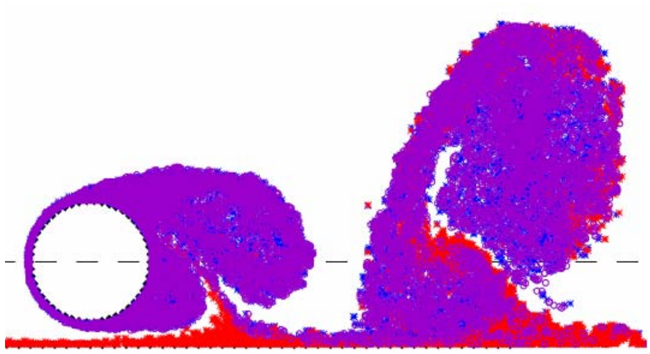


Figure 3. Flow pattern around a circular cylinder, for A = 0.05, λ= 1.3, h/d=0.75, Pr=0.71 and Re=10^5 at t= 19.9.

Pottebaum (2003) previously observed that heat transfer enhancement associated with oscillations at frequencies near the natural shedding frequency and its harmonics were shown to be limited to amplitudes of less than about 0.5 cylinders diameters. Obviously the ground effect has influence at enhancing heat transfer. "Figure 4" shows the temperature distribution at t= 19.9.

From the study of the heat convection from a cylinder as function of forced oscillation frequency and amplitudes, we need to investigate the relation between the incensement of natural vortex shedding frequency and heat transfer in ground effect.

Pottebaum (2003) founded that wake structure and heat transfer both significantly affect one another. The wake mode, a label indicating the number and type of vortices shed in each oscillation period, is directly related to the observed heat transfer enhancement. The dynamics of the vortex formation process, including the trajectories of the vortices during roll-up, explain this relationship.

According to the Pottebaum (2003), the cylinder's transverse velocity was shown to influence the heat transfer by affecting the circulation of the wake vortices. For a fixed wake structure, the effectiveness of the wake vortices at enhancing heat transfer depends on their circulation. Also, the cylinder's transverse velocity continually changes the orientation of the wake with respect to the freestream flow, thereby spreading the main source of heat transfer enhancement – the vortices near the cylinders base – over a large portion of the cylinder surface.

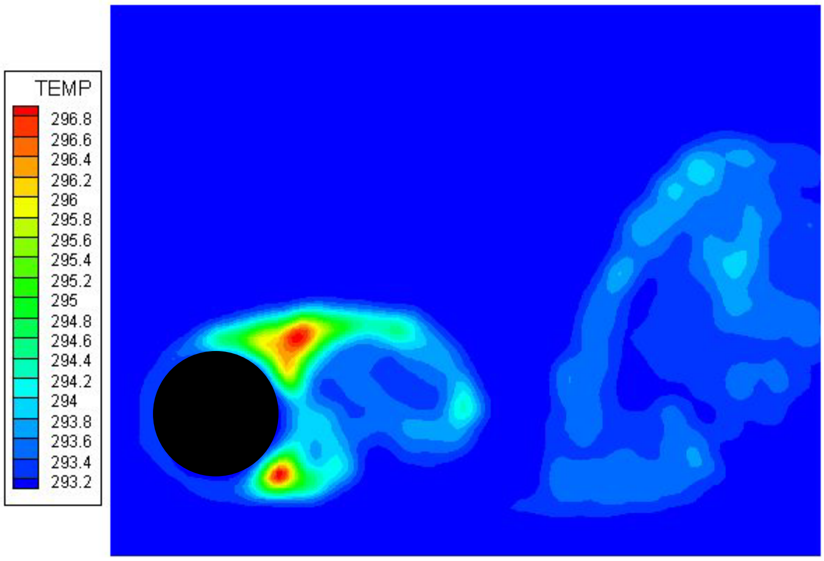


Figure 4. Circular cylinder: temperature distribution (when A = 0.05, λ= 1.3, h/d=0.75, Pr=0.71 and Re=10^5) at t=19.9.

As future work we need to investigate the relationship between the fundamentals effects of h/d and heat transfer for an oscillating circular cylinder in cross-flow. The influence of the boundary layer formed on the ground is much more complicated and is still unclear despite several intensive studies reported so far. Recently, Zdravkovich (2003) reported the drag behavior for cylinder placed near a moving ground running at the same speed as the freestream for higher Reynolds number of 2.5×10^5, which lies within the critical flow regime rather than the subcritical flow regime. The experiment by Zdravkovich (2003) showed contrast to all the above studies. First, practically no boundary layer on the ground. Second, the decrease in drag due the decrease in h/d did not occur in his measurements. The differences encountered were attributed to the non-existence of the wall boundary layer or the higher Reynolds number, or any other influencing factors. Because the distributed vorticity and heat of the mainstream flow has been replaced in the numerical model by two clouds of particles, the CPU time for particle-particle interaction turns expensive. No attempts to simulate the flow for M greater than 50 were made since the operation count of our algorithm is proportional to the square of N. As M increases N also tends to increase, and the computational efforts becomes expensive. This is a major source of difficulties, and it can only be handled through the utilization of faster schemes for the induced velocity calculations, such as the multipole technique (Greengard and Rokhlin, 1987) and/or parallel computers to run long simulations (Takeda *et al.,* 1999).

The present 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. As a future work, the effect of buoyancy will be carried out. A new method to simulated diffusion will be carried out (Rossi, 2006). Finally, the results are promising and encourage performing additional tests in order to explore the phenomena in more details.

5. ACKNOWLEDGEMENTS

The authors would like to acknowledge FAPEMIG (Proc. TEC-748/04) and FAPERJ (Proc. E26/170403/07) for the financial support during the time of this project.

6. REFERENCES

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.

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.

Blevins, R.D., 1984, "Applied fluid dynamics hanbook", Van Nostrand Reinhold Co.

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

Greengard, L. and Rokhlin, V., 1987, "A Fast Algorithm for Particle Simulations", Journal of Computational Physics, Vol. 73, pp. 325-348.
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., 1993, "Procedure to Estimate Unstead Pressure Distribution for Vortex Method" (In Japanese), Trans. Jpn. Soc. Mech. Eng., Vol. 59, No. 568 B, pp. 3708-3713.
Kezios, S.P. and Prasanna, K.V., 1966, "Effect of Vibration on Heat Transfer from a Cylinder in Normal Flow", ASME Paper No. 66-WA/HT-43.
Leonard, A., 1980, "Vortex Methods for Flow Simulations", J. Comput. Phys., 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.
Milne-Thompson, L.M., 1955, Theorical Hydrodynamics; MacMillan & Co.
Moura, W.H., Alcântara Pereira, L.A. and Hirata, M.H., 2006, "Analysis of the Flow around an Oscillating Circular Cylinder in Ground Effect", 11th Brazilian Congress of Thermal Sciences and Engineering, Proceedings of ENCIT 2006, December 5-8, Curitiba, Brazil.
Mustto, A.A., Hirata, M.H. and Bodstein, G.C.R., 1998, "Discrete Vortex Method Simulation of the Flow Around a Circular Cylinder with and without Rotation", A.I.A.A. Paper 98-2409, Proceedings of the 16th A.I.A.A. Applied Aerodynamics Conference, Albuquerque, NM, USA, June.
Nishino, T., Roberts, G.T. and Zhang, X., 2007, "Vortex Shedding from a Circular Cylinder near a Moving Ground", Physics of Fluids, Vol. 19, 025103.
Ogami, Y., 2001, "Simulation of Heat-Fluid Motion by the Vortex Method", J.S.M.E. International Journal, Series B, Vol. 44, No. 4, pp. 513-519.
Pottebaum, T.S., 2003, "The Relationship between Near-Wake Structure and Heat Transfer for an Oscillating Circular Cylinder in Cross-Flow", PhD Thesis, California Institute of Technology.
Roshko, A., Steinolfson, A. and Chattoorgoon, V., 1975, "Flow Forces on a Cylinder near a Wall or near Another Cylinder", Proceedings of the 2nd US Nation Conference on Wind Engineering Research, Fort Collins, Paper IV-15.
Recicar, J.N., Alcântara Pereira, L.A. and Hirata, M.H., 2006, "Harmonic Oscillations of a Circular Cylinder Moving with Constant Velocity in a Quiescent Fluid", 11th Brazilian Congress of Thermal Sciences and Engineering, Proceedings of ENCIT 2006, December 5-8, Curitiba, Brazil.
Rossi, L.F., 2006, "A Comparative Study of Lagrangian Methods Using Axisymmetric and Deforming Blobs", SIAM J. Sci. Comput., Vol. 27, No. 4, pp. 1168-1180.
Sarpkaya, T., 1989, "Computational Methods with Vortices - The 1988 Freeman Scholar Lecture", Journal of Fluids Engineering, Vol. 111, pp. 5-52.
Sarpkaya, T., 1989, "Computational Methods with Vortices - The 1988 Freeman Scholar Lecture", Journal of Fluids Engineering, Vol. 111, pp. 5-52.
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.
Sreenivasan, K. and Ramachandram, A., 1961, "Effect of Vibration on Heat Transfer from a Horizontal Cylinder to a Normal Air Stream", International Journal of Heat ans Mass Transfer, Vol. 3, pp. 60-67.
Son, J.S., Hanratty, 1969, "Velocity Gradients at the Wall for Flow Around a Cylinder at Reynolds Number from $5x10^3$ to 10^5"; Journal of Fluid Mechanics, Vol. 35, no. 2, pp. 353-368.
Stock, M.J., 2007, "Summary of Vortex Methods Literature (A lifting document rife with opnion)", April 18, 2007.
Takeda, K., Tutty, O.R. and Nicole, D.A., 1999, "Parallel Discrete Vortex Method on Commodity Supercomputers; an Investigation into Bluff Body Far Wake Behaviour", ESAIM: Proceedings, Third International Workshop on Vortex Flows and Related Numerical Methods, Vol. 7, pp. 418-428.
Uhlman, J.S., 1992, "An Integral Equation Formulation of the Equation of an Incompressible Fluid", Naval Undersea Warfare Center, T.R. 10-086.
Williamson, C.H.K. and Roshko, A., 1988, "Vortex Formation in the Wake of an Oscillating Cylinder", Journal of Fluids and Structures, Vol. 2, pp. 355-381.
Yang, R.J. and Huang, Y.G., 1999, "Simulation of High Reynolds Flow Over a Flat Plate by a Deterministic Vortex Method", Comput. Fluid Dynamics Journal, Vol. 8, No. 2.
Zdravkovich, M.M., 2003, "Flow around Circular Cylinders" Vol. 2: Applications, Oxford University Press, Oxford, UK.
Zdravkovich, M.M., 1985, "Forces on a Circular Cylinder near a Plane Wall", Applied Ocean Research, Vol. 7, pp. 197-201.

7. RESPONSIBILITY NOTICE

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