SIMULATION OF VORTEX–SHEDDING FLOW AROUND TWO OSCILLATING SQUARE CYLINDERS IN A TANDEM ARRANGEMENT

Luiz Antonio Alcântara Pereira
Instituto de Engenharia Mecânica, UNIFEI, CP. 50, Itajubá, Minas Gerais, 37500-903, Brasil
luizantp@unifei.edu.br

Miguel Hiroo Hirata
FAT/UERJ, Campus Regional de Resende, Estrada Resende-Riachuelo, Resende, Rio de Janeiro, Brasil
hirata@fat.uerj.br

Abstract. *Vortex shedding over two oscillating square cylinders in a tandem arrangement is modeled using a lagrangian mesh-free vortex method. Lamb vortices are generated along the cylinders surface, whose strengths are determined to ensure that the no-slip condition is satisfied and that circulation is conserved. The impermeability condition is imposed through the application of a source panels method. The amplitude of the oscillatory motion is considered to be small compared to the square cylinder length, therefore, to the first approximation, one is allowed to transfer the body boundary condition from the actual position to the mean position of the body surface. The changes in the aerodynamics loads due the oscillatory motion are analyzed using an integral formulation derived from the pressure Poisson equation.*

Keywords*: vortex method, panels method, small-amplitude oscillations, interference, aerodynamics loads.*

1. Introduction

The investigation of flow around bluff bodies is a complex problem of great practical importance because of the high aerodynamics loads, which can result on neighbouring structures (Hussain and Lee, 1980). Investigation of the characteristics of flow around simple configurations of objects is helpful for understanding the flow around more complex and larger-scale structures. In many cases of engineering practices, objects often appear in the form of groups, e.g. groups of buildings, chimneys, stacks, chemical reaction-towers, supports of off-shore platform, etc.

In the literature buffeting refers to the unsteady aerodynamics forces that result from the interference of a second body situated in the wake of the first (Havel *et al.,* 2001). A two-dimensional obstacle is one of sufficient span to allow neglect of ends. Traditionally they are modeled as prisms or cylinders immersed in a uniform oncoming stream. The main characteristic of these flows is due the fact that the large-scale vortical structures rotate about a spanwise axis.

Most of the previous research on 2D geometries has concentrated on circular cylinders. Zdravkovich (1977) and Ohya *et al.* (1989) presented an extensive review of the state of knowledge of flow across two cylinders in various arrangements. Previous investigations of tandem configurations by Biermann and Herrnstein (1933), Kostic and Oka (1972), Novak (1974), Zdravkovich and Pridden (1975, 1977), Okajima (1979), Igarashi (1981, 1984), Hiwada *et al.* (1982), Arie *et al.* (1983), Jendrzejczyk and Chen (1986) have revealed considerable complexity in fluid dynamics as the spacing or gap between the cylinders is changed.

The interference phenomena are highly non-linear and there are many discrepant points in previous works. Arie *et al.* (1983) pointed out that fluctuation in drag force acting both cylinders is weakly dependent on spacing. On the other hand, Igarashi (1981) reported that the fluctuation in pressure associated with fluctuation in aerodynamics forces (lift and drag) acting on a downstream cylinder is strongly dependent on gap between the cylinders. Alam *et al.* (2003) presented an experimental study in which fluctuating lift and drag forces acting on the cylinders was measured. In their work they elucidated the discrepant points and clarified the flow patterns over the cylinders.

Based on surface pressure measurements and flow visualizations, Hangan and Vickery (1999) identify five general buffeting regimes for various upstream to downstream body dimensions: (1) close spacing with insignificant gap flow, (2) intermittent reattachment of upstream shear layer on the downstream cylinder, (3) a synchronized shedding regime, (4) quasi-isolated regime with two vortex shedding frequencies corresponding the two wake flows with a little interferences, (5) isolated without interference. These regimes are qualitatively similar to those found by Zhang and Melbourne (1992) for circular cylinders.

The Vortex Method 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).

Vortex cloud modeling offers great potential for numerical analysis of important problems in fluid mechanics. A cloud of free vortices is used in order to simulate the vorticity, which is generated on the body surface and develops into the boundary layer and the viscous wake. Each individual free vortex of the cloud is followed during the numerical simulation in a typical Lagrangian scheme. This is in essence the foundations of the Vortex Method (Chorin, 1973; Sarpakaya, 1989; Sethian, 1991; Lewis, 1999, Ogami, 2001; Alcântara Pereira *et al.*, 2002 and Kamemoto, 2004). Vortex Method offers a number of advantages over the more traditional Eulerian schemes: (a) the absence of a mesh avoids stability problems of explicit schemes and mesh refinement problems in regions of high rates of strain; (b) the

Lagrangian description eliminates the need to explicitly treat convective derivatives; (c) all the calculation is restricted to the rotational flow regions and no explicit choice of the outer boundaries is needed a priori; (d) no boundary condition is required at the downstream end of the flow domain.

For the grid methods, such as finite difference method and finite element method, the governing Navier-Stokes equations are solved directly. However, the flow around cylinder arrays is usually computed at Reynolds number (Re) up to a few hundred (Fornberg, 1985 and Jackson, 1987) while the Re for flows around cylinders in many engineering applications is of much higher order O (10^6). In such circumstance, the traditional Eulerian schemes will not give a satisfactory prediction within a reasonable computational cost. Also, the pre-processing and mesh-generation are time-consuming for the grid method in numerical simulations.

Alcântara Pereira and Hirata (2006) employed the Vortex Method to simulate the interference effects for a group of finite cylinders at Re=$1.3\text{x}10^4$. The interference phenomena are highly non-linear and at present beyond a reliable theoretical or numerical analysis. The main feature of their vortex code is to simulate numerically the two-dimensional, incompressible, unsteady flow around of pipe clusters: (a) two pipes, three-pipes clusters, (c) regular square multiple clusters, (d) and irregular multi-pipe clusters. The Vortex Method was used to simulate the macro scale phenomena, however, the effect of small scale was not considered.

Oscillatory motions of small amplitude are important in the analysis of immersed vibrating bodies and special care should be taken in the lock-in condition. Large amplitude motions, on the other hand, are of relevance in the analysis of bodies located in waves and currents such as the ones found in the offshore structures (Williamson and Roshko, 1988).

The oscillatory motion of small amplitude mainly modifies the near field changing the boundary layer flow and, as a consequence, having an important effect on the aerodynamic forces and the pressure distribution. If the amplitude of the oscillatory motion is large one observes, additionally, substantial changes in the far field wake which can be of importance in the presence of other bodies or near by surfaces.

In the present paper, the Vortex Method is employed to simulate the vortex-shedding flow from two oscillating square cylinders in a tandem arrangement. The amplitude of the oscillatory motion is considered to be small compared to the body length; therefore, to the first approximation one is allowed to transfer the body boundary condition from the actual position to a mean position of the body surface (Moura *et al.,* 2006).

The present Vortex Method has been used 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 this present approach, the effect of small scale is not considered.

2. Formulation of the physical problem

Consider the incompressible fluid flow of a Newtonian fluid around two oscillating square cylinders in a tandem arrangement an unbounded two-dimensional region. Figure 1 shows the incident flow, defined by free stream speed U and the domain Ω with boundary $S = S_1 \cup S_2 \cup S_\infty$; being S_1 and S_2 the cylinders surface and S_∞ the far away boundary.

Figure 1. Flow around two square cylinders in a tandem arrangement.

The viscous and incompressible fluid flow is governed by the continuity and the Navier-Stokes equations, which can be written in the form

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

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla p + \frac{1}{Re} \nabla^2 \mathbf{u} . \qquad (2)$$

In the equations above $\mathbf{u}$ is the velocity vector field and p is the pressure. As can be seen the equations are non-dimensionalized in terms of U and b (cylinder diameter). The Reynolds number is defined by

$$Re = \frac{bU}{\upsilon} , \qquad (2a)$$

where υ is the fluid kinematics viscosity coefficient; the dimensionless time is b / U .

The impermeability and no-slip conditions on the two square cylinders surface are written as

$$u_n = \mathbf{u} \cdot \mathbf{e}_n = 0 \qquad (3a)$$

$$u_\tau = \mathbf{u} \cdot \mathbf{e}_\tau = 0 \qquad (3b)$$

$\mathbf{e}_n$ and $\mathbf{e}_\tau$ being, respectively, the unit normal and tangential vectors. One assumes that, far away, the perturbation caused by the bodies fades as

$$|\mathbf{u}| \to 1 \text{ at } S_\infty . \qquad (3c)$$

An oscillatory moving with finite amplitude A and constant angular velocity φ is added to bodies motion. This is represented in Fig. 2, for example, by a heaving square cylinder immersed in a uniform incoming flow with velocity U. In this figure the (x,o,y) is the inertial frame of reference and the (X,O,Y) is the coordinate system fixed to the cylinder; this coordinate system oscillates around the x-axis as y_0=Acos(φt).

In the body fixed coordinate system, the surface $S_1 \equiv S_b$ is defined by the function

$$F_b(X,Y) = Y_b - \eta(X) = 0 \qquad (4)$$

Thus, in the inertial frame of reference

$$S_b : F_b(x,y,t) = y_b - [y_0(t) + \eta(x)] = 0 , \qquad (5)$$

and, for a symmetrical body

$$F_b(x,y,t) = y_b - y_0(t) \mp \eta(x) = 0 . \qquad (6)$$

Is considered an small amplitude around the axis x, therefore

$$\frac{A}{b} = O(\varepsilon) \text{ , where } \varepsilon \to 0 \text{ and } \varphi = O(1). \qquad (7)$$

Thus, the boundary conditions on S_1 and S_2 are written directly in the inertial frame of reference as

$$u_n(x,y,t) \equiv \left[v_n(x,y,t) \right] \text{ on } S_1 \text{ and } S_2 \text{, the impenetrability condition} \qquad (8a)$$

$$u_\tau(x,y,t) \equiv \left[v_\tau(x,y,t) \right] \text{ on } S_1 \text{ and } S_2 \text{, the no-slip condition.} \qquad (8b)$$

The transference of the boundary conditions on S_1 and S_2 from actual position to the mean position is defined as

$$y_c = y_0 + \eta(x) \;\to\; \bar{y}_c = \eta(x) + O(y_0) \qquad (9a)$$

$$u_n(x_c, y_c, t) = u_n\left(x_c, y_0 + \eta, t\right) = u_n\left(x_c, \eta(x_c), t\right) + y_0 \frac{\partial u_n\left(x_c, \eta(x_c), t\right)}{\partial y} + \cdots \tag{9b}$$

$$u_\tau(x_c, y_c, t) = u_\tau\left(x_c, y_0 + \eta, t\right) = u_\tau\left(x_c, \eta(x_c), t\right) + y_0 \frac{\partial u_\tau\left(x_c, \eta(x_c), t\right)}{\partial y} + \cdots \tag{9c}$$

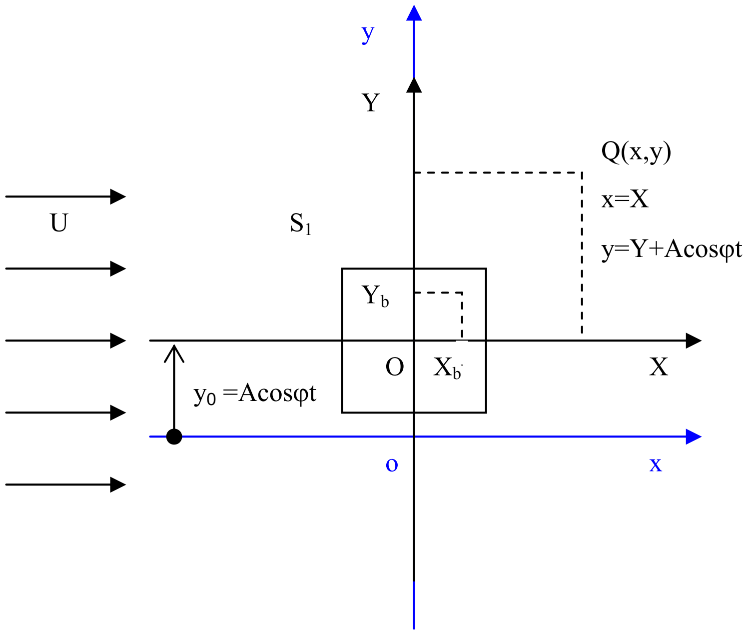

Figure 2. Definitions

3. Vortex method algorithm

The dynamics of the fluid motion, governed by the boundary-value problem (1), (2) and (9), can be studied in a more convenient way when is taked the curl of the Navier-Stokes equations to obtain the vorticity equation. For a 2-D flow this equation is scalar, and it can be written as

$$\frac{\partial \omega}{\partial t} + \mathbf{u} \cdot \nabla \omega = \frac{1}{Re} \nabla^2 \omega \tag{10}$$

in which ω is the only non-zero component of the vorticity vector (in a direction normal to the plane of the flow). One of the advantages of working with the Eq. (10) is the elimination of the pressure term, which always requires special treatment in most numerical experiments.

The left hand side of the above equation carries all the information needed for the convection of vorticity while the right hand side governs the diffusion. Following Chorin (1973) we use the viscous splitting algorithm, which, for the same time step of the numerical simulation, says that

Convection of vorticity is governed by

$$\frac{\partial \omega}{\partial t} + \mathbf{u} \cdot \nabla \omega = 0 \,. \tag{11}$$

Diffusion of vorticity is governed by

$$\frac{\partial \omega}{\partial t} = \frac{1}{Re} \nabla^2 \omega \,. \tag{12}$$

Convection is governed by Eq. (11) and the velocity field is given by

$$u - iv = \mathbf{u}_{\text{main stream}} + \mathbf{u}_{\text{cylinders}} + \mathbf{u}_{\text{vortex cloud}} . \tag{13}$$

Here, u and v are the x and y components of the velocity vector $\mathbf{u}$ and $i = \sqrt{-1}$. The first term in the right hand sides is the contribution of the incident flow; the second term in the right hand sides is the summation of integral terms comes from the source panels distributed on each cylinder surface. The third term is associated to the velocity induced by the cloud of N free vortices; it represents the vortex-vortex interactions.

The incident flow and the vortex-vortex interactions calculations present no problems and they follow the usual Vortex Method procedures; to the first approximation the same happens with the summation of $2\,M$ integral terms when the bodies oscillation amplitude is small; see Moura *et al.* (2006). For large amplitude body oscillations, however, the bodies' boundary conditions can not be transferred from the actual position to the mean position; see Recicar *et al.* (2006).

The fluid velocity on the square cylinder surfaces is written as

$$\mathbf{u}(X, Y; t) = U\mathbf{i} - \dot{y}_0(t)\mathbf{j} \text{ ; with } \dot{y}_0(t) = \frac{d}{dt}\left[A\cos(\varphi t)\right] \tag{14}$$

As a consequence of the j component of the right hand side of the fluid velocity (in the above expression) one gets an additional singularities distribution on the bodies' surface. Of course, the induced velocity due to this additional singularities distribution fades away from the bodies.

The velocity induced by the bodies, according to the Panels Method calculations, is indicated by [uc(X,Y), vc(X,Y)]; this is the velocity induced at the vortex (i), located at the point [x(t), y(t)]; thus

$$uc^{(i)}(x, y; t) = uc(X, Y; t) \tag{15a}$$

$$vc^{(i)}(x, y; t) = vc(X, Y; t) \tag{15b}$$

where the following relations remains

$$x^{(i)}(t) = X \tag{16a}$$

$$y^{(i)}(t) = y_0(t) + Y \tag{16b}$$

The process of vorticity generation is carried out from Eq. (9c), so as to satisfy the no-slip condition. According to the discussion above the Panels Method guaranties that the impermeability condition is satisfied in each straight-line element, or panel, at pivotal point. At each instant of the time news vortices are created a small distance ε of the cylinders surface, whose strengths are determined from Eq. (9c) applied at 2M point's right below the newly created vortices, along the radial direction. This procedure yields an algebraic system of 2M equations and 2M unknowns (the strengths of the vortices).

The vorticity is simulated by cloud of Lamb vortices, whose mathematical expression for the induced velocity of the kth vortex with strength $\Delta\Gamma_k$ in the circumferential direction u_{θ_k} , is (Mustto *et al.*, 1998)

$$u_{\theta_k} = \frac{\Delta\Gamma_k}{2\pi r}\left\{1 - \exp\left[-5.02572\left(-\frac{r}{\sigma_0}\right)^2\right]\right\} \tag{17}$$

where σ_0 is core radius of the Lamb vortex.

In this particular equation r is the radial distance between the vortex center and the point in the flow field where the induced velocity is calculated.

Each vortex particle distributed in the flow field is followed during numerical simulation according to the Adams-Bashforth second-order formula (Ferziger, 1981)

$$z(t + \Delta t) = z(t) + [1.5u(t) - 0.5u(t - \Delta t)]\Delta t + \xi \tag{18}$$

in which z is the position of a particle, Δt is the time increment and ξ is the random walk displacement. According to Lewis (1999), the random walk displacement is given by

$$\xi = \sqrt{4\beta\Delta t \ln\left(\frac{1}{P}\right)}\left[\cos(2\pi Q) + i\sin(2\pi Q)\right] \tag{19}$$

where $\beta = \mathrm{Re}^{-1}$; P and Q are random numbers between 0.0 and 1.0.

Having determined the vorticity field the pressure calculation starts with the Bernoulli function, defined by Uhlman (1992) as

$$Y = p + \frac{u^2}{2}, \; u = |\mathbf{u}| \tag{20}$$

Following Shintani and Akamatsu (1994) this function is then obtained using the following integral formulation

$$H\overline{Y}_i - \int_{S_{1,2}} \overline{Y}\nabla G_i \cdot \mathbf{e}_n dS = \iint_\Omega \nabla G_i \cdot (\mathbf{u} \times \boldsymbol{\omega}) d\Omega - \frac{1}{\mathrm{Re}} \int_{S_{1,2}} (\nabla G_i \times \boldsymbol{\omega}) \cdot \mathbf{e}_n dS \, . \tag{21}$$

Here H is 1.0 inside the flow (at domain Ω) and is 0.5 on the boundaries S_1 and S_2. $G_i = (1/2\pi)\log R^{-1}$ 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. (21) is solved by mean of a set of simultaneous equations for pressure Y_i. The pressure coefficient on a panel control point i is calculated according to $C_{p_i} = 1 + Y_i$.

4. Discussion and results

Each cylinder surface was discretized into M small straight panels of equal lengths. In the calculations, each cylinder surface was represented by eighty (M=80) straight-line source panels with constant density. All runs were performed with 800 time steps of magnitude Δt=0.04. Here, only the case Re=10^5 was chosen. The time increment was evaluated according to $\Delta t = 2\pi k/M$, $0<k\leq 1$ (Mustto *et al.*, 1998).

The process of vorticity generation is carried out so as to satisfy the no-slip condition, Eq. 9c. In each time step the nascent vortices were placed into the cloud through a displacement $\varepsilon = \sigma_0$=0.0009b normal to the panels. The aerodynamics loads starts at t=8. The aerodynamics forces are calculated through the integration of the pressure coefficient distribution on the each cylinders surface.

Table 1 provides an easy way to compare the present results of the flow around a single square cylinder for the drag coefficient and Strouhal number to other experimental (with 10% uncertainty) and numerical results available in the literature.

Table 1. Comparison of the mean drag coefficient and Strouhal number with experimental results.

Results at Re=10^5	Aspect Ratio	C_D	C_L	St
Blevins (1985): Experimental	1.0	2.20	--	0.120
Vickery (1966)	1.0	2.05	--	0.118
Guedes *et al.* (2003)	1.0	1.88	--	0.138
Present simulation	1.0	2.13	0.02	0.153

The present mean drag coefficient is 4% lower than the experimental obtained by Blevins (1985), whereas the Strouhal number is 20% higher. The discrepancy may be attributed to errors in the simulation of the vortex shedding mechanism near the corners of the cylinder that result from the panel distribution on the forward face. Also, 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. In the present method all nascent vortices were placed into the cloud through a same displacement normal to the panels. This kind of flow is relatively insensitive to the Reynolds number (Blevins, 1984) due the fact that separation is fixed at the cylinder corners, for high values of Re. Regardless of the overall good comparison, this present results indicate that the algorithm still needs investigations in order to yield more accurate results.

The flow around a square cylinder presents several interesting characteristics, which can be described starting with the occurrence of the separation phenomenon. Figure 3 shows the vortex positions in the wake, after 800 time steps of the simulation. This figure shows the formation of the Von Karman vortex street, which is comprised of large vortices generated and shed alternately from upper and lower surfaces of the cylinder. The vortices in the wake are connected in pairs by a vortex sheet.

Figure 3. Positions of the wake vortices for $Re=10^5$ at t=32; M=80, Δt=0,04, A=0, φ=0, $\varepsilon=\sigma_0$=0.0009.

Recicar *et al.* (2006) identified three different types of flow regime as the circular cylinder oscillation frequency increases. The first type – Type I – is observed for low frequency range of the cylinder oscillation; in this situation the Strouhal number remains almost constant. Type I is followed by an intermediate range of frequency – Type II, the transition regime – where apparently the shedding frequency does not correlate to the frequency of the cylinder oscillation. Finally in Type III – high frequency of cylinder oscillation – the vortex shedding frequency is locked-in with the cylinder oscillation frequency.

The graph for the variation with time of the lift and drag coefficients with oscillatory motion can be seen in Fig. 4 for single square, with A=0.05 and φ=1.5. As can be observed the lift coefficient oscillates with the same frequency of the body oscillation and its amplitude can be reach values as high as 2.0 to 2.5. This phenomenon is the lock-in regime. This shows that the present vortex code is able to predict a very good estimate of the lock-in regime. The amplitude of oscillatory is considered to be small compared to the body length; therefore, to the first approximation one is allowed to transfer the body boundary condition from the actual position to a mean position of the body surface.

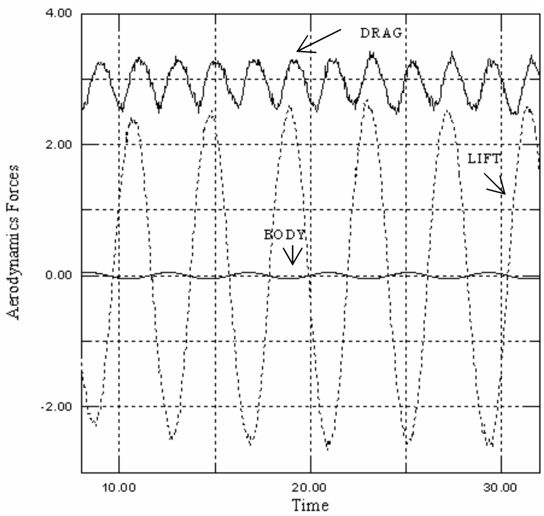

Figure 4. Variation of C_D and C_L with time for single square cylinder, $Re=10^5$, M=80, Δt=0.05, A=0.05, φ=1.5.

As second case is considered two square cylinders in tandem with A=0, φ=0 and d/b=1. The time histories of the lift and drag coefficients are reveled in Fig. 5. Some discrepancies observed in the determination of the aerodynamics forces may be also 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. In the present method all nascent vortices were placed into the cloud through a displacement $\varepsilon=\sigma_0$ =0.0009b normal to the panels.

No attempts to simulate the flow for M greater than 80 were made since the operation count of the algorithm is proportional to the square of N. As M increases N also tends to increase, and the computation becomes expensive.

The use of a fast summation scheme to determine the vortex-induced velocity, such as the Multiple Expansion scheme, allows an increase in the number of vortices and a reduction of the time step, which increases the resolution of the simulation, in addition to a reduction of the CPU time, which allows a longer simulation time to be carried out. The present calculation required 21 h of CPU time in an Intel(R) Pentium(R) 4 CPU 1700 MHz.

Future work will investigate the variation in Strouhal number with increase in spacing d/b between two cylinders in a tandem arrangement. Also, use of the turbulence modeling (Alcântara Pereira *et al.*, 2002) will produce a better numerical result.

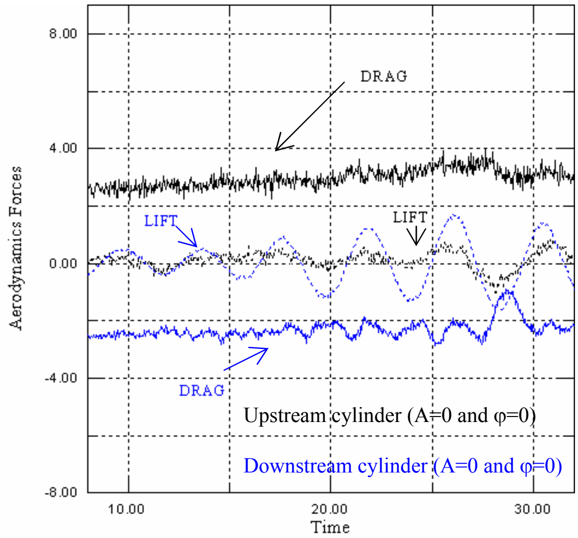

Figure 5. Variation of C_D and C_L with time for two cylinders in tandem (d/b=1.0), Re=10^5, M=80, A=0, φ=0, Δt=0.04.

The influence on the aerodynamics forces of the oscillatory motion, with A=0.05 and φ=1.5, for both square cylinders, is preliminary presented in Fig. 6. News simulations will be carried out to investigate the present phenomena.

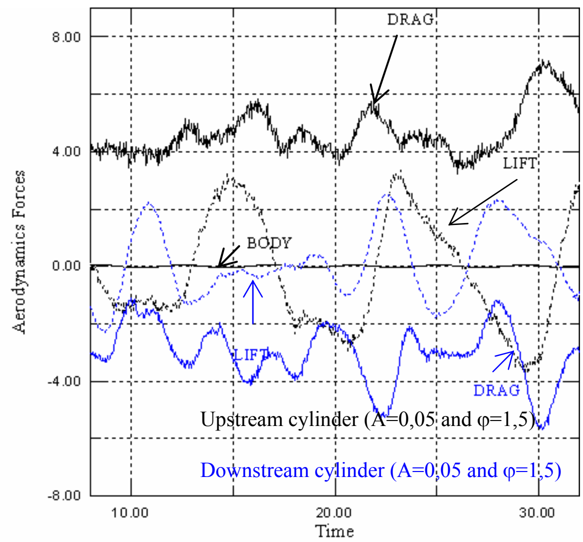

Figure 6. Variation of C_D and C_L with time for two cylinders in tandem (d/b=1.0), Re=10^5, M=80, A=0.05, φ=1.5.

Figure 7 shows the position of the wake vortices for the same case at last step of the computation (t=32), where we can clearly observe the formation and shedding of large eddies in the wakes.

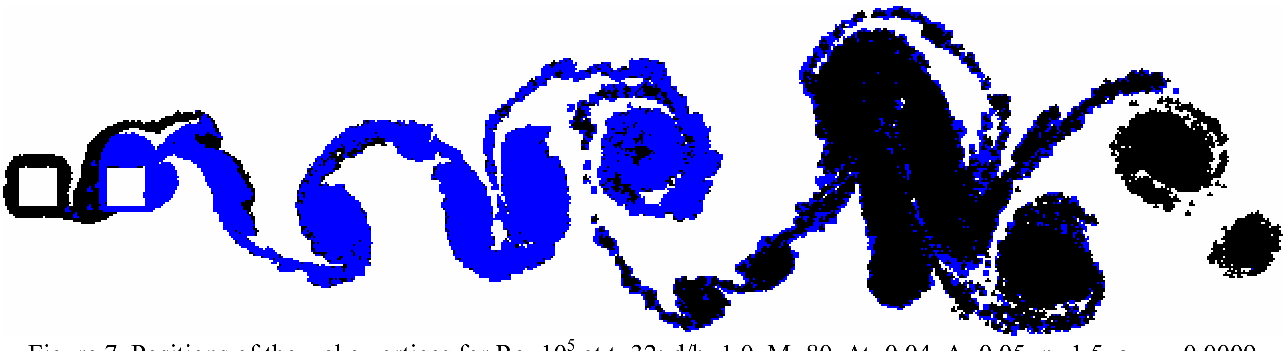

Figure 7. Positions of the wake vortices for Re=10^5 at t=32; d/b=1.0, M=80, Δt=0.04, A=0.05, φ=1.5, $\varepsilon=\sigma_0$=0.0009.

Finally, 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.

5. Conclusions

The main objective of the work was to implement the algorithm and to get some insight into the potentialities of the model developed; this was accomplished since the results show that the behavior of the quantities of interest is the expected one.

The results for drag and lift coefficients predicted by the simulation in Fig. 5 and Fig. 6 need further investigation. This seems to indicate that a higher value of M would improve the resolution and probably produce a better simulation with respect to the aerodynamics forces. More investigations are needed and one can imagine that with the use of more panels (and therefore more free vortices in the cloud) the results tend to be in closer agreement with the experiments.

Some discrepancies observed in the determination of the aerodynamics forces may be also 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. In the present method all nascent vortices were placed into the cloud through a displacement $\varepsilon = \sigma_0 = 0.0009b$ normal to the panels.

All the 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 results of this analysis, taking into account the sub-grid turbulence modeling, are also being generated and will be presented in due time, elsewhere.

6. Acknowledgements

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

7. References

Alam, M. M., Moriya, M., Takai, K. and Sakamoto, 2003, "Fluctuating Fluid Forces Acting on Two Circular Cylinders in a Tandem Arrangement at a Subcritical Reynolds Number", J. Wind Eng. Ind Aerodyn., 91, pp. 139-154.

Alcântara Pereira, L.A. and Hirata, M.H., 2006, "On Interference between Circular Cylinders in Cross Flow at High Reynolds Number Using Surface Vorticity Method", 11th Brazilian Congress of Thermal Sciences and Engineering, Proceedings of ENCIT 2006, December 5-8, Curitiba, 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.

Arie, M., Kiya, M., Moriya, M. and Mori, H., 1983, "Pressure Fluctuations on the Surface of Two Circular Cylinders in Tandem Arrangement", ASME J. Fluids Eng. 105, pp. 161-167.

Biermann, D. and Herrnstein, Jr., 1933, "The Interference between Struts in Various Combinations", National Advisory Committee for Aeronautics, Thecnical Report 468.

Blevis, R.D., 1984, Applied Fluid Dynamics Handbook, Van Nostrand Reinhold Co.

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

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

Fornberg, B., 1985, "Steady Viscous Flow Past a Circular Cylinder up to Reynolds number 600", Journal of Fluid Mechanics, Vol. 61, pp. 297-320.

Guedes, V.G., Bodstein, G.C.R. and Hirata, M.H., 2003, "Numerical Simulation of the Flow Around Rectangular Cylinders Using Vortex Method", 3st International Conference on Computational Heat and Mass Transfer, Proceedings of 3ICCHMT, May 26-30, Banff, Canada.

Hangan, H. and Vickery, B.J., 1999, "Buffeting of a 2D (Sharp-Edge) Bluff Bodies", J. Wind Eng. Eng. Aerodyn., 82, 173-187.

Havel, B., Hangan, H. and Martinuzzi, R., 2001, "Buffeting for 2D and 3D Sharp-Edge Bluff Bodies", J. Wind Eng. Aerodyn., 89, pp. 1369-1381.

Hiwada, M., Mabuchi, I. and Yanagihara, H., 1982, "Flow and Heat Transfer from Two same Size Circular Cylinders in Tandem Arrangement" (In Japanese), Trans. JSME 48, pp. 499-508.

Hussain, M. and Lee, B.E., 1980, "A Wind Tunnel Study of the Mean Pressure Forces Acting on Large Groups of Low-Rise Buildings", J. Wind Eng. Aerodyn., 6, pp. 207-225.

Igarashi, T., 1984, "Characteristics of the Flow Around Two Circular Cylinders Arranged in Tandem", (Second Report), Bull. JSME 27 (233), pp. 2380-2387.

Igarashi, T., 1981, "Characteristics of the Flow Around Two Circular Cylinders Arranged in Tandem", (First Report), Bull. JSME 24 (188), pp. 323-331.

Jackson, C.P., 1987, "A Finite-Element Study of the Onset of Vortex Shedding in Flow Past Variously Shaped Bodies", Journal of Fluid Mechanics, Vol. 182, pp. 23-45.

Jendrzejczk, J.A. and Chen, S.S., 1986, "Fluid Forces on Two Circular Cylinders in Cross Flow", ASME, PVP 14, pp. 141-143.

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.

Kostic, Z.G. and Oka, .S.N, 1972, "Fluid Flow and Heat Transfer with Two Circular Cylinders in Cross Flow", Int. J. Heat Mass Transfer, 15, pp. 279-299.

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.

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.

Novak, J., 1974, "Strouhal Number of a Quadrangular Prism, Angle Iron and Two Circular Cylinders Arranged in Tandem", Acta Tech. CSAV 19 (3), pp. 361-373.

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.

Ohya, Y., Okajima, A. and Hayashi, M., 1989, "Wake Interference and Vortex Shedding", In: Encyclopedia of Fluid Mechanics, Chap. 10 (Gulf Publishing, Houston, 1989).

Okajima, A., 1979, "Flow Around Two Tandem Circular Cylinders at very High Reynolds Numbers", Bull. JSME 22 (166), pp. 504-511.

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.

Sarpkaya, T., 1989, "Computational Methods with Vortices - The 1988 Freeman Scholar Lecture", Journal of Fluids Engineering, Vol. 111, pp. 5-52.

Sethian, J.I., 1991, "A Brief Overview of Vortex Method, Vortex Methods and Vortex Motion", SIAM. Philadelphia, pp. 1-32.

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.

Uhlman, J.S., 1992, "An Integral Equation Formulation of the Equation of an Incompressible Fluid", Naval Undersea Warfare Center, T.R. 10-086.

Vickery, B.J., 1966, "Fluctuating Lift and Drag on a Long Cylinder of Square Cross-Section in a Smooth and a Turbulent Stream", Journal of Fluid Mechanics, Vol. 25, pp. 481-494.

Williamson, C.H.K. and Roshko, A., 1988, "Vortex Formation in the Wake of an Oscillating Cylinder", J. Fluids and Structures, Vol. 2, pp. 239-381.

Zhang, H and Melbourne, 1992, "Interference Between Two Circular Cylinders in Tandem in Turbulent Flow", J. Wind Eng. Aerodyn., 41-44, pp. 582-600.

Zdravkovich, M.M., 1977, "Review of Flow Interference between Two Circular Cylinders in Various Arrangements", Trans. SAME, J. Fluid. Eng., 99, pp. 618-633.

Zdravkovich, M.M. and Pridden, D.L., 1977, "Interference between Two Circular Cylinders; Series of Unexpected Discontinuites", J. Ind. Aerodyn. 2, pp. 255-270.

Zdravkovich, M.M. and Pridden, D.L., 1975, "Flow Around Two Circular Cylinders", Research Report, Proceedings of the Second US National Conference on Wind Engineering Research, Fort Collins, IV (18).

8. Responsibility notice

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