Proceedings of the 11th Brazilian Congress of Thermal Sciences and Engineering -- ENCIT 2006
Braz. Soc. of Mechanical Sciences and Engineering -- ABCM, Curitiba, Brazil, Dec.5-8, 2006

Paper CIT06-0344

ON INTERFRENCE BETWEEN CIRCULAR CYLINDERS IN CROSS FLOW AT HIGH REYNOLDS NUMBER USING SURFACE VORTICITY METHOD

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

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

Abstract. *A two-dimensional discrete-vortex model is used to investigate vortex interaction inside the near wakes of circular cylinders in cross flow. The numerical simulation is accomplished by using the vortex method, which takes into account the viscous effect in the flow field. The dynamics of the wakes is computed using the convection-diffusion splitting algorithm, where the convection process is carried out with a Lagrangian second-order Adams-Bashforth time-marching scheme, and the diffusion process is simulated using the random walk method. Aerodynamics loads are calculated using an integral equation derived from the pressure Poisson equation. Comparisons of the computed results with the experimental measurements showed that the present discrete vortex method is able to replicate most of the salient features observed in cylinder array experiments.*

Keywords: *vortex method, panels method, interference, flow patterns, aerodynamics loads.*

1. Introduction

In engineering practice, most structures on land and in the ocean often appear in groups and are confronted by a fluid flow. The flow field around pipes in a cluster is very complex and has been studied extensively. The main goals of that research line have been among others, the measure the fluid force and pressure distribution acting on each cylinder, flow velocity profile, vortex shedding, and to understand the resultant flow patterns.

Due the mutual interference between cylinders at close proximity, the aerodynamics characteristics, such as fluctuating lift and drag forces, vortex-shedding patterns and fluctuating pressure distributions, for each member of a group are completely different from isolated ones. When a cylinder is placed in the wake of another in cross-flow its unsteady loading becomes dependent not only on the flow activities in its wake, but also on those in the wake of the upstream cylinder.

A host of studies have addressed the interference effects between two, three, and even four cylinders in a uniform and/or turbulent flow.

Numerous investigations have been made of the flow past two circular cylinders, which is the simplest case of a group, in the last three decades. 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.

Recently, the Vortex Method was employed by Teixeira da Silveira *et al. (2005)* to simulate the vortex-shedding flow from two tandem cylinders in cross-flow; the aerodynamic characteristics are investigated at a Reynolds number of 6.5×10^4 and comparisons are made with experimental results presented by Alam *et al.* (2003). As the simulations showed, the numerical results obtained are in overall good agreement with the experimental results used for comparison, especially in the simulations for the upstream cylinder. Some discrepancies observed in the determination of the aerodynamics loads for the downstream cylinder 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. In the present method all nascent vortices were placed into the cloud through a same displacement normal to the panels.

When either one or both cylinders are elastic and vibrate, the flow field becomes significantly more complicated because of the interaction of the fluid flow and the cylinders motion. Efforts have been made to understand the phenomena involved. Motivated by concern over the large oscillations frequently occurring in transmission lines exposed to the high wind, most of the studies of elastic cylinders have focused on characterizing the motion of two cylinders in tandem. The disturbed flow caused by the windward cylinder striking the leeward cylinder can induce dynamic instability, called wake induced flutter. In addition, cylinders are also subjected to other fluctuating forces associated with vortex shedding and turbulence in the incoming flow or generated by the cylinder motion. The phenomena have been studied, experimentally and analytically, by Zdravkovich (1977), Simpson and Flower (1977) and Williamson and Roshko (1988).

Kareem *et al* .(1998) presented an experimental study of the interference effects between two and three cylinders of finite height immersed in a turbulent boundary layer at subcritical Reynolds number utilizing a pneumatic averaging manifold system to measure the fluctuating force at various levels.

Lam *et al.* (2001a) employed the Vortex Method to simulate flows around four equi-spaced cylinders. The surface of the each cylinder was represented by straight-line panels with a point vortex located at the control point; the coalescence of vortices when they are too close to each other was introduced, and the pressure on the airfoil surface was calculated according to the inviscid flow analysis. A comparison of the aerodynamics loads obtained with the experimental results by Lam and Fang (1995) showed a better agreement in the drag direction. A more sophisticated scheme is required to obtain better prediction on pressure and force characteristics. He and Su (1994) showed that the results for the pressure calculation could be improved by considering the nonlinear acceleration terms.

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). In this method, the vorticity in the fluid region is numerically simulated using a cloud of discrete vortices with a viscous core (Lamb vortex). To simulate the vorticity at the solid surfaces, nascent vortices are generated there at each time step of the simulation. In order to take care of the convection and the diffusion of the vorticity one makes use of the convection-diffusion splitting algorithm; accordingly the convection of the vortices in the cloud is carried out independently of the diffusion for each time step of the simulation. This is in essence the foundation of the Vortex Method (e.g. references Chorin, 1973; Sarpkaya, 1989; Sethian, 1991; Kamemoto, 1994; Lewis, 1999; Ogami, 2001 and Alcântara Pereira *et al.*, 2002). Please note that with the Lagrangian formulation a grid for the spatial discretization of the fluid region is not necessary. Thus, special care to handle numerical instabilities associated to high Reynolds numbers is not needed. Also, the attention is only focused on the regions of high activities, which are the regions containing vorticity; on the contrary, Eulerian schemes consider the entire domain independent of the fact that there are sub-regions where less important, if any, flow activity can be found. With the Lagrangian tracking of the vortices, one need not take into account the far away boundary conditions. This is of important in the wake regions (which is not negligible in the flows of present interest) where turbulence activities are intense and unknown, a priori.

In the present paper, the Vortex Method is employed to simulate the interference effects for a group of finite cylinders. The interference phenomena are highly non-linear and at present beyond a reliable theoretical or numerical analysis. The main feature of the present vortex code is to simulate numerically the two-dimensional, incompressible, unsteady flow around of pipe clusters: (a) for two pipes, for three-pipes clusters, (c) for regular square multiple clusters, (d) and for irregular multi-pipe clusters.

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 approach, the effect of small scale is not considered.

2. Formulation of the Physical Problem

Consider, e.g., the incompressible fluid flow of a Newtonian fluid around three-pipe clusters an unbounded two-dimensional region. Figure 1 shows the incident flow, defined by free stream speed U and the domain Ω with boundary $S = S_1 \cup S_2 \cup S_3 \cup S_4$, S_1 being the upstream cylinder surface, S_2 being the intermediate cylinder surface, S_3 being the downstream cylinder surface and S_4 the far away boundary.

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

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

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

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 (a reference length). The Reynolds number is defined by

$$Re = \frac{bU}{\upsilon} \tag{2a}$$

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

Figure 1. Flow around three circular cylinders.

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

$$u_n = \mathbf{u} \cdot \mathbf{e}_n = 0 \tag{3a}$$

$$u_\tau = \mathbf{u} \cdot \mathbf{e}_\tau = 0 \tag{3b}$$

$\mathbf{e}_n$ 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

$$|\mathbf{u}| \to 1 \text{ at } S_4 . \tag{3c}$$

The dynamics of the fluid motion, governed by the boundary-value problem (1), (2) and (3), 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

$$\frac{\partial \omega}{\partial t} + \mathbf{u} \cdot \nabla \omega = \frac{1}{Re} \nabla^2 \omega \tag{4}$$

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. (4) is the elimination of the pressure term, which always requires special treatment in most numerical experiments.

3. The discrete vortex method

According to the convection-diffusion splitting algorithm (Chorin, 1973) it is assumed that in the same time increment the convection and the diffusion of the vorticity can be independently handled and are governed by

$$\frac{\partial \omega}{\partial t} + \mathbf{u} \cdot \nabla \omega = 0 \tag{5}$$

$$\frac{\partial \omega}{\partial t} = \frac{1}{Re} \nabla^2 \omega . \tag{6}$$

In a physical sense vorticity is generated on the circular cylinders surface so as to satisfy the no-slip condition, Eq. (3b). The discrete vortex method represents the vorticity by discrete vortices, whose transport at each time increment is carried out in sequence. Convection is governed by Eq. (5) and the velocity field is given by

$$u - iv = 1 + \frac{i}{2\pi} \sum_{n=1}^{M} \gamma(S_n) \int_{\Delta S_n} \frac{d}{dz} \ln(z - \zeta) d\zeta + \frac{i}{2\pi} \sum_{k=1}^{N} \frac{\Delta\Gamma_k}{z - z_k} . \tag{7}$$

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 summation of α M integral terms comes from the panels distributed on the circular cylinders surfaces. The second summation is associated to the velocity induced by the cloud of N free vortices; it represents the vortex-vortex interaction.

In this paper, an improvement was also introduced in the convective step of the simulation; by using the anti symmetry property of the vortex-vortex velocity induction, the computational effort was reduced; this is an important feature, since the vortex-vortex velocity induction calculation is the most time consuming part of the simulation.

In order to remove the singularity in the second summation of Eq. (7) Lamb vortices are used, 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)

$$u_{\theta_k} = \frac{\Delta\Gamma_k}{2\pi r} \left\{ 1 - \exp\left[-5.02572 \left(-\frac{r}{\sigma_0} \right)^2 \right] \right\} \tag{8}$$

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

$$z(t + \Delta t) = z(t) + [1.5u(t) - 0.5u(t - \Delta t)]\Delta t + \xi \tag{9}$$

in which z is the particle position of a vortex particle, Δt is the time increment and ξ is the random walk displacement. According to Lewis (1991), the random walk displacement is given by

$$\xi = \sqrt{4\beta\Delta t \ln\left(\frac{1}{P}\right)} [\cos(2\pi Q) + i \sin(2\pi Q)] \tag{10}$$

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

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

$$Y = p + \frac{u^2}{2}, \; u = |\mathbf{u}| \tag{11}$$

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 & Akamatsu, 1994)

$$H\overline{Y_i} - \int_{S_1} \overline{Y} \nabla G_i \cdot \mathbf{e}_n dS = \iint_{\Omega} \nabla G_i \cdot (\mathbf{u} \times \omega) d\Omega - \frac{1}{Re} \int_{S_1} (\nabla G_i \times \omega) \cdot \mathbf{e}_n dS \tag{12}$$

where 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. (12) 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. Results and discussion

The numerical simulations were restricted to the interference effects between two, three, and four cylinders in a uniform flow. Here, L is the spacing between cylinders, and D is the cylinders diameter. In the calculations, each boundary S_i, i = 1, 2, 3 or 4, of Fig. 1 was by fifty (M=50) straight-line vortex panels with constant density. All runs were performed with 600 time steps of magnitude Δt=0.05. The time increment was evaluated according to $\Delta t=2\pi k/M$, $0<k\leq 1$ (Mustto *et al.*, 1998). In each time step the nascent vortices were placed into the cloud through a displacement $\varepsilon = \sigma_0$ =0.03b normal to the panels.

All the aerodynamics forces and pressure distributions computations starts at t=15. The aerodynamics force coefficients are calculated through the integration of the pressure coefficient distribution on the each cylinders surface.

Computed values for the distribution of the mean pressure coefficient along the single cylinder surface is shown in Fig. 2. As expected the simulation predicts the occurrence of a (mean) stagnation point at the front of the cylinder. The value of C_p then decreases and reaches a plateau, in the separated flow region. The magnitude of pressure becomes zero at $\theta=32^\circ$ and becomes maximum negative at $\theta=72^\circ$, whereas the experimental values is $\theta=34^\circ$ and $\theta=69^\circ$, respectively.

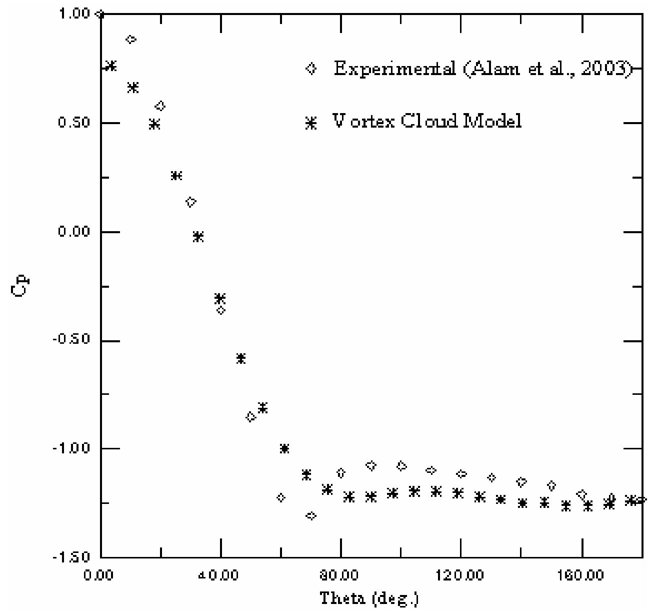


Figure 2. Comparison of the single cylinder case, experimental and numerical results of C_p, for Re=6.5x10^4.

Table 1 presents all cases studied for two circular cylinders in a tandem arrangement at a subcritical Reynolds number of 6.5x10^4 (Teixeira da Silveira *et al.,* 2005).

Within the results presented in Table 1, we observe a disagreement of the numerical results to the experimental results (Alam *et al.*, 2003) of cases III, IV and VII on the time-averaged drag coefficient, C_D, of the downstream cylinder. The mean drag coefficients of the downstream cylinder are much higher than the experimental values and, therefore, do not reflect a good simulation of the flow. The differences encountered in the comparison of the numerical results with the experimental results 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. A purely two-dimensional computation of such flow must produce higher values for the drag coefficient, as obtained for our simulation.

No attempts to simulate the flow for M greater than 50 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.

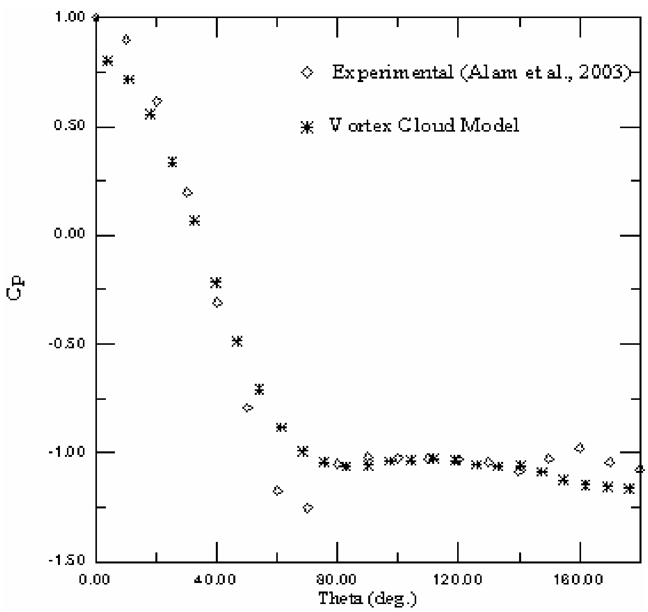
Table 1. Comparison of the mean drag coefficient with experimental results, for Re=6.5x10^4.

Case	L/D	Upstream cylinder		Downstream cylinder	
		$\overline{C_D^+}$	$\overline{C_D^*}$	$\overline{C_D^+}$	$\overline{C_D^*}$
I	0.1	1.0953	1.1500	-0.5697	-0.5447
II	0.5	---	0.9866	-0.3884	-0.2997
III	1.0	1.0531	1.3664	-0.2366	0.1130
IV	2.0	0.9866	1.3434	-0.1345	0.3652
V	3.5	1.2612	1.3677	0.2766	0.4613
VI	4.0	1.2319	1.4174	0.2661	0.3015
VII	8.0	1.2040	1.4324	0.3604	0.8693

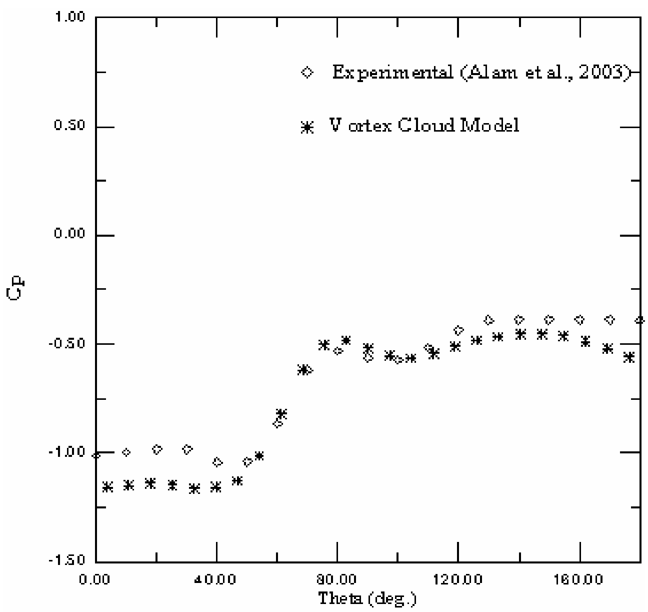
$^+$Experimental results (Alam *et al.*, 2003) *Present results (Teixeira da Silveira *et al.,* 2005)

Experiments (Alam *et al.*, 2003) were conducted in a low-speed, closed-circuit wind tunnel with a test section of 0.6 m height, 0.4 m width, and 5.4 m length. The level of turbulence in the working section was 0.19%. The cylinders used as test models were made of brass and were each 49 mm in diameter. The geometric blockage ratio and aspect ratio at the test section were 8.1% and 8.2, respectively. None of the results presented were corrected for the effects of wind-tunnel blockage.

Figure 3 presents the distribution of pressure coefficient, C_p along the surface of the upstream and downstream cylinders for spacing L/D=0.1 (case I).



Upstream cylinder (L/D=0.1)



Downstream cylinder (L/D=0.1)

Figure 3. Pressure distribution along the surface of the upstream and downstream cylinders, for Re=6.5x10^4.

Figure 4 shows the position of the wake vortices for three cylinders last step of the computation (t=60), where we can clearly observe the formation and shedding of large eddies in the wakes. This process occurs alternately on the upper and lower surfaces of each cylinder arranged in tandem. We can also visualize the vortex pairing process, where the vortices rotate in opposite directions and are connected to each other by a vortex sheet.

Numerical simulations of the flow around four cylinders in a square configurations have been carried out at sub-critical Reynolds number (Re=1.3 x10^4) and several spacing ratios. However, typical results for four equal size cylinders at spacing ratios L/D=1.5 and L/D=4.0, and with flow incident angles that vary from 0° (normal configuration) to 45° (rotate square configuration) have been investigated. The reason for this choice of parameters is due the experimental data obtained at subcritical Reynolds numbers (2.0 x10^2 - 1.3 x10^4) can be found in Lam and Lo (1992) Lam and Fang (1995) and Lam *et al.* (2001b).

The results of the mean drag coefficient and mean lift coefficient for L/D=4 at α=45° obtained in the present simulation are presented in Table 2. In this table one can find also experimental (Lam and Fang, 1995) and numerical results Lam *et al.* (2001a). The numerical results of Lam *et al.* (2001a) were also obtained using the Vortex Method where surface of the each cylinder was represented by straight-line panels with a point vortex located at the control point and the pressure on the airfoil surface was calculated according to the inviscid flow analysis.

As one can see, the agreement between the two numerical methods is very good for the mean drag coefficient and both results are close to the experimental values. However, as mentioned earlier, the three-dimensional effects present in the experiments are very important for the Reynolds number used in the simulations. Therefore a purely two-dimensional computation of such flow must produce higher values for the drag coefficient, as obtained for our simulation. Lam and Fang (1995) do not furnish results for the Strouhal numbers.

Table 2. Comparison of the mean drag coefficient and lift coefficient for L/D=4 and α=45°, for Re=1.3x10^4.

Results	L/D	Cylinder 1	Cylinder 2	Cylinder 3	Cylinder 4
C_D^+	4.0	1.02	1.3	0.62	1.3
$\overline{C_D^*}$	4.0	1.06	1.1	0.5	1.29
$\overline{C_D^\circ}$	4.0	1.22	1.27	0.83	1.26
C_L^+	4.0	-0.01	-0.02	-0.02	-0.03
$\overline{C_L^*}$	4.0	-0.14	-0.15	-0.13	-0.16
$\overline{C_L^\circ}$	4.0	-0.07	-0.09	-0.12	-0.14

$^+$ Experimental results (Lam and Fang, 1995)

* Numerical results (Lam *et al.,* 2001a)

$^\circ$ Present simulation

No attempts to simulate the flow for M greater than 50 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 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).

Figure 4. Position of the wakes vortices at t=60; Re=6.5x10^4, $\varepsilon = \sigma_0$ =0.03b, Δt=0.05, M=50, L/D=2.0.

The sub-grid turbulence modeling is of significant importance for the present numerical simulation. The development of Lagrangian LES models for Vortex Method has been discussed in the literature. In the present study, a methodology for the numerical simulation of a viscous turbulent flow will be carried out (Alcântara Pereira *et al.,* 2002), where the large structures of the flow are described using a cloud of discrete Lamb vortices, which is used to simulate the vorticity present in the fluid region. In order to take into account the smaller scale manifestations of the flow and still keeping the computational effort within a manageable range, a second-order velocity structure function model was adapted to the Lagrangian scheme. This methodology has been used to simulate the flow around a cascade of NACA 65-410 airfoils (Alcântara Pereira *et al.*, 2004, 2005).

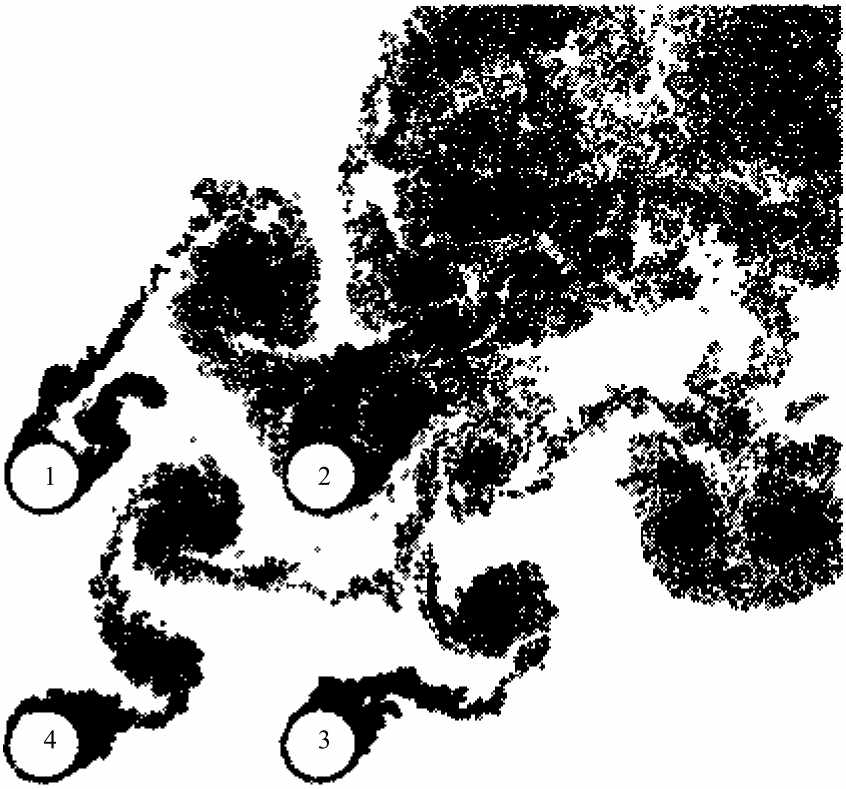


Figure 5. Position of the wakes vortices at t=60 for L/D=4, α=45°, Re=1.3x10^4, $\varepsilon=\sigma_0$=0.03b, Δt=0.05, M=50.

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 three-dimensional effects present in the experiments are very important for the Reynolds number used in the simulations. Therefore a purely two-dimensional computation of the flow must produce differences in the comparison of the numerical results with the experimental results. The differences encountered in the comparison of the computed values with the experimental results for the distribution of the mean pressure coefficient along the cylinder surface as shown in Figure 2 are attributed mainly to the inherent three-dimensionality of the real flow for such a value of the Reynolds number, which is not modeled in the simulation. The results for the pressure distribution indicated that there was a lack of resolution near the stagnation point and the position of the separation point. The position of the separation point is predicted to occur at about 72°, whereas the experimental value (Alam *et al.*, 2003) is about 69°. This seems to indicate that a higher value of M would improve the resolution and probably produce a better simulation with respect to the pressure distribution. 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 loads 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 $\varepsilon=\sigma_0$=0.03b normal to the panels.

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 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.

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.

6. Acknowledgement

The authors would like to acknowledge FAPEMIG (Process 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.

Alcantara Pereira, L.A., Ricci, J.E.R. and Hirata, M.H., 2005, "Simulation of Linear Cascade Vortex Cloud Interaction with Turbulence Modeling", Intern'l Society of CFD, Vol. 14, No. 2, July, pp. 1-10.

Alcântara Pereira, L.A., Hirata, M.H. and Manzanares Filho, N. 2004, "Wake and Aerodynamics Loads in Multiple Bodies - Application to Turbomachinery Blade Rows", J. Wind Eng. Ind Aerodyn., 92, pp. 477-491.

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.

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.

Greengard, L. and Rokhlin, V., 1987, "A Fast Algorithm for Particle Simulations", Journal of Computational Physics, Vol. 73, pp. 325-348.

He, F. and Su, T.C., 1998, "A Numerical Study of Bluff Body Aerodynamics in High Reynolds Number Flows by Viscous Vortex Element Method", J. Wind Eng. Ind. Aerodyn., 77/88, pp. 393-407.

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.

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.

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.

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.

Kareem, A., Kijewski, Po-Chien, L., "Investigation of Interference Effects for a Group of Finite Cylinders", 1998, J. Wind Eng. Ind. Aerodyn. 77&78, pp. 503-520.

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.

Lam, K., So, R.M.C. and Li, J.Y., 2001a, "Flow Around Four Cylinders in a Square Configurations Using Surface Vorticity Method", Proceedings of The Second International Conference on Vortex Methods, September 26-28, Istanbul, Turkey.

Lam, K., Chan, K.T., So, R.M.C. and Li, J.Y., 2001b, "Velocity Map and Flow Pattern of Flow around Four Cylinders in a Square Configuration at Low Reynolds Number and Large Spacing Ratio Using Particle Image Velocimetry", Proceedings of The Second International Conference on Vortex Methods, September 26-28, Istanbul, Turkey.

Lam, K. and Fang, X., 1995, "The Effect of Interference of Four Equal-Distance Cylinders in Cross Flow on Pressure and Forces Coefficientes", Journal of Fluid Structures, Vol. 9, pp. 195-214.

Lam, K. and Lo, S.C., 1992, "A Visualization Study of Cross-Flow around Four Cylinders in a Square Configuration", Journal of Fluids and Structure, Vol. 6, pp. 109-131.

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.

Lewis, R. I., 1991, "Vortex Element Method for Fluid Dynamic Analysis of Engineering Systems", Cambridge Univ. Press, Cambridge, England, U.K..

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.

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.

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.

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.

Simpson, A. and Flower, J.W., 1977, "An Improved Mathematical Model for the Aerodynamic Forces on Tandem Cylinders in Motion with Aerolastic Applications", J. Sound Vib., Vol. 51, pp. 183-217.

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.

Teixeira da Silveira, L., Alcântara Pereira, L.A. and Hirata, M.H., 2005, "The Effects of Interference between Two Circular Cylinders Arranged in Tandem by Vortex Method", 18th International Congress of Mechanical Engineering, Proceedings of COBEM 2005, November 6-11, Ouro Preto, MG.

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", J. Fluids and Structures, Vol. 2, pp. 239-381.

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).