THE EFFECTS OF INTERFERENCE BETWEEN TWO CIRCULAR CYLINDERS ARRANGED IN TANDEM BY VORTEX METHOD

Lucas Teixeira da Silveira
Instituto de Engenharia Mecânica, UNIFEI, CP. 50, Itajubá, Minas Gerais, 37500-903, Brasil
lucastsilveira@hotmail.com

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. *This work presents an investigation of the aerodynamics characteristics of two circular cylinders in a tandem arrangement for various values for the gap between the cylinders at high Reynolds number using the viscous vortex element method. 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 scheme. The aerodynamics forces and pressure distribution acting on two circular cylinders are computed using the integral equation derived from the pressure Poisson equation; comparisons are made with experimental results available in the literature.*

***Keywords**: vortex method, panels method, aerodynamics loads, tandem arrangement, lagrangian description*

1. Introduction

Flow behavior around circular cylinders is a classical problem in fluid mechanics with a variety of practical applications, e.g. groups of buildings, chimneys, stacks, chemical reaction-towers, supports of off-shore platform, etc. 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, the so-called tandem arrangement, 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.

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 a 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), Jendrzejczk 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. Recently, 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.

In the present paper, the Vortex Methods is employed to simulate the vortex-shedding flow from two tandem cylinders in cross-flow; the aerodynamic characteristics are investigated at a Reynolds number of $6.5x10^4$ and comparisons are made with experimental results presented by Alam *et al.* (2003).

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).

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 Methods (Chorin, 1973; Sarpakaya, 1989; Sethian, 1991; Lewis, 1999, Alcântara Pereira *et al.*, 2002 and Kamemoto, 2004). Vortex Methods 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 are 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 are 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.

The present algorithm used to investigate numerically the flow around two cylinders in a tandem arrangement by Vortex Methods can be easily adapted for the analysis of the flow around cylinder arrays.

2. Formulation of the physical problem

Consider the incompressible fluid flow of a newtonian fluid around two circular cylinders in a tandem arrangement an unbounded two-dimensional region. Figure 1 shows the incident flow, defined by freestream speed U and the domain Ω with boundary $S = S_1 \cup S_2 \cup S_3$, S_1 being the upstream cylinder surface, S_2 being the downstream cylinder surface and S_3 the far away boundary.

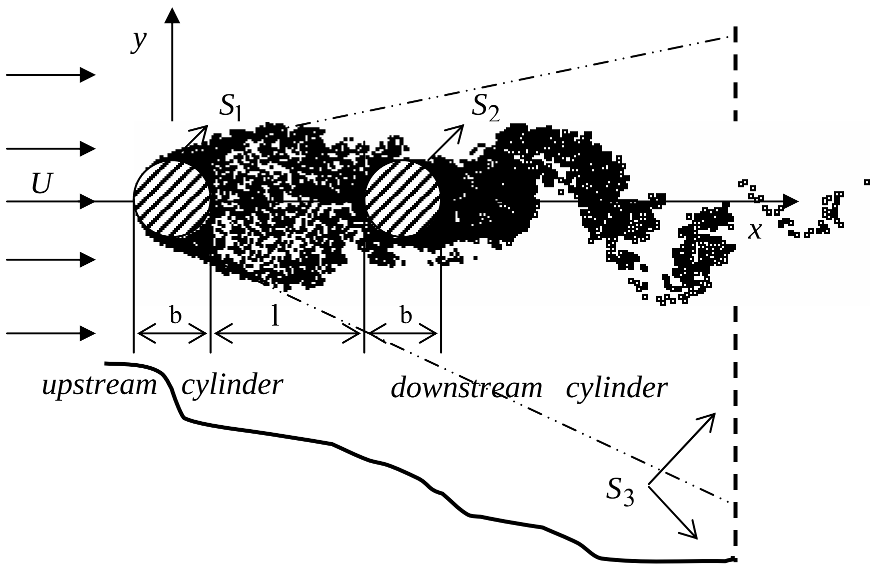


Figure 1. Flow around two circular cylinders in a tandem arrangement.

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

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

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

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 .

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

$$|\mathbf{u}| \rightarrow 1 \text{ at } S_3 . \tag{3c}$$

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

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

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 $2M$ integral terms comes from the panels distributed on the two circular cylinders surface. 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

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

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

Table 1 presents all cases studied for two circular cylinders in a tandem arrangement at a subcritical Reynolds number of $6.5x10^4$. In the calculations, each cylinders surface was represented 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. 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.

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

Case	l/b	Upstream cylinder		Downstream cylinder	
		C_D^+	$\overline{C_D^*}$	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 calculation

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 the 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.

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 2 shows the position of the wake vortices for case VI at 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.

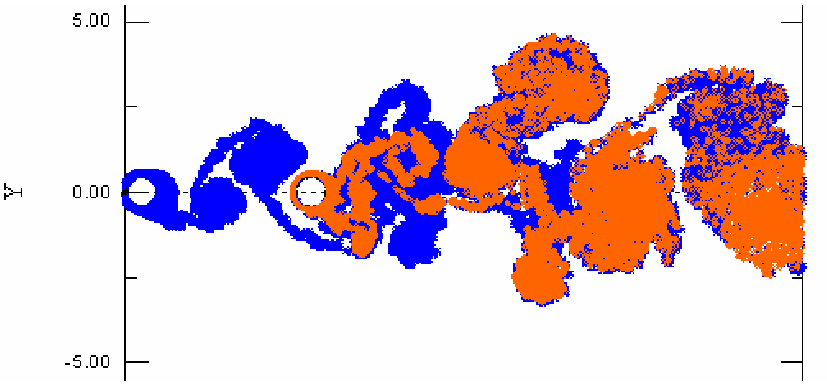


Figure 2. Position of the wakes vortices at t=60 for case VI; $Re=6.5x10^4$, $\varepsilon=\sigma_0=0.03b$, $\Delta t=0.05$, M=50, l/b=4.0.

Computed values for the distribution of the mean pressure coefficient along the single cylinder surface is shown in Fig. 3. 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.

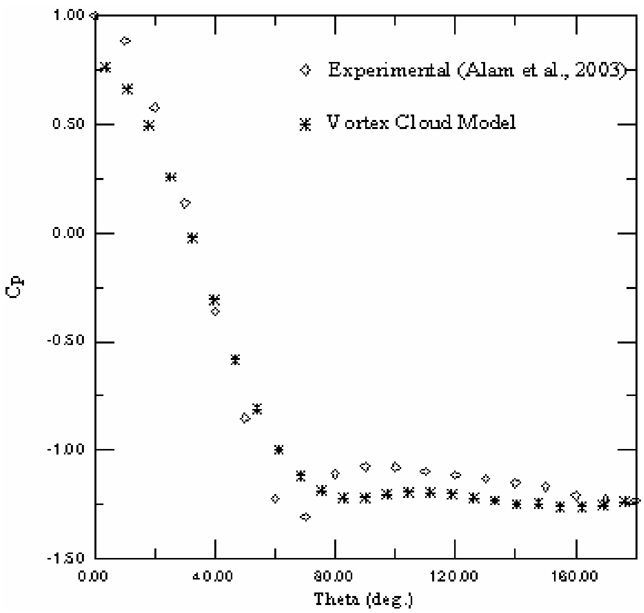
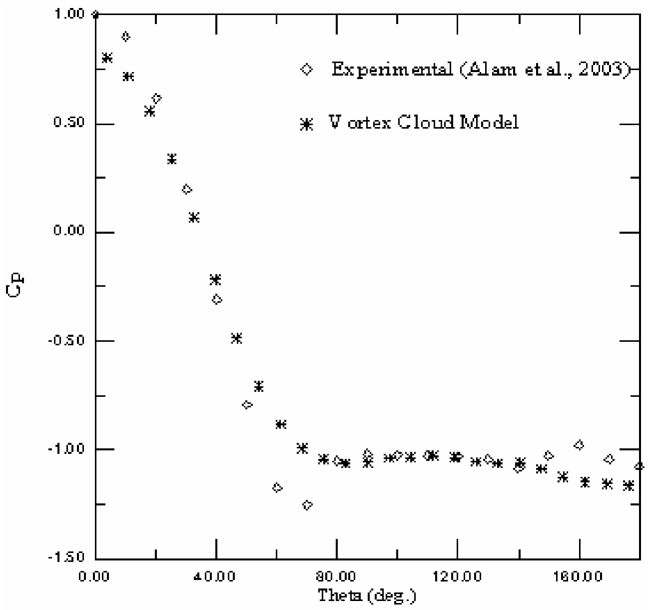
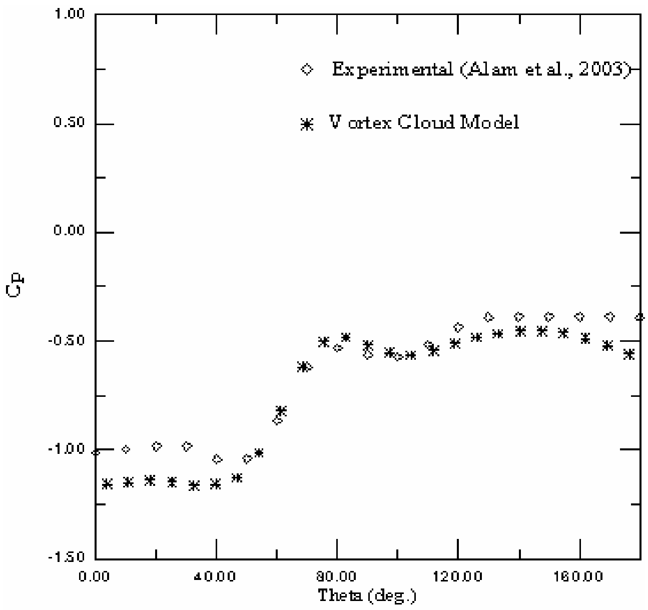


Figure 3. Comparison of the single cylinder case, experimental and numerical results of C_p, for $Re=6.5x10^4$.

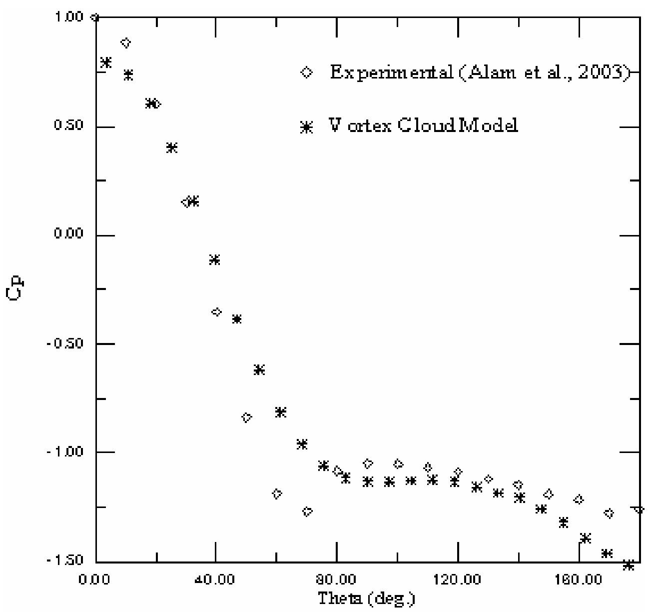
Figure 4 presents the distribution of pressure coefficient, C_p, along the surface of the upstream and downstream cylinders for spacing l/b=0.1 (case I) and l/b=3.5 (case V).



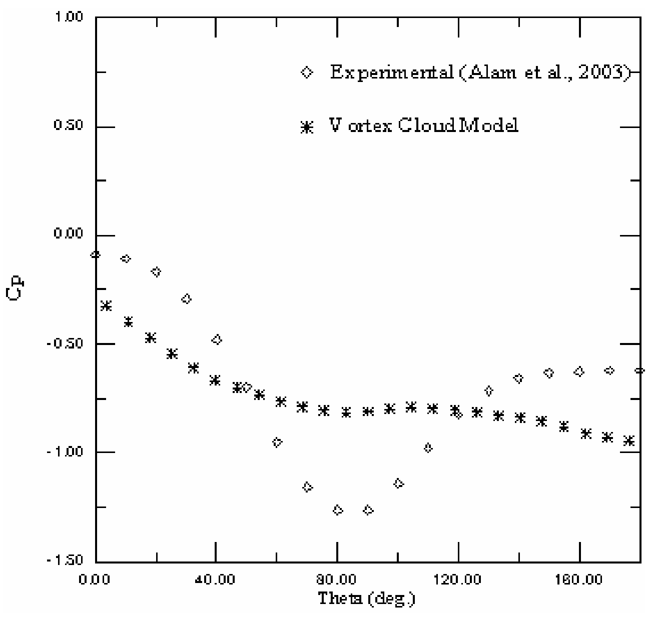
Upstream cylinder (l/b=0.1)



Downstream cylinder (l/b=0.1)



Upstream cylinder (l/b=3.5)



Downstream cylinder (l/b=3.5)

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

5. Conclusions

The new discrete vortex method presenting the analysis of flow around two circular cylinders in a tandem arrangement at a subcritical Reynolds number has been discussed.

As the simulations show, 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 for spacing l/b=1.0, l/b=3.5 and l/b=8.0 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 flowfield. 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 numerical parameters used also require further tests to find more adequate values. 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 18 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 l/b between two cylinders in a tandem arrangement. The sub-grid turbulence modeling 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. Acknowledgements

The authors would like to acknowledge FAPEMIG (Proc. TEC-748/04) 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., 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..

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

Hiwada, M., Mabuchi, I. and Yanagihara, H., 1982, "Flow and Heat Tranfer 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.

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.

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.

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.

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.
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.
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.