COBEM 99
XV Congresso Brasileiro de Engenharia Mecânica
15th Brazilian Congress of Mechanical Engineering
22 - 26 de Novembro de 1999 / November 22 - 26, 1999 Águas de Lindóia, São Paulo,

AN INVESTIGATION OF THE FLOW AROUND TWO CIRCULAR CYLINDERS IN TANDEM ARRANGEMENTS

Julio R. Meneghini
University of São Paulo, EPUSP,
Department of Mechanical Engineering, São Paulo, SP, 05508-900, Brazil
Cesareo de La Rosa Siqueira
University of São Paulo, EPUSP,
Department of Naval & Ocean Engineering, São Paulo, SP, 05508-900, Brazil
Fábio Saltara
University of São Paulo, EPUSP,
Department of Mechanical Engineering, São Paulo, SP, 05508-900, Brazil
José Alfredo Ferrari Jr.
Petrobrás, Rio de Janeiro, Brazil

Abstract: *The shedding of vortices and flow interference between two circular cylinders in tandem arrangements are investigated numerically in this paper for various values for the gap between the cylinders. The calculations are carried out on a three-noded unstructured finite element mesh. All the simulations are performed through the finite-element method. The mesh is finer close to the cylinder wall in order to have a better description of the boundary-layer. The main application of the present investigation is to determine the effective drag coefficient of a bundle of risers existing in offshore platforms used for oil exploration. These risers are subject to currents and oscillatory flow due to waves, flows with a very high degree of complexity, with changes of intensity and direction as the depth of water increases. The Brazilian offshore platforms are located in the Atlantic Ocean where depths of water of more than 1,000 m are very common. Production of more than 100,000 barrels of oil per day in one single platform has already been established. In such conditions a better understanding of the vortex dynamics causing vibration of risers is essential.*

Key-words: *Vortex Shedding, Bluff Body Flow, Computational Fluid Dynamics, Marine Technology*

1. INTRODUCTION

The main application of the present investigation is to determine the effective drag coefficient of a bundle of risers which links the seabed to the offshore platforms used for oil exploration. These risers are subject to shear and oscillatory flows due to currents and waves respectively, flows with a very high degree of complexity, with changes of intensity and direction the deeper the water depth. Most of the Brazilian floating platforms are installed along the continental shelf of the Atlantic Ocean where water depths over 1,000m are common. Production of more than 100,000 barrels of oil per day in one single platform has already been established. In such conditions a better understanding of the vortex dynamics causing vibration of risers is essential.

It is already known that vortex shedding is a three dimensional phenomenon. However, two-dimensional simulations at low Reynolds number, as a first approximation to the problem, can be used to give some insight about the details of the vortex dynamics in the wake and vortex impingement occurring when there are a group of cylinders. Very few studies in the literature can be found about this subject. The attraction of applying numerical methods to such problem is that the flow can be studied in closer detail. In this sense, fundamental knowledge can be achieved performing a parametrical analysis of the phenomena in a relative fast way via CFD.

As can be seen in Zdravkovich(1987), when more than one bluff body is placed in a fluid flow, the resulting forces and vortex shedding pattern may be completely different from those found on a single body at the same Reynolds number. As the spacing or gap between two circular cylinders is varied, a range of flow regimes, characterised by the behaviour of the wake region, was observed by Bearman & Wadcock(1973) and Kim & Durbin(1988).

There are infinite numbers of possible arrangements of two parallel cylinders positioned at right angles to the approaching flow direction. Of the infinite arrangements, two distinct groups were selected: the first group comprises two cylinders in a tandem arrangement, one behind the other at some longitudinal spacing or gap; and the second group consists of a pair of cylinders facing the flow in a side by side fashion.

The interference phenomena are highly non-linear and at present beyond a reliable theoretical or computational analysis. The peculiar orientations presented here may serve as useful guides for the interpretation the predicted forces on cylinders array, mainly when a qualitative approach is required.

Following Zdravkovich(1977), it is a widespread practice to assume that two cylinders should behave in a flow in a similar, or even identical, manner to a single cylinder. This supposition is justified only when the two cylinders are sufficiently apart. Nevertheless, the interference between two cylinders at close proximity dramatically changes the flow around them and produces unexpected forces and pressure distributions. In addition, it intensifies or suppress vortex shedding. These changes of the flow pattern are also systematically described and analysed in this paper.

2. NUMERICAL METHOD

An explicit computational method was developed in order to investigate the flow around two cylinders at Reynolds number 200. An arbitrary Eulerian formulation following Meling and Dalheim(1997) is employed to incorporate the interface conditions between the body and the fluid. A Fractional step method with Galerkin finite element formulation was employed on unstructured meshes, and a velocity correction projection method is used to solve the Navier-Stokes equations. Following Manna(1997), the fractional step concept is more a generic approach than a particular method; there are many variations of it. The particular method shown here is described in detail in Siqueira et al (1999). In the present paper only an outline of the method is given.

The governing equations for a Newtonian, incompressible viscous flow are the conservation of mass and momentum. The non-dimensional Navier-Stokes equations may be written as follows

$$\frac{\partial u_i}{\partial x_i} = 0 \, , i = 1, 2 \tag{1}$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = - \frac{\partial p}{\partial x_i} + \frac{1}{\mathrm{Re}} \frac{\partial^2 u_i}{\partial x_j \partial x_j} \tag{2}$$

where the summation convention applies, Re = $U_\infty D / \nu$ denotes the Reynolds number, U_∞ is the free stream velocity, D the cylinder diameter and ν is the kinematic viscosity of the fluid. The outer boundary conditions for the velocity is the free-stream value and for the pressure is a prescribed value equal to P_∞. On the circular cylinder surface a no slip condition is applied, which implies that the fluid velocity is zero.

A fractional step formulation is applied for the time discretization of the governing equations (1-2). First, an intermediate velocity is computed by neglecting the pressure. The pressure field is then calculated by means of a Poisson equation, and the velocity field is finally corrected by including the pressure effect. For the purpose of determining a intermediate velocity a two-step Taylor-Galerkin formulation is employed.

Considering a time increment $\Delta t = t^{(n+1)} - t^{(n)}$, the fractional step method algorithm is given as follows, as can be seen in Meling and Dalheim(1997):

1) At time $t^{(n+1)}$ an intermediate velocity $\tilde{u}_i$ is calculated by means of an integration of the reduced momentum equation omitting the pressure.
2) The complete velocity u_i at $t^{(n+1)}$ is computed by including the pressure gradient. The pressure field is evaluated by solving the Poisson equation for pressure (see below).
3) The final velocity field is obtained in order to satisfy the continuity equation.

The computational steps 2 and 3 are responsible for the computation of the complete velocity field that satisfies the continuity equation are combined by means of a pressure equation (Poisson's equation). An alternative form of the algorithm could include the known pressure field for the prediction of the intermediate velocity and then computing the pressure difference instead of including the complete pressure field. However, as can be seen in Meling and Dalheim(1997), the removal of the pressure term has advantages since no spurious pressure modes are obtained. An equal order of interpolation can still be used for velocity and pressure.

2.1 Finite Element Formulation for the Spatial Discretization

A Galerkin finite element formulation is applied to the spatial discretization of the algorithm described previously. The well known shape functions are used to interpolate the velocity and pressure field. Applying a weighted residual formulation to the governing equations, one obtains a set of matrix equations. The linear systems that should be solved in the present method require an efficient, fast and accurate solver for sparse, symmetric and banded matrices. The Preconditioned Conjugate Gradient method with Diagonal Scalling was implemented in this work for this purpose.

2.2 Force Evaluation

Force coefficients are calculated by suitably integrating the pressure and skin friction contributions. After considering the contributions from skin friction and pressure, the force components are resolved in the two directions *(x,y)*, yielding F_x and F_y . These forces are then non-dimensionalised as follows:

$$Cl = \frac{2F_y}{\rho U_\infty^2 D} \tag{3}$$

$$Cd = \frac{2F_x}{\rho U_\infty^2 D} \tag{4}$$

where ρ is the fluid density, and U_∞ is the free stream velocity.

3. RESULTS AND CONCLUDING REMARKS

3.1 Code Validation

In all results shown in this paper, *Re*=200, with the Reynolds number defined in terms of the cylinder diameter (*D*) and free stream velocity (*U*), $Re=UD/\nu$. A non-dimensional time step, *Ut/D*, equal to 0·0050 has been used. Primarily, comparisons in the drag coefficient and Strouhal number for a single cylinder at Reynolds number 200 were carried out and the results compared to literature data in order to validate the computational method. An unstructured finite element mesh with 27062 elements and 13696 nodes has been used with a suitable boundary layer discretization. This mesh can be seen in figure 1 and details near the boundary layer are shown in figure 2. The extension of the outer boundary has been set in order not to cause any perturbation in the near wake flow.

The wake structure, represented by the streaklines, is given in figure 3. Lift and drag coefficients are shown in fig. 4. The average value of drag coefficient is 1.30 and the peak to peak value of the lift is 1.40. The Strouhal number for this case is 0.19. Those values are in very good agreement with literature data for this Reynolds number, as can be seen in table 1. Pressure contours near the body showing the stagnation region and vorticity contours are shown in figure 5. The vorticity values were non-dimensionalised by the cylinder diameter and free stream velocity.

3.2 Tandem arrangements

For both tandem and side by side arrangements, the distance or gap between the centres of the cylinders, which have the same diameter, were chosen as follows: 1.5D, 2D, 3D and 4D. Fundamentally, those values were selected in accordance with the conclusions reached by Zdravkovich(1987) and Bearman & Wadcock(1973), who stated that asymmetrical flow pattern can rise for a critical range of gap values, whereas at high distances the flow field regains its symmetry.

A typical computational unstructured finite element mesh for the tandem arrangement is shown in figure 6. The example given refers to a gap of 3 diameters and consists of 26064 elements and 13219 nodes. The figure 7 shows a detail of the mesh near the cylinder walls, stressing the importance of a suitably concentrated node distribution in the boundary layer.

Table 2 gives a summary of the mean average drag (Cd_m 1;2) and lift coefficients (Cl_m 1;2) for all the tandem arrangements carried out in this paper. Figure 8 gives a plot of the coefficients against non-dimensional time. The wakes, also represented by the respective streaklines, are shown in figure 9. Contours of vorticity non-dimensionalised by the cylinder diameter and free stream velocity are provided in figure 10.

For values of the gap between the centres of the cylinders up to 3D, it can be noticed that the pair of cylinders denote a behaviour close enough to a single circular cylinder. As also showed in Zdravkovich(1987), for a tandem arrangement three different flow regimes have been observed at low Reynolds numbers in the laminar state, referring to value of the gap between the centres of cylinders with same diameter from 1.5 to 4.0. From the gap between 1.5 to 3.0, the vortex street behind the latter cylinder is actually formed by the free shear layers detached from the former. As the gap increases, the free shear layers separated from cylinder 1 reattach on the upstream side of cylinder 2. A vortex street is formed only behind the downstream cylinder 2. From gaps greater than 3.0, the separated shear layers roll up alternately and form vortices behind cylinder 1 in front of the downstream cylinder 2. Two vortex streets are formed behind the cylinders.

The drag coefficient for gap values up to 3D is positive for cylinder 1 and negative for cylinder 2, indicating that a repulsion force may rise between the cylinders. The drag coefficient for cylinder 2 tends to become less negative and the amplitude of the oscillation increases as the gap rises. It can also be noticed in Zdravkovich (1987). When the gap is increased to 4D both cylinders denote oscillation in the lift coefficient (with a higher amplitude for the cylinder 2) and the drag coefficient in cylinder 2 becomes positive, even with a small value.

3.3 Figures and Tables

Table 1. Average drag coefficients and Strouhal numbers for a single cylinder at Re=200. Comparison with literature data.

	S	Cd_{av}
Current Fractional Step Method	0.19	1.30
Literature Results		
Meneghini (1993)	0.196	1.25
Arkell (1995)	0.196	1.30
Giannakidis (1997)	0.19	1.25
Saltara (1998)	0.195	1.30
Experiments:		
Roshko (1954)	0.17-0.19	
Williamson (1991)	0.196	
Norberg (1993)		1.30

Table 2. Average Drag and Lift coefficients for the Tandem configuration at Re=200.

Average Force Coefficients Tandem Configuration		
Gap	**Cd_m 1**	**Cd_m 2**
1.5D	1.06	-0.18
2.0D	1.03	-0.17
3.0D	1.0	-0.08
4.0D	1.18	0.38

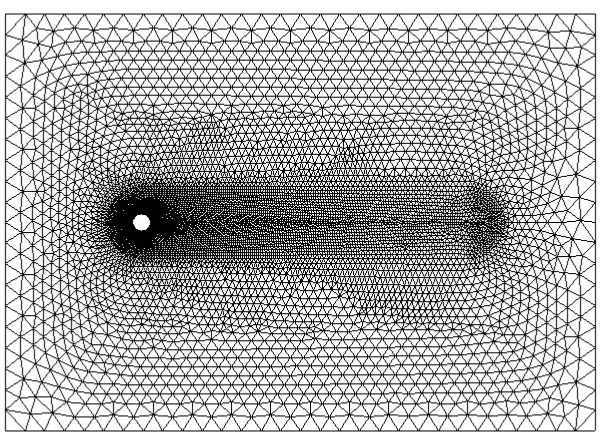


Figure 1. Unstructured finite element mesh for a single cylinder.

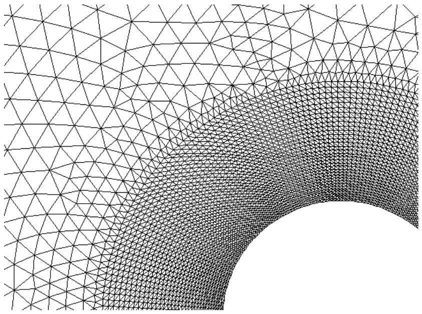


Figure 2. Details of the mesh used in the boundary layer.

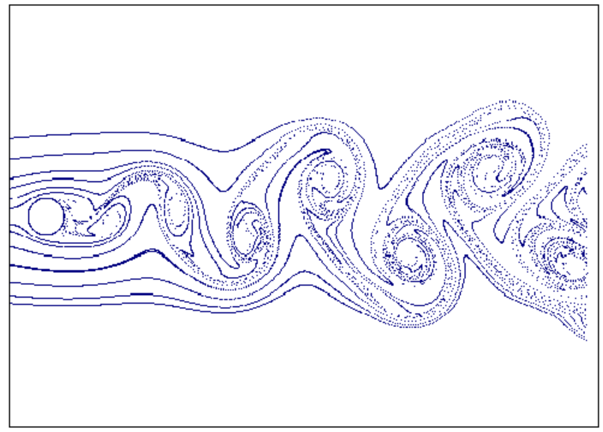


Figure 3 - Wake structure for a single circular cylinder at Re=200.

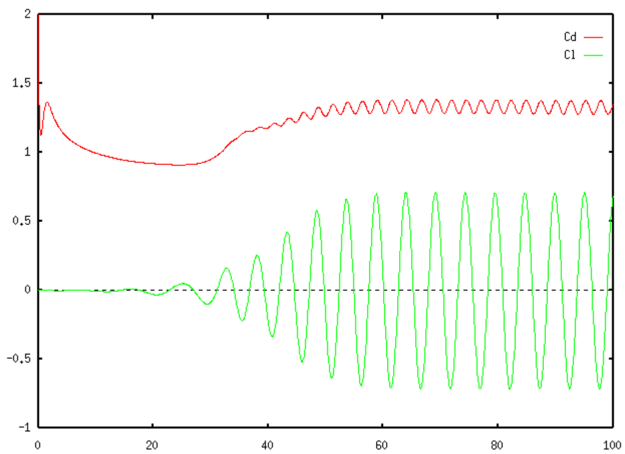
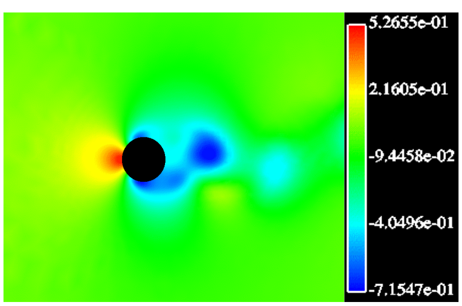
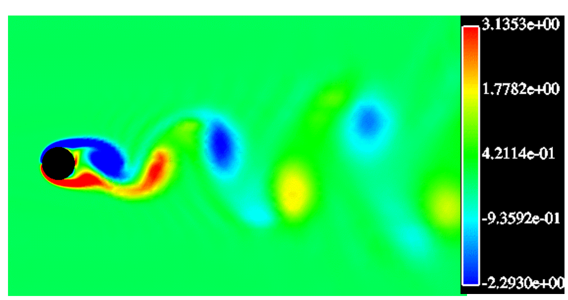


Figure 4 - Force coefficients for *Re*=200. Single Cylinder. The abscissa is the non-dimensional time (*Ut/D*).



a)



b)

Fig. 5 - Pressure contours (a) and Vorticity contours (b) for a single cylinder (Re=200).

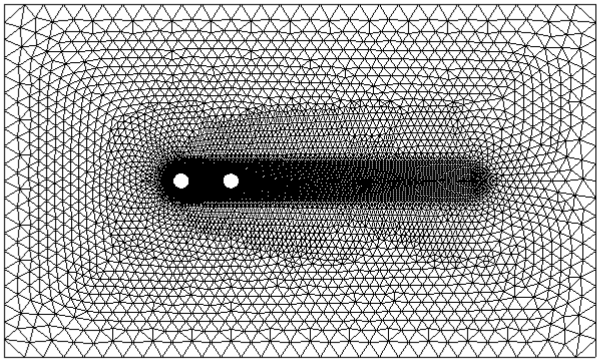


Fig. 6 - Typical unstructured finite element for the tandem arrangement. Distance between the centres equal to 3D.

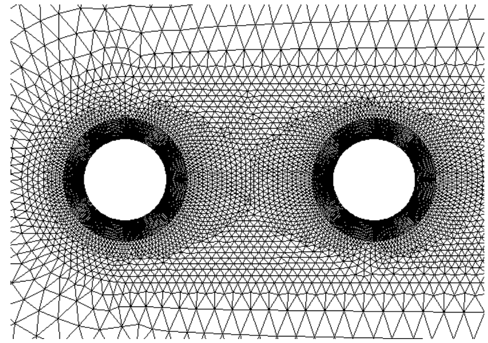


Fig. 7 - Detail of the mesh near the wall for both cylinders. Distance between the centres equal to 3D..

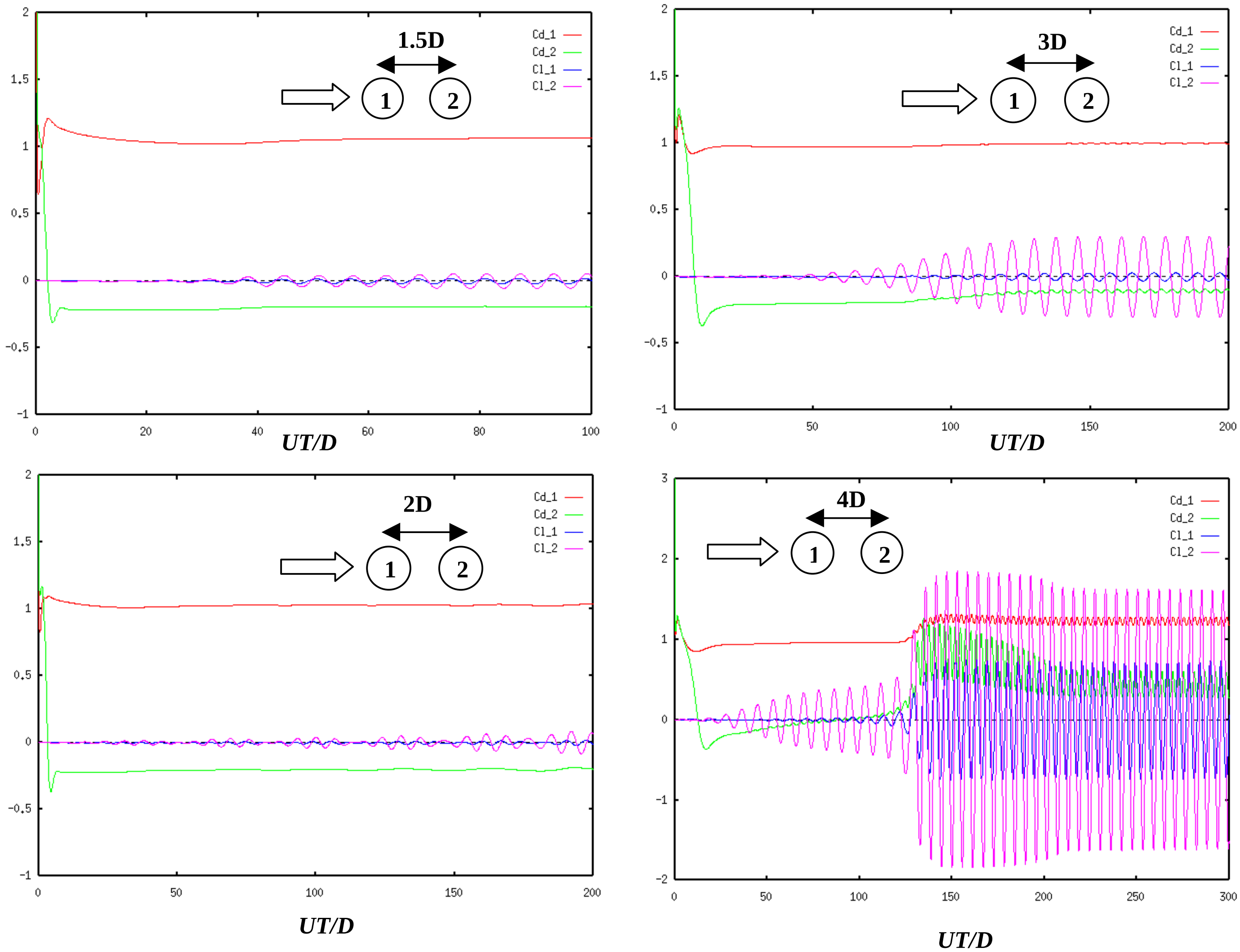


Fig. 8 - Force coefficients against non-dimensional time. Tandem arrangement (Re=200).

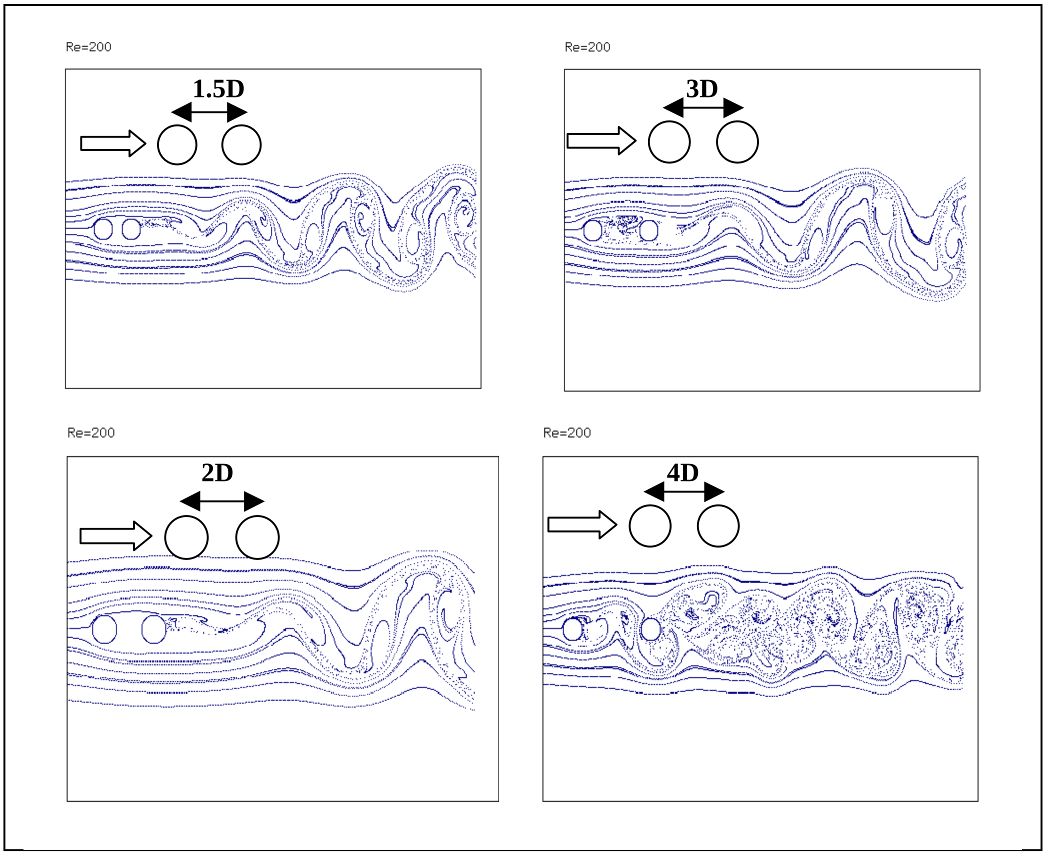


Fig. 9 - Wake represented by streaklines. Tandem arrangement (Re=200).

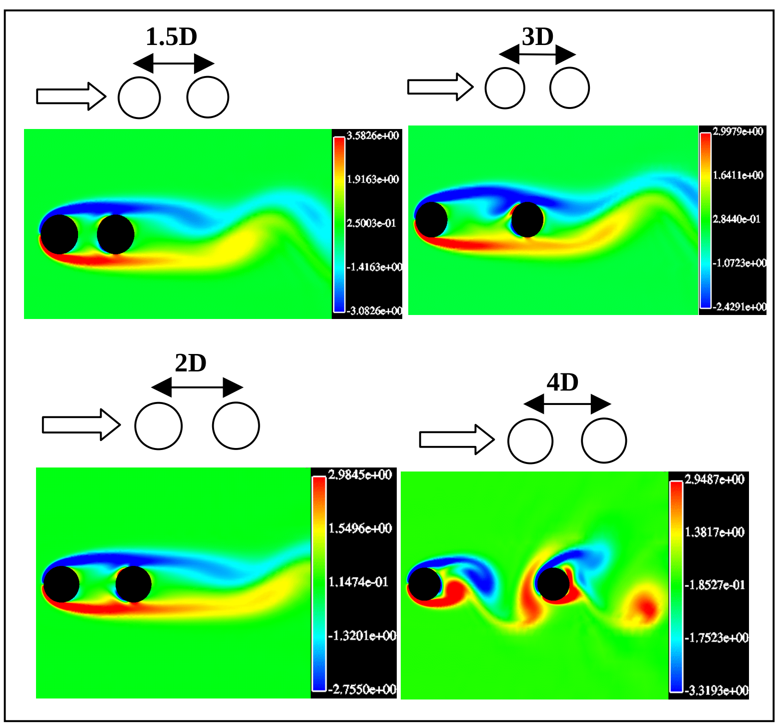


Fig. 10 - Vorticity contours. Tandem arrangement (Re=200).

4. Acknowledgments

The authors are grateful to FAPESP (grant 94/3057-0 and 94/3528-3) and PETROBRAS (The Brazilian State Oil Company) for providing them a Research Grant for this project and CNPq for a partial Grant. The LCCA (USP – Brazil) provided facilities for some flow visualisations.

5. References

Arkell, R. H. (1995*) Wake dynamics of cylinders encountering free surface gravity waves, PhD thesis*, University of London, UK.

Bearman, P.W.; Wadcock, A.J. (1973). *The interaction between a pair of circular cylinders normal to a stream*. Journal of Fluid Mechanics, vol. 61, pp 499-511.

Giannakidis, G., J.M.R. Graham. (1997) "Prediction of the loading on a Horizontal Axis Wind Turbine Rotor including effects of stall" European Wind Energy Conference.

Graham, J. M. R., (1988) Computation of viscous separated flow using a particle method, *Numerical Methods in Fluid Mechanics*, Vol.3, pp. 310-317,Ed. K.W. Morton, Oxford University Press.

J. Zhang and C. Dalton, (1994). Interaction of a steady approach flow and a circular cylinder undergoing forced oscillation. Proc. Int. Conf. on Offshore Mech. And Artic. Engng. Vol.1, pp.117-122. Houston, Texas

Kim, H.J.; Durbin, P.A. (1988). *Investigation of the flow between a pair of circular cylinders in the flopping regime*. Journal of Fluid Mechanics, vol. 196, pp.431-448.

Manna, M. (1997). *Introduction to the Modelling of Turbulence*, von Karman Institute for Fluid Dynamics, Lecture Series 1997-03, Belgium.

Meling, T. S., Dalheim, J., (1997). Numerical Prediction of the Response of a Vortex-Excited Cylinder at Low Reynolds Numbers, 7th International Offshore and Polar Engineering Conference (ISOPE 97), Honolulu, USA.

Meneghini, J. R., (1993). Numerical Simulation of Bluff Body Flow Control Using a Discrete Vortex Method, *PhD thesis*, University of London, UK.

Norberg, C. (1993). Private communication.

Roshko, A. (1954). *On the drag and shedding frequency of two dimensional bluff bodies*. Technical note 3169, National Advisory Committee for Aeronautics (NACA), July 1954, Washington.

Saltara, F. (1998), private communication.

Sumner, D.; Price, S.J.; Païdoussis, M.P. (1998). *Investigation of side-by-side circular cylinders in steady cross-flow by Particle Image Velocimetry*, Proc. of the1998 ASME Fluids Eng. Division Summer Meeting (FEDSM 98).

Williamson, C. H. K., (1991). *2-D and 3-D Aspects of the Wake of a Cylinder, and their Relation to Wake Computations*, C. R. Anderson and C. Greengard editors, Vortex Dynamics and Vortex Methods. Lectures in Applied Mathematics, 719-751.

Williamson, C.H.K. and Roshko, A., (1988) Vortex formation in the wake of an oscillating cylinder, *J. Fluids and Structures*, Vol. 2, pp 355-381.

Zdravkovich, M.M. (1987). *The effects of interference between circular cylinders in cross flow*. Journal of Fluids and Structures, N° 1, pp. 239-261.

Zdravkovich, M.M. (1977). *Review of flow interference between two circular cylinders in various arrangements.* ASME Journal of Fluids Engineering, vol. 99, pp. 618-633.