Comparative Numerical Simulation of Two-Dimensional Flows over a Pair of Circular Cylinders, Disposed in Tandem, using Immersed Boundary and Lattice-Boltzmann Methods

A. L. F. Lima e Silva [1] , R. Surmas [2], L.O. Emerich dos Santos[2], P. C. Philippi[2] and A. Silveira-Neto[1]

Summary

This paper presents a two-dimensional numerical simulation of fluid flow around a pair of circular cylinders of different diameters, disposed in tandem. at several distances. An *Immersed Boundary (IB)* and a *Lattice-Boltzmann (LB)* Method were used. Simulations were performed for Reynolds Number equal to 200. Strouhal number, the drag and the lift coefficients are obtained with the referred methods and compared with others existing results.

Introduction

The flow interference between two circular cylinders has been studied in different hydrodynamic problems. The interaction between the upstream cylinder and the downstream cylinder wakes changes the flow behavior. The distance between the cylinders is an important parameter on the vortex shedding. In IB method, the force field is obtained using the *Physical Virtual Model* (*PVM*) [4]. This model is new and based on the momentum equations, enabling to calculate the force field over the immersed boundary using, only, the properties of the neighboring fluid. It avoids adjusting constants and only a simple interpolation scheme is used to obtain the parameters over the interface in modeling fluid-body interaction. In Lattice-Boltzmann simulation [1] the macroscopic variables are obtained by up-scaling the mesoscale particles distribution function, which gives the number N_i of particles, populating direction i of a given site X, at time T and which follows discrete lattice-Boltzmann equation. In this way, numerical LB simulation gives N_i (X, T): velocity, u (X, T), and pressure, P(X, T), fields are obtained as derived quantities.

Immersed Boundary Method

An Eulerian-Lagrangean formulation is used to represent the flow and the immersed boundary. A Cartesian grid, describes the flow using a Finite

[1] Federal University of Uberlândia, Uberlândia, Minas Gerais, Brazil
[2] Federal University of Santa Catarina, Florianópolis, Santa Catarina, Brazil

Difference Method and a Lagrangean grid, describes the immersed body. The momentum and mass conservation equations for an incompressible, two-dimensional viscous flows can be written as:

$$\rho\left[\frac{\partial \vec{V}}{\partial t} + \left(\vec{V}.\vec{\nabla}\right)\vec{V}\right] = -\vec{\nabla}P + \mu\nabla^2\vec{V} + \vec{F}\ , \qquad \vec{\nabla}.\vec{V} = 0\ , \tag{1)-(2}$$

where $\vec{F}$ is given by

$$\vec{F}_{ij} = \sum D_{ij}\vec{f}_k \Delta s^2\ , \tag{3}$$

where $\vec{f}_k$ is a Lagrangean force density and $\vec{F}(\vec{x})$ is the Eulerian force that is not equal to zero only over the immersed boundary. The distribution function, D_{ij}, proposed by Juric [2], was used in order to calculate the Eulerian force, in a discrete form. Δs is the distance between the Lagrangean points.

In the present work, the *Physical Virtual Model* (*PVM*) was proposed to calculate the Lagrangean force field, based only upon momentum equation. All the Navier-Stokes terms are calculated over the Lagrangean points. Therefore the force $\vec{f}(\vec{x}_k)$ should be expressed by:

$$\vec{f}(\vec{x}_k) = \vec{f}_a(\vec{x}_k) + \vec{f}_i(\vec{x}_k) + \vec{f}_\upsilon(\vec{x}_k) + \vec{f}_p(\vec{x}_k). \tag{4}$$

where these forces components are given by:

$$\vec{f}_a(\vec{x}_k) = \rho\frac{\partial \vec{V}}{\partial t}(\vec{x}_k), \qquad \vec{f}_i(\vec{x}_k) = \rho\left(\vec{V}.\vec{\nabla}\right)\vec{V}(\vec{x}_k), \tag{5)-(6}$$

$$\vec{f}_\upsilon(\vec{x}_k) = -\mu\nabla^2\vec{V}(\vec{x}_k), \qquad \vec{f}_p(\vec{x}_k) = \vec{\nabla}p(\vec{x}_k). \tag{7)-(8}$$

The equations were approximated using the *Finite Difference Method* and the *Fractional Step Method* was used for their solution as described in [4]. The Linear system for the pressure correction, was solved by the *Modified Strongly Implicit Procedure* (*MSI*), [5].

Lattice-Boltzmann Method

Presently used lattice-Boltzmann method was based on the single relaxation-time Lattice-Boltzmann equation ([1], [6], [7] e [8]),

$$N_i(\mathbf{X}+\mathbf{c}_i,T+1)-N_i(\mathbf{X},T)=\Omega_i=\frac{N_i^o(\rho,\mathbf{u})-N_i(\mathbf{X},T)}{\tau}, \tag{9}$$

appropriated to low-Mach number, incompressible flows. In Eq. (9), X designates a site, T is the time step, i is an arbitrary lattice direction, c_i is the unitary velocity vector pointing to direction *i*. Eq. (9) means that, at site X and time T, the distribution N_i will be affected by collisions, modeled by a Bhatnagar-Gross-Krook (BGK) collision term in the right hand side and written as proportional to the difference between the N_i value at site X and the value this distribution would have in equilibrium. Relaxation time τ is related to the fluid kinematics viscosity, ν. After collision, the new value $N_i'(X,T)$ is propagated to next, direction-i, neighbor site $X+c_i$ and this will be the value for the distribution of particles in that site, at time $T+1$. LB method is, intrinsically, parallel, since at each time step, particles distribution at a site X depends, only, on the values of N_i in the next-neighbors sites, at the previous time step T-1.

Boundary conditions at solid surfaces are simulated by using the bounce-back condition

$$N_{-i}(\mathbf{X}_b,T+1)=N_i(\mathbf{X}_b,T), \tag{10}$$

for a boundary site X_b in the fluid, when direction i points to the solid surface. This assures the adherence condition $u=0$ at the boundary.

When N_i particles are reflected back at boundary sites X_b, they exchange with the solid surface the momentum $2N_i c_i$, since lattice-Boltzmann particles have unitary mass. Considering that this exchange is performed in each unitary time step, this momentum exchanged represents a force F_i, on the solid surface adjacent to site X_b. In this way, the total force on the solid body can be calculated by.

$$\mathbf{F}=\sum_{X_b}\sum_i \mathbf{F}_i(\mathbf{X}_b). \tag{11}$$

Numerical Results

IB simulations have been carried out for the domain shown in Fig. 1a, which shows a global view of the domain with the main geometrical characteristics. Figure 1b shows a closer view of the cylinders with the Cartesian and the Lagrangean grid. The simulations were done for the distances between the cylinders centers L = 1.5d, 2d, 2.5d, 3d and 3.5d. The Reynolds number is equal to 200 and the flow direction is from the bottom to the top of the domain. The upstream and the downstream diameters are d and $d/2$, respectively. LB simulation was performed on a two-dimensional projection of a face centered hyper-cubic lattice, FCHC, Fig. 1c, [9].

The vorticity contours can be seen in Fig. 2 for the different distances between the cylinders.

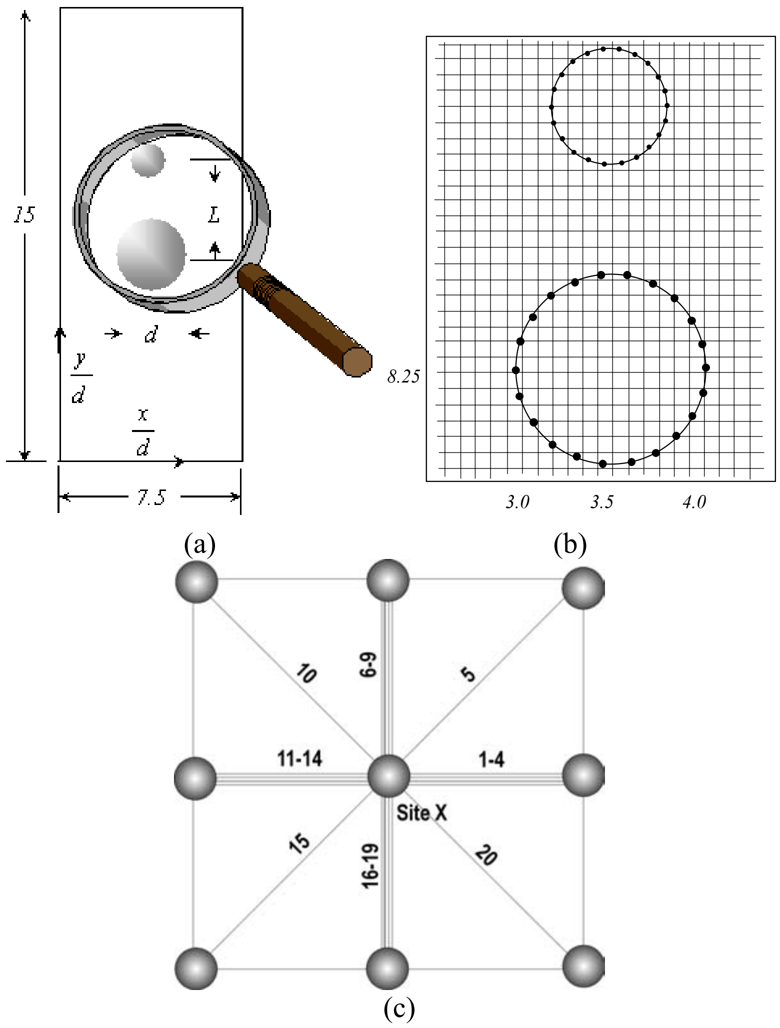

Figure 1. Illustration of the domain used in the simulations: (a) global view and (b) closer view with the two circular cylinders for the IB simulation and (c) LB elementary cell simulation.

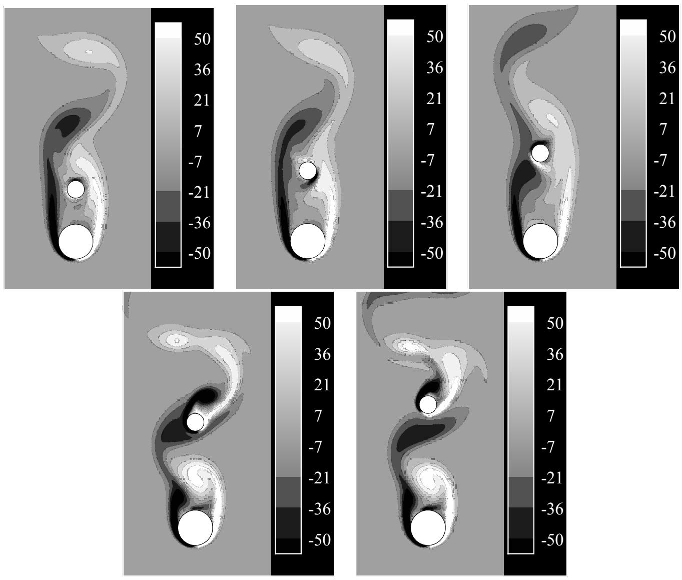

Figure 2. Vorticity Contours for a gap between the centers of L = 1.5d, 2d, 2.5d, 3d and 3.5d, respectively.

The mean drag coefficient and the Strouhal number were obtained using the force field calculated over the immersed body. The drag coefficient as a function of the distance between the cylinders is shown in Fig 3a. In this figure, index *1* and *2* refers to the upstream and downstream cylinder, respectively.

The Strouhal number was obtained by *Fast Fourier Transformer* (*FFT*) of the lift coefficient distribution and plotted as a function of the distance between the cylinders in Fig. 3b. It has the same value for both cylinders because it was calculated with the upstream cylinder diameter. The IB and the LB results were compared with a Finite Volume Method, Meneguini *et al*. [10].

Conclusions

The Immersed Boundary and the Lattice-Boltzmann methods were used to simulate a complex flow and compared with a Finite Volume method. The results obtained with different methods agree very well and show that these methods are potential tools to analyse complex flows without the difficulty present in classical methods, like boundary conditions establishment and remeshing process. In the present work the results are related with a body that is not in movement. There are no theoretical limitations to apply these methods to analyse flows over mobile immersed boundary. It is also important to emphasise that a Cartesian mesh is being used to simulate flows over complex geometry.

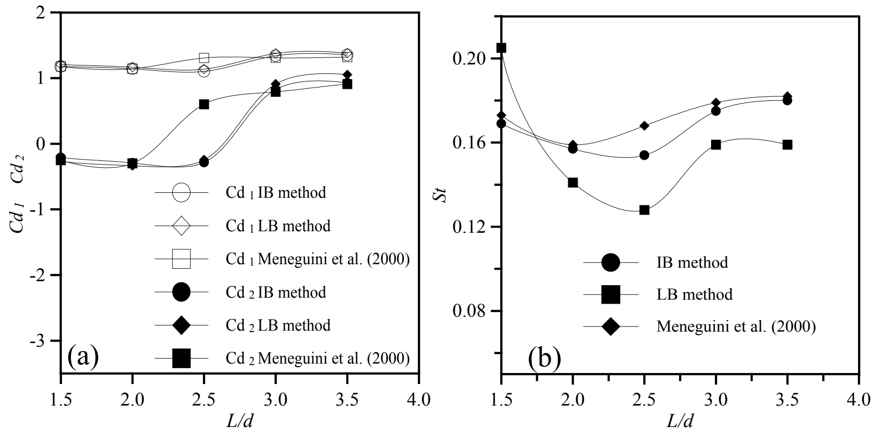

Figure 3. (a) Mean drag coefficient and (b) Strouhal number as a function of the gap between the cylinders centres.

References

1 McNamara G. G. and Zanetti G., 1988, “Use of the Boltzmann Equation to Simulate Lattice-Gas Automata” *Physical Review Letters* 61, 2332-2335.

2 Juric, D., 1996, “Computation of phase change”, Ph. D. Thesis, Mech. Eng. Univ. of Michigan, USA.

3 Armfield, S. e Street, R., 1999, “The Fractional-Step Method for the Navier-Stokes Equations on Staggered Grids: The Accuracy of three Variations”, J. Comp. Phys, 153, 660.

4 Lima e Silva, A. L. F., 2002, Desenvolvimento e Implementação de uma Nova Metodologia para Modelagem de Escoamentos sobre Geometrias Complexas: Método da Fronteira Imersa com Modelo Físico Virtual, Doctorate Thesis.

5 Schneider, G. E. e Zedan, M., 1981, “A Modified Strongly Implicit Procedure for the Numerical Solution of Field Problems”, Numerical Heat Transfer 4, 1.

6 Higera, F. and Jimenez, J., 1990, “Boltzmann approach to lattice gas simulations.” *Europhys. Lett.* 9, 663-668.

7 Qian, Y. H., d’Humières D., Lallemand P., 1992, “Lattice BGK Models for Navier- Stokes Equation” *Europhys. Lett.* 17, 479-484.

8 Zou, Q. and He, X., 1997, “On pressure and velocity boundary conditions for the lattice Boltzmann BGK model” *Phys. Fluids* 9, 1591-1598.

9 d’Humières, D., Lallemand, P., and Frisch, U.,1986, “Lattice gas models for 3D hydrodynamics” *Europhys. Lett.* 2, 291-297.

10 Meneguini, J. R., Flatschart, R. B., Saltara, F., Siqueira, C. L. R. e Ferrari Jr., J. A., 2000, “Numerical Simulation of Flow Interference Between two Circular Cylinders in Tandem”, 19th International Conference on Offshore Mechanics and Artic Engineering, February 14-17, New Orleans, La, USA.