NUMERICAL SIMULATION OF THE FLOW PAST AN OSCILLATING CONFINED CYLINDER USING THE PHYSICAL VIRTUAL MODEL

Débora Gleice da Silva Del Rio Vieira, vieira@fnb.tu-darmstadt.de
Graduate School of Computational Engineering der Technischen Universität Darmstadt, Dolivostraße 15, Darmstadt - Germany

Augusto Salomao Bornschlegell, augusto.s.b@gmail.com
Universidade Estadual Paulista Júlio de Mesquita Filho, Av. Brasil centro, 56, Ilha Solteira-SP, Brasil.

Sérgio Said Mansur, mansur@dem.feis.unesp.br
Universidade Estadual Paulista Júlio de Mesquita Filho, Av. Brasil centro, 56, Ilha Solteira-SP, Brasil.

Julio Militzer, julio.militzer@dal.ca
Dalhousie University, 1360 Barrigton St., Halihax –NS, Canadá

Abstract. *Diverse Immersed Boudary Methods were developed along the years since the first model proposed for Charles S. Peskin in early 70´s. Many authors applied modifications on the original force calculation intending a suitable numerical representation of the solid fluid interface. The Physical Virtual Model is one of the Immersed Boundary Methods which use the continuous forcing approach. This method was chosen in the present work for simulating the flow over a cylinder. This model was implemented on a solver for Navier Stokes using Finite Volume Method, Central Difference Scheme and SIMPLEC. Two-Dimension simulations were performed for moderate Reynolds number from 60 to 200. Drag and Lift coefficients were investigated and compared to results found on the literature. The Physical Virtual Model performance was verified for an oscillating confined cylinder. Also fluid structure interaction was verified considering a solid cylinder moving due to a force balance between drag and weight. Advantages and disadvantages of the model were considered on the present work.*

Keywords*: Immersed Boundary, Physical Virtual Model, fluid structure interaction, oscillating cylinder.*

1. INTRODUCTION

Typical engineering problems are commonly not correctly represented by simple geometries. This leads some difficulties when dealing with flow simulation, once to represent numerically a complex boundary is not an easy task. One can find diverse tools available on the literature intending a correct domain representation. Each one of these tools has its advantages and disadvantages and was developed for some specified kind of application, due to its author´s needs.

Once dealing with a numerical simulation, a solid body immersed in a flow can be described by different approaches. Traditional methods are based on prescribed boundary conditions on the fluid-solid interfaces, or yet, consider a very high viscosity inside the solid region. The Immersed Boundary Methods, or just IB Methods, were originally developed by Charles S. Peskin for simulating the flow over a human heart valve, Peskin (1977). Once this problem consists of a complex moving and elastic boundary, IB Methods are an attractive alternative in numerical simulation due to the use of only a simple Cartesian grid representing the whole domain, and also a Lagrangian mesh representing the boundaries. The big advantage of this approach is the fact that the Lagrangian points do not need to be conformed to the Cartesian mesh. A force is added in each Lagrangian point for representing numerically the body on the flow.

Nowadays, several methods found in the literature derivate from the original model proposed by Peskin . They keep the same idea and differ each other basically in the form of evaluation of the forcing term, responsible for indicate the presence of the body. Basically, two groups are found: the first one consider the forcing term before the discretization process and it is called the Continuos Forcing Methods; the second groups consider the forcing term after the discretization and it is called the Discrete Fording Methods. For any case, the mathematical expression for the forcing term evaluation is only an approximation and one should consider the characteristics of a specific problem before choosing about evaluating the force before or after the domain discretization. When choosing an IB from a Discrete Forcing Method, one should consider that the Lagrangian points are usually not coincident with the mesh nodes, so that, it is also necessary to define the exact points where the force is going to be applied.

About the discrete forcing methods, diverse applications of IB Methods can be found in the literature for solving low and moderate Reynolds number flows, but the use of a forcing term for high Reynolds numbers is still a challenging task. When dealing with turbulent flows, it is necessary a better grid resolution near the boundary, because for these cases, the local precision is extremely important and the diffusive effects of the force distribution are not desirable. For this reason, some approaches consider the IB like an interface where the computational domain is modified and some boundary condition is directly applied.

The well known Ghost-Cells method defines internal cells which have at least one neighbor on the immersed boundary. In these cells associate values of some generic variable are locally extrapolated using a function obtained with the values of the boundary conditions. By the use of this function, it is possible the non direct utilization of the contour variables.

Another alternative for the boundary cells is the Finite Volume with Cut Cells. This method was first introduced using a Cartesian mesh for a non viscous flow by Clarke *et al*. (1986). Afterwards it was also used by Udaykumar *et al*. (1996), (2001), (2002), and Ye *et al.* (1999). The main idea consists on re-shaping the contour cells and evaluating the local fluxes. The new cells are fitted to the boundary surface. This way, the boundary condition imposition is satisfied. The advantage of this method is the strong mass and momentum conservation.

Even thought the discrete forcing methods need some mesh modification near the Immersed Boundary, they are still not suitable for high Reynolds numbers, according to Mittal & Iaccarino (2005).

The second group of IB methods, as mentioned before, includes the forcing term $\underline{f}$ on the Navier-Stokes equations before the domain discretization. The domain can be than divided in a simple mesh, like a Cartesian mesh. In the first model, developed by Peskin, the flow around a heart valve was investigated, considering a group of elastic fibers composed by Lagrangian points which were moved by the local fluid velocity. A distribution function was used despite of the Delta Dirac function required, suitable for a discrete domain. This model can also be used for modeling a solid interface by considering a high elasticity constant, but this would lead into a system of equation with several restrictions, Mittal & Iaccarino (2005).

The authors Beyer & Leveque (1992) e Lai & Peskin (2000), considered a fixed structure attached to some equilibrium point by a group of springs, represented by a restoring force. This strategy leads to a bad conditioned system of equations, and many time steps are required until some physical solution can be achieved.

Goldstein et al. (1993) present a particular case of the original solution proposed by Peskin. In this case, it was necessary the adjustment of two constants, α and β, for that the problem can reach convergence.

Some recent modification of the model proposed by Goldstein et al. (1993) was suggested by Góis (2007), when using only one *ad hoc* parameter in Goldstein previous model. The authors use multigrid model for solving the Poison equation and compare the results of the lock-in phenomena with experimental results of other authors.

Diverse other ways of force evaluating on IB Methods can be found in the literature. The physical Virtual Model was chosen in the present work. It consists on the force evaluating based only on the solid fluid interaction. The force is evaluated for every Lagrangian point and distributed to the neighbor control volumes due to interpolation functions, better explained in the following item.

2. MATHEMATICAL FORMULATION

The isothermal non-compressible Newtonian flow can be represented by the Navier-Stokes equations and the mass conservation. For a two-dimensional Cartesian domain:

$$\frac{\partial u_m}{\partial x_m} = 0 \tag{1}$$

$$\rho\left[\frac{\partial u_m}{\partial t} + \frac{\partial u_m\, u_n}{\partial x_n}\right] = \frac{\partial p}{\partial x_m} + \frac{\partial}{\partial x_n}\left[\mu\left(\frac{\partial u_m}{\partial x_n} + \frac{\partial u_n}{\partial x_m}\right)\right] \tag{2}$$

where ρ is the fluid densisty, μ is the dynamic viscosity, p is the pressure, u_m and u_n are velocity components. To numerically represent an immersed body in a flow, one can add a forcing term to the Navier-Stokes equations given by F_m., as it follows:

$$\rho\left[\frac{\partial u_m}{\partial t} + \frac{\partial u_m u_n}{\partial x_n}\right] = \frac{\partial p}{\partial x_m} + \frac{\partial}{\partial x_n}\left[\mu\left(\frac{\partial u_m}{\partial t} + \frac{\partial u_n}{\partial x_n}\right)\right] + F_m \tag{3}$$

When properly evaluated, this force causes the flow to go around the complex geometry by the IB represented without any local boundary condition. According to Lima e Silva (2002), the forcing term is evaluated like a function of the position vector $\boldsymbol{X}_k$, and the time t. This force can also be regrouped in a combination of an acceleration force $\boldsymbol{Fa}$, an inertial force $\boldsymbol{Fi}$, a pressure force $\boldsymbol{Fp}$,and a viscous force , which can be evaluated in each one of the Lagrangian points and interpolated to the Eulerian mesh.

$$\boldsymbol{F}(\boldsymbol{X}_k,t) = \boldsymbol{Fa}(\boldsymbol{X}_k,t) + \boldsymbol{Fi}(\boldsymbol{X}_k,t) + \boldsymbol{Fp}(\boldsymbol{X}_k,t) + \boldsymbol{Fd}(\mathbf{X}_k,t) \tag{4}$$

Where

$$\boldsymbol{Fa}(\boldsymbol{X}_k,t)=\rho\frac{\partial u}{\partial t} \quad (5)$$

$$\boldsymbol{Fi}(\boldsymbol{X}_k,t)=\rho(u\,\nabla u) \quad (6)$$

$$\boldsymbol{Fp}(\boldsymbol{X}_k,t)=\nabla p \quad (7)$$

$$\boldsymbol{Fd}(\boldsymbol{X}_k,t)=-\nabla(2\,\mu\,d) \quad (8)$$

All these terms are evaluated on the interface, using the pressure and velocity fields interpolated from the Eulerian mesh. The acceleration force was evaluated by:

$$\boldsymbol{Fa}^{n+1}\left(\boldsymbol{X}_k^{n+1},t\right)=\rho\left(\frac{\boldsymbol{V}^{n+1}-\boldsymbol{u}_{fk}^{n}}{\Delta t}\right) \quad (9)$$

Where:

$$\boldsymbol{u}^{n+1}\left(\boldsymbol{X}_k^{n+1}\right)=\boldsymbol{V}_k^{\;n+1} \quad (10)$$

$$\boldsymbol{u}^{n}\left(\boldsymbol{X}_k^{n+1}\right)=\boldsymbol{u}_{fk}^{n} \quad (11)$$

$\boldsymbol{V}^{n+1}$ is the velocity of the solid at the time t^{n+1} and $\boldsymbol{u}_{fk}^{n}$ is the velocity of the fluid at the time t_n at the point $\boldsymbol{X}_k^{n+1}$. The inertial force, pressure force and viscous force were evaluated by an expression which contains the *x* and *y* of the velocity and pressure in the interface. Lagrangian polynoms were used for evaluating *u, v* and *p* on the normal directions. Two auxiliary points were used to determine the polynoms,which were obtained by parallel lines traced from the interface, as illustrated on Figure1.

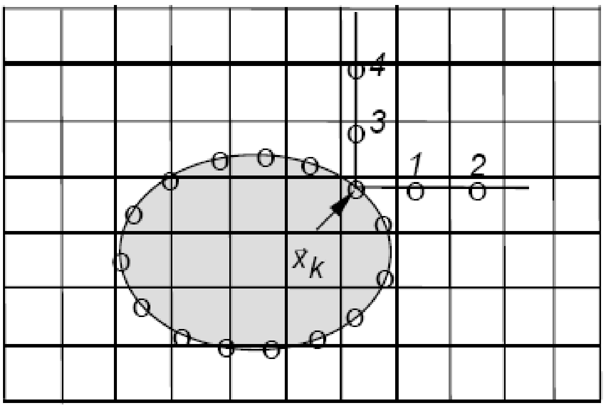


Figure 1: Point for interpolation of velocity and pressure fields, by Lima e Silva (2002).

The points x_1 e x_2 ,are separated by the same distance Δx from each other and, x_3 and x_4, by a distance Δy. The procedure consists on tracing two perpendicular lines for each fixed point *k* over the interface and defining these points for proprieties interpolation. The velocity components are evaluated on these points using the Navier-Stokes equations and the Lagrangian polymons are used for evaluating the first and second derivatives.

For evaluating the velocities on the interface, it is used:

$$u_i=\int_\Omega u(x)\;\delta(x-x_i)\;\Delta x\,\Delta y \quad (12)$$

$$v_i=\int_\Omega v(x)\;\delta(x-x_i)\;\Delta x\,\Delta y \quad (13)$$

Pressure and velocity are evaluated and distributed to the interface using an approximation function for the Delta Dirac function as follows:

$$\delta\left(x-x_i\right)=\frac{D\left(\frac{x-x_i}{\Delta x}\right)\ D\left(\frac{y-y_i}{\Delta y}\right)}{\Delta x\ \Delta y} \tag{14}$$

Afterwards the pressure is evaluated like:

$$p_i=\frac{\int_\Omega p\left(x\right)\ \delta\left(x-x_i\right)\ \left(1-I\left(x\right)\right)\ \Delta x\ \Delta y}{\int_\Omega \delta\left(x-x_i\right)\ \left(1-I\left(x\right)\right)\ \Delta x\ \Delta y} \tag{15}$$

Where I is an indicative function, for determining the region inside and outside the immersed boundary.

The *in-house* code named Fluids, used for performing the numerical simulations, was initially developed by Campegher (2002). Fluids solves the Navier-Stokes equations in two dimensional Cartesian coordenadinates using the Finite Volume Method with time implicit discretization. For the present work, the authors have chosen the SIMPLEC model for the velocity-pressure coupling and Central Difference for the advective terms treatment.

To avoid numerical instabilities, a dumping function was implemented in the outlet of the computational domain according to Góis (2007). This function multiplies the vorticity for another ramp function with range from 0 up to 1:

$$w_z\left(x,y\right)=f_2\left(x\right)w\left(x,y,t\right) \tag{16}$$

Where $w(x,y,t)$ is the disturbed vorticity and $f_2(x)$ is the ramp function. As suggested by Souza (2005), the ramp function can be used as bellow:

$$f_2(x)=f\left(\varepsilon\right)=1-6\varepsilon^5+15\varepsilon^4-10\varepsilon^3 \tag{17}$$

When

$$\varepsilon=\frac{\left(i-i_3\right)}{i_4-i_3} \tag{18}$$

and $i_3 \le i \le i_4$, corresponding points at the *x* direction.

3. RESULTS AND DISCUSSION

Figure 1 shows stream lines of a periodic flow over a cylinder positioned after two solid barriers. The cylinder was modeled using the Physical Virtual Model and the solid obstacles were modeled imputing a very high viscosity inside the control volumes which they are composed. The results show a good behavior of the immersed boundary responding to the different velocities applied. The immersed boundary needs some time steps that the forcing term can be correctly evaluated by the numerical solver and the flow can recognize the presence of a solid body. For moderate Reynolds number, the time necessary for that the flow can move itself around the cylinder and not thought the boundary is smaller than the same case for a low Reynolds number. When simulating a pulsate flow, the immersed boundary forcing term evaluation presents also better results when the inlet velocity is maximum. For very low frequencies, after the minimum inlet velocity, for each cycle, a few time steps are needed that the flow can come around the body again, and not thought the boundary.

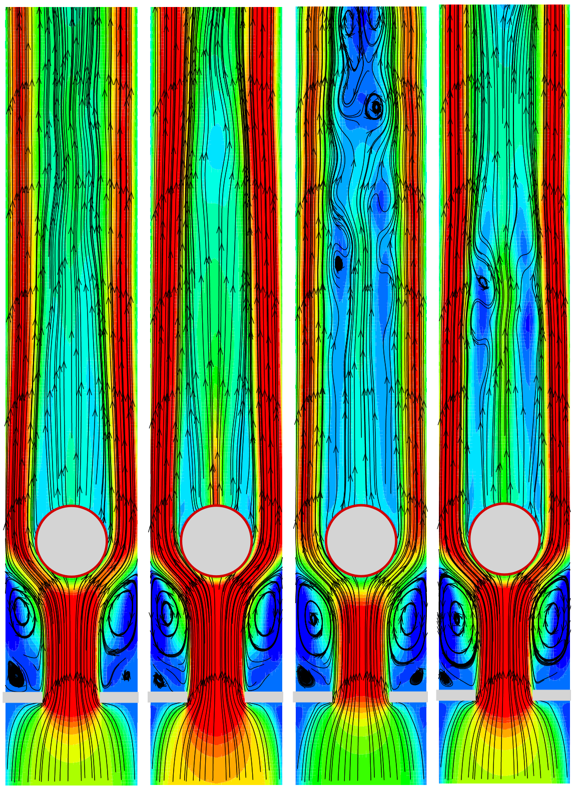
Results for simulations of the confined cylinder show that for high oscillation specific frequencies, the code becomes very instable, leading to divergence of the results.

Figure 2 shows results for a fluid structure interaction where the cylinder moves itself due to a force balance between the drag force and the weight. For reaching a numerical convergence, it was necessary a called “waiting time” before starting the cylinder movement. This waiting time depends of the frequency of the inlet flow. One can notice that when the cylinder reaches the region near the solid boundaries, some stream lines goes inside the cylinder area; due to the fact the Physical Virtual Model has some problem to represent an immersed boundary near to a domain wall.

In cases like this, one can also notice the disagreement between the real position of the Immersed Boundary and the flow behavior in that region. It can be possible that, once the method needs some time steps for that the interface solid-fluid be stabilized, in that region of lower velocity, there was not a waiting time enough that the IB be correctly identified by the fluid flow.

For future works, some improvements will be considered on the acceleration force evaluation, to avoid the problems found by now.

Other authors already proposed some modification in the acceleration force evaluation proposed by the original Physical Virtual Model. As a part of the future work, these new modifications will be implemented and verified in the same Navier-Stokes solver. These results will be compared with the purpose to improve the Method development and understanding about the Immersed Boundary behavior.



$t^* = 8{,}7$ $t^* = 9{,}1$ $t^* = 9{,}5$ $t^* = 9{,}9$

Figure 1: Oscillating flow around a cylinder after the solid boundaries inside a channel. Stream lines and velocity map. $Re = 100$ $KC = 0{,}5$, $\beta = 200$ e $f = 0{,}1$ Hz.

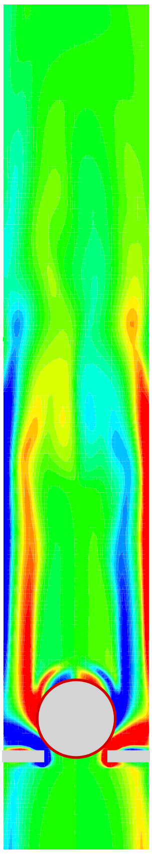


Figure 2: Vorticity map of the flow around a moving cylinder. Force balance between drag force and the weight. *Re* = 100 *KC* = 0,5. Inlet flow is a quadratic wave.

3. ACKNOWLEDGEMENTS

The authors of this paper would gladly like to express their thankfulness to:

- FAPESP for financially supporting this research;
- Government of Canada for supporting six months of research at Dalhousie University;
- Ace-Net systems that allowed this test cases being finished on the pre-determined time.

4. REFERENCES

Beyer R. P., Leveque R. J., "Analysis of a one dimensional model for the immersed boundary method", *SIAM J, Numer. Anal.* 29:332– 64, 1992.

Clarke D., Salas M., Hassan H., "Euler calculations for multi-element airfoils using Cartesian grids". *AIAA J.* 24:1128–35, 1986.

Góis, E. R. C. "Simulação numéricado escoamento em torno de um cilindro utilizando o método das fronteiras imersas", Masters of Science Dissertation, ICMC – USP São Carlos, november, 2007

Goldstein, D., Hadler, R., Sirovich, L.,"Modeling a no-slip flow boundary with an external force field", Journal of Computational Physics, 105, 354, 1993.

Lai, M. C., Peskin, C. S., "An Immersed Boundary Method with Formal Second Oredr Accuracy and Reduced Numerical Viscosity", Journal of. Computational Physics, 160, 705, 2000.

Lima e Silva, A.L.F., 2002, "Desenvolvimento e Implementação de uma Nova Metodologia para Modelagem de Escoamento sobre Geometrias Complexas: Método da Fronteira Imersa com Modelo Físico Virtual", Ph.D. Thesis, University of Uberlândia – UFU, Uberlândia, Brazil, 138 p.

Mittal, R. , Iaccarino, G., "Immesed boundary methods", *Annu. Rev. Fluid Mech.*, 239 – 261, 2005

Peskin, C.S., 1977, "Numerical analysis of blood flow in the heart", J. Comput. Phys., Vol.25, pp.220-252.

Udaykumar H. S., Mittal R., Rampunggoon P., "Interface tracking finite volume method for complex solid-fluid interactions on fixed meshes", *Commun. Numer. Methods Eng.* 18:89–97, 2002.

Udaykumar H. S., Mittal R., Rampunggoon P., Khanna A. "A sharp interface Cartesian grid method for simulating flows with complex moving boundaries". *J. Comput. Phys.* 174:345–80, 2001.

Udaykumar H. S., Shyy W., Rao M. M., "Elafint: A mixed Eulerian-Lagrangian method for fluid flows with complex and moving boundaries", *Int. J. Numer. Methods Fluids* 22:691–705, 1996.

Ye T., Mittal R, Udaykumar H. S., Shyy W. " An accurate Cartesian grid method for viscous incompressible flows with complex immersed boundaries", *J. Comput. Phys.* 156:209–40, 1999.