A new modeling of fluid-structure interaction problems through Immersed Boundary Method/ Virtual Physical Model (IBM/VPM)

Santos Alberto Enriquez-Remigio, remigio@mecanica.ufu.br
Aristeu da Silveira Neto, aristeus@mecanica.ufu.br
Laboratory of Heat and Mass Transfer and Fluid Dynamics
Mechanical Engineer College-Federal University of Uberlândia, João Naves de Àvila Avenue.
Campus Santa Mônica, Minas Gerais, Brazil

Abstract.*The Immersed Boundary Method is a powerful method for simulating flows in complex geometries (structures). In this method, complex geometries boundaries are always embedded in rectangular domain analysis, and the presence of the structure is modelled by some way around immersed boundaries (interfaces): (1) modification of the stencil of calculation around the interface; (2) modeling of one forcing term at the interface and spreading this over fluid, and another mechanism. This work shows a new mathematical modeling for problems of interaction between an incompressible fluid and structures, and it is based on the concept of forcing term on the interface. In this case, a modeling of forcing term is based on philosophy of Virtual Physical Model (determining the force using its own lay that govern the movement of fluid and desirable dynamic to the interface). The bidimensional equations of methology are solved in the context of finite difference method, regular staggered grid and explicit Projection Method with second-order accuracy in space and time. Two problems are simulated, where one or two regions in the fluid separated by the interface are interesting to study: flow over a stationary circular cylinder and flow inside of three rotating concentric cylinders. Results show a good agreement with the reported ones in literature.*

Keywords: *Finite difference method, Navier-Stokes equations, Immersed Boundary Method, Fluid-structure interaction.*

1. Introduction

Mathematical Modeling represents special relashionships between different unknows related to some problems. Depending on the problem, different mathematical modeling could be found. For example, for fluid-structure interaction can be mentioned, between anoter: (1) modeling that imposes (internal) boundary presence by contour condition (Hu, 1996); (2) modeling that considers a bigger, simpler rectangular domain to the analyse domain and imposes interface presence by a procedure, based on physical problem. This last modeling is known as the Immersed Boundary Method (Mittal and Iaccarino, 2005).

The Immersed Boundary Method presents the advantage of being computational cheap related to methodologies that use boundary conditions (Fadlun et al. (2001)), because the use of a Cartesian grid allows the use of existing, eficiente methods to solver discret equations associated to the fluid. Moreover, in problems involving structures in movement, it is not necesary to reconstruct the Eulerian grid of fluid. The disadvantage is there is not exact expression to the forcing term to be used in the fluid equations properly. This feature leads to appear a lot of methodologies which presented good behaviour in the experiences presented for the authors.

Peskin was the first in presenting a mathematical model based on IBM and then the Peskin model is known by Immersed Bounday Method (Peskin, 1972). Following, it is used the Peskin IBM expression to refer to Peskin specific model. The aplicability of Peskin methodology was demonstrated in numerical simulations of different problems (Peskin, 2002) and even though he has deduced his mathematical model based for boundaries with elastic properties, it was used to simulate interaction between fluid and rigid-structure. For example, the study of flow around a rigid circular cylinder immersed in the fluid (Lai and Peskin, 2001). In this work, to impose physical characteristic of the cylinder rigidity it had used a force model based on the Hooke's law defined on the interface. This force modeling needs the value of a constant to be determined in such a way to impose non movement on the interface.

Another expression for the force field on the interface was presented by Lima e Silva et al. (2003). The force modeling was based on both the law that governs the fluid and the desirable dynamics for the interface. This force modeling is not dependent the constants to be adjusted and it is known as Virtual Physical Model. Different numerical simulations for problems about fluid-structure (rigid) interaction, presented in literature, had shown the applicability of the methodology IBM/VPM. These are some examples: flow around a circular cylinder; flow around a sphere; flow around an airfoil; movement of a pendulum immersed in a fluid, between another.

To problems of fluid-structure (rigid) interaction, methodologies of immersed boundary mentioned above and other ones that was not described on this work (Fadlun et al. (2001)), were used to study problems where the study focus is one the regions defined by the interface: internal or external flow. The actual proposal is to consider problems where one or both interface defined regions as interesting parts to study.

A fundamental step to prove if there is or not mathematical coherence in a mathematical modeling is by numerical verifying of solution convergence. In the case of Peskin methology, they are works like Lai and Peskin (1998), Enriquez-

Remigio and Roma (2005), etc. that had shown such methodology presents first-order spatial convergence. A similar result for a IBM/VPM methodology was presented by Arruda (2004) and Enriquez-Remigio (2005). These results was found to fluid-structure interaction problems in which the structure presents a prescribed motion.

In this work, it is presented a mathematical model based on IBM/VPM to solving interaction problem between a incompressible fluid and a immersed rigid-structure. The presentation of this work is divided in that way: In Section 2, it is presented the mathematical modeling proposal; the mumerical method as well as some used numerical relationships are presented on Section 3; numerical results of two simulated problems are presented on the Section 4, showing that: (1) there is convergence to a problem with known analytical solution; (2) as well as coherences on the computational parameters found to a known problem; conclusions and future works are indicated on Section 5.

2. Mathematical Model

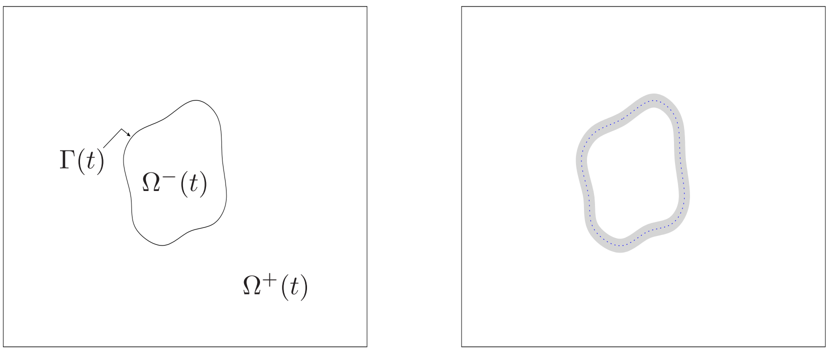
The actual mathematical modeling is based on Immersed Boundary Method that uses a forcing term (in the fluid equations) to model the presence of the immersed structure. Equations to incompressible fluid in rectangular domain are given by the Navier-Stokes equations:

$$\rho\Big(\frac{\partial \mathbf{u}}{\partial t}(\mathbf{x},t) + \mathbf{u}\cdot\nabla\mathbf{u}(\mathbf{x},t)\Big) = -\nabla p(\mathbf{x},t) + 2\mu\mathbf{d}(\mathbf{x},t) + \mathbf{f}(\mathbf{x},t), \quad \mathbf{x}\in\Omega, \tag{1}$$

$$\nabla\cdot\mathbf{u}(\mathbf{x},t) = 0, \quad \mathbf{x}\in\Omega, \tag{2}$$

where, $\mathbf{u}$ is the velocity and p, the pressure of the fluid. Physical parameters ρ and μ, constant on space and time, are respectively the specific mass and the dynamic viscosity, $\mathbf{d} = \frac{(\nabla\mathbf{u}+\nabla^t\mathbf{u})}{2}$ is the defomation rate measurer and $\mathbf{f}$ is the external force field that represents the presence of a immersed structure.

On the Figure 1-(a), it is drawn a tipycal domain of fluid-structures interaction (Ω) with the interface ($\Gamma(t)$) that represents the boundary of the immersed structure. In this case, the interface divide the fluid domain in two disjoint regions $\Omega^-(t)$ and $\Omega^+(t)$ (internal and external region of the interface, respectively). In the other side, in the Figure 1-(b), it is presented the force field around the interface that imposes the presence of the structure.



(a) (b)

Figure 1. Immersed Boundary Method: (a) geometrical configuration of fluid-stuture interaction; (b) force field on the equations of fluid imposing structure presence.

To model the forcing term, it is used the ideas of Virtual Physical Model (Lima e Silva et al. (2003)): (1) determination of the forcing term on the interface based on both the fluid equations and the condition of structure motion; (2) the use of data of interesting region (one of the regions in which the interface divide the fluid domain) to determine the value of the forcing term expression. In the actual modeling, two forces are calculated ($\mathbf{f}^-$ and $\mathbf{f}^+$), one in such defined region by the interface. The expression of the total forcing term is

$$\mathbf{f}(\mathbf{x},t) = \begin{cases} \mathbf{f}^+(\mathbf{x},t) + \mathbf{f}^-(\mathbf{x},t), & \mathbf{x}\in\Gamma(t) = \{\mathbf{X}(s,t)/s\in[a,b]\ \ e\ \ t\geq 0\}, \\ 0, & \text{otherwise}, \end{cases} \tag{3}$$

where $\mathbf{X}(s,t)$ is the parametric form of the closed curve $\Gamma(t)$ and expressions to terms $\mathbf{f}^+$ and $\mathbf{f}^-$ are:

$$\mathbf{f}^+(\mathbf{X}(s,t),t) = \Big(\rho\Big(\frac{\partial \mathbf{u}}{\partial t}^+ + (\mathbf{u}\cdot\nabla\mathbf{u})^+\Big) + \nabla p^+ - \nabla\cdot(2\mu\mathbf{d})^+\Big)(\mathbf{X}(s,t),t). \tag{4}$$

$$\mathbf{f}^-(\mathbf{X}(s,t),t) \quad = \quad \Big(\rho\Big(\frac{\partial \mathbf{u}}{\partial t}^- + (\mathbf{u}\cdot\nabla\mathbf{u})^-\Big) + \nabla p^- - \nabla\cdot(2\mu\mathbf{d})^-\Big)(\mathbf{X}(s,t),t). \tag{5}$$

The superscript (+) or (-) in Eq. (4) and Eq. (5) indicate that these forces are calculated by using informations of regions Ω^+ and Ω^-, respectively. The Figure 2, illustrates the immersed interface with the two forcing terms, as well as the resulting forcing term **f**.

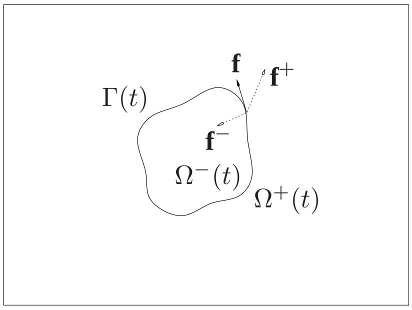


Figure 2. Resulting forcing field illustration, Eq. (3), in the new modeling

Temporal derivates in Eq. (4) and Eq. (5), are approximated by

$$\frac{\partial \mathbf{u}}{\partial t}^+(\mathbf{X}(s,t_n),t_n) \quad \approx \quad \frac{\mathbf{u}^+(\mathbf{X}(s,t_{n+1}),t_{n+1}) - \mathbf{u}^+(\mathbf{X}(s,t_n),t_n)}{\Delta t}, \tag{6}$$

$$\frac{\partial \mathbf{u}}{\partial t}^-(\mathbf{X}(s,t_n),t_n) \quad \approx \quad \frac{\mathbf{u}^-(\mathbf{X}(s,t_{n+1}),t_{n+1}) - \mathbf{u}^-(\mathbf{X}(s,t_n),t_n)}{\Delta t}. \tag{7}$$

For immersed bodies where the immersed struture velocity, $\mathbf{V}(s,t_{n+1})$, is prescribed or determined by another relation (no-slip condition):

$$\mathbf{u}^+(\mathbf{X}(s,t_{n+1}),t_{n+1}) = \mathbf{u}^-(\mathbf{X}(s,t_{n+1}),t_{n+1}) = \mathbf{V}(s,t_{n+1}). \tag{8}$$

By using the Eq. (8) in the equations (6) and (7), we have:

$$\frac{\partial \mathbf{u}}{\partial t}^+(\mathbf{X}(s,t_n),t_n) \quad = \quad \frac{\mathbf{V}(s,t_{n+1}) - \mathbf{u}^+(\mathbf{X}(s,t_n),t_n)}{\Delta t}, \tag{9}$$

$$\frac{\partial \mathbf{u}}{\partial t}^-(\mathbf{X}(s,t_n),t_n) \quad = \quad \frac{\mathbf{V}(s,t_{n+1}) - \mathbf{u}^-(\mathbf{X}(s,t_n),t_n)}{\Delta t}. \tag{10}$$

The expression to the force term **f** given in Eq. (3), as well as calculation of its composing terms (Section 3.1), they are this proposal features.

To maintain the convection about using capital letters to variables defined on the interface, it is denoted the force in Eq. (3) by $\mathbf{F} = \mathbf{F}(\mathbf{X}(s,t),t)$, because it has only different values from zero on the interface. Such force is also named a Lagrangian force

3. Numerical Method

In this work it is used a finite difference method of second order in a grid of regular MAC type. The advancing time was based on a explicit second order Runge-Kutta method and the Navier-Stokes equation solutions was made by a projection method.

The curve $\mathbf{X}(s,t)$ was discrete in Npl points equally spaced (Δs_k), denominated by Lagrangian grid (spacing between the Lagrangian grid points equals to the middle of regular Eulerian grid spacing points, $\Delta x = \Delta y$).

In practise the Lagrangian grid points do not coincide to the Eulerian grid points (fixed), and this do not allow us to determine the terms that composes the force in Eq. (4) and Eq. (5) and the total forcing term formulate Eq. (3), directly. The first difficulty could be solver by an interpolation process, and the second one, by a extapolation process (or distribution), Following, it is presented the interpolation and extrapolation process that were used in this work.

3.1 Calculation process of the force given by VPM

Field forcing terms given in Eq. (4) and Eq. (5) have first and second order derivate on the space. To calculate such terms, it is adopted a similar precedure to that one adopted by Lima and Silva et al. (2003), i.e. to consider aditional points on the interesting region, detemining values on these points for velocity and pressure (by an interpolation process) and then determine the term values that are finding. In this work, it is considered six aditional points in each region determined by the interface. Three of these six points are on the x-direction and three other ones are on y-direction. The Figure 3, presents an example of six aditional points in each of regions defined by the interface ($\mathbf{X}_i^+$, i = 1,2,.. , 6 e $\mathbf{X}_i^-$, i = 1,2,.. , 6.).

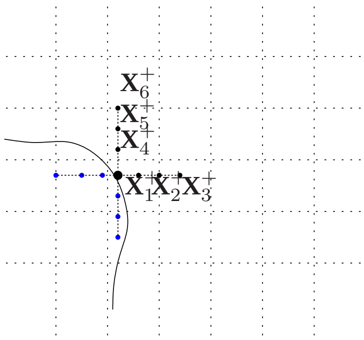


Figure 3. Additional points $\mathbf{X}_1^+, \ldots, \mathbf{X}_6^+$ e $\mathbf{X}_1^-, \ldots, \mathbf{X}_6^-$ used on determination process of force term components (4) and (5), respectivelly.

In these auxiliary points and in the Lagrangian point we determine velocity and pressure denoted by: (u_i^*, v_i^*, p_i^*) $i = 1, 2, \ldots, 6$ and $(u_{fk}^*, v_{fk}^*, pre_{fk}^*)$, where * could be + or - , depending on the region calculation. It is adopted the following interpolation process:

$$\phi^*(\mathbf{X}_i) = \frac{\int_\Omega \phi(\mathbf{x})\delta(\mathbf{x}-\mathbf{X}_i)I^*(\mathbf{x})dx\,dy}{\int_\Omega \delta(\mathbf{x}-\mathbf{X}_i)I^*(\mathbf{x})dx\,dy}, \tag{11}$$

where the function ϕ represents the variable to be interpolated (velocity or pressure component). Function $I^*(\mathbf{x})$ is a function that indicates if a point is from interesting region * ($\Omega^+(t)$ or $\Omega^-(t)$). It is 1 if the **x** point is from the region * and 0, in another case.

3.2 Extrapolation process to calculate the Euleriana grid force

Lima and Silva et al. (2003) used an extrapolation process of the force calculated from Lagrangian points to Eulerian points. Such process is given by the following discrete fomula:

$$\mathbf{f}(\mathbf{x}_{ij}) = \sum_{k=1}^{Npl} \mathbf{F}(\mathbf{X}_k)\frac{D(\mathbf{x}_{ij}-\mathbf{X}_k)}{\Delta x \Delta y}(\Delta s_k)^2, \tag{12}$$

where function D is a function of distribution with the property of $\sum_{i,j} D(\mathbf{x}_{i,j}) = 1$ (Peskin, 2002).

A reason to the statement (12) could be found in Enriquez-Remigio (2005) . The extrapolation formula (12), allows to satisfy the following discrete identity

$$\sum_{i,j} \mathbf{f}_{ij}\Delta x \Delta y = \sum_{k=1}^{Npl} \mathbf{F}_k \Delta s_k \Delta s_k, \tag{13}$$

Such statement, Eq. (13), indicates that interface force is completely spread to the fluid domain.

3.3 Calculation of the drag and lift coefficient

The $\mathbf{f} = (\mathbf{f}_x, \mathbf{f}_y)$ force defined in Eq. (3) is the force with the interface acts over the fluid to impose the boundary presence. Because the action-reaction principle, the body experiments a force with the same intensity but in opposite direction; based on it and on the statement (13) and supposing that the fluid movement is in the x-direction, the drag and lift forces are given by F_D and F_L, respectively, where

$$F_D \;=\; \sum_{i,j}(-\mathbf{f}_{x_{ij}})\Delta x \Delta y = \sum_{k=1}^{Npl}(-(\mathbf{F}_k)_x)\Delta s_k \Delta s_k, \tag{14}$$

$$F_L \quad = \quad \sum_{i,j}(-\mathbf{f}_{y_{ij}})\Delta x \Delta y = \sum_{k=1}^{Npl}(-(\mathbf{F}_k)_y)\Delta s_k \Delta s_k, \tag{15}$$

Therefore, the drag and lift coeficient are given by:

$$C_D = \frac{F_D}{(1/2)\rho U_\infty^2 d}, \qquad C_L = \frac{F_L}{(1/2)\rho U_\infty^2 d}, \tag{16}$$

where, U_∞ and d represent an specific speed and distance, respectively.

4. Results

In this section they are presented numerical results obtained with the new proposal. Two problems are simulated: (1) internal flow in three rotating concentric cylinders and (2) flow around a stationary cylinder. With the first one we can verify: (a) the applicability of the methology to numerical simulation fluid-structure interaction where two regions defined by the interface are interesting to study and (b) the approach order of this method. In the second one, we can verify the applicability of the methodology to solver problems where only one region are interesting to study.

4.1 Internal flow of three rotating concentric cylinders

Flow in three rotating concentric cylinders is an extension of interior flow of two rotating concentric cylinders. This last one is known as circular Couette flow, that has analytical solution. Suppose that w_1 and w_2 are rotation velocities of concentric cylinders with r_1 and r_2 ratio ($r_1 < r_2$), respectively. So, the exact velocity to the flow between the two cylinders is given in cylindrical Coordinates by:

$$v(r) = Ar + \frac{B}{r}, \tag{17}$$

where A and B are constants to be determined by the wall conditions, values of these constants are, respectively:

$$A \quad = \quad \frac{w_2 r_2^2 - w_1 r_1^2}{r_2^2 - r_1^2},$$
$$B \quad = \quad \frac{(w_1 - w_2) r_1^2 r_2^2}{r_2^2 - r_1^2}.$$

To represent the cylinder presences on Eulerian bidimensional domain it is used three Lagrangian grid. The first grid represents the first internal cylinder, with r_1 ratio and w_1 rotational velocity; the second grid represents the second cylinder, with r_2 ratio and w_2 rotational velocity and the third grid represents the third cylinder with r_3 ratio and w_3 rotational velocity (Figure 4).

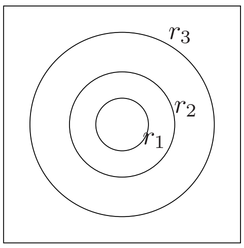


Figure 4. Three concentric internal (r_1), middle (r_2), external (r_3) cylinders.

In this work it is simulated the case in which the second cylinder is rotating in counter-clockwise direction and the other cylinders are stopped.

Simulation parameters are:

- $\Omega = [0, 0.009] \times [0, 0.009]$
- $\rho = 1000 Kg/m^3$, $\mu = 0.001 kg/m\, s$
- $r_1 = 0.001m$, $r_2 = 0.002m$, $r_3 = 0.0035m$, d_{12}= $0.001m$, d_{23}= $0.0015m$, $r_2/r_1 = 2$ e $r_3/r_2 = 1.75$
- $w_1 = 0.0 rad/s$, $w_2 = 2.0 rad/s$ e $w_3 = 0.0 rad/s$

- Initial condition: $\mathbf{u}(\mathbf{x}, 0) = (0, 0)$
- Boundary condition in external Eulerian contour for the velocity is homogeneous Dirichlet

It was realized simulations to three different sizes of Eulerian and Lagrangian grids. In Table 1, it is presented descriptions related to the Eulerian and Lagrangian grids, where: Npl_1, Npl_2 and Npl_3 are the quantity of Lagrangian points to the first, second and third cylinders, respectivelly; and NCd_{ij} is the quantity of Eulerian cells of separation between the cylinder i and j.

Table 1. Circular Couette flow:Eulerian and Lagrangian grid sizes.

$N_x = N_y$	Npl_1	Npl_2	Npl_3	Δx	NCd_{12}	NCd_{23}
90	126	252	440	1.0e-02	10	15
180	252	504	880	5.0e-03	20	30
360	504	1008	1760	2.5e-03	40	60

The criterion to determine if the steady state was reached is based on Oliveira (2000):

$$\|\frac{\partial(u, v)}{\partial t}\|_{max} < \frac{\epsilon}{N_x N_y},$$

where the value of ϵ is a constant, and in that case it is adopted the value of 0.001.

It is calculated the maximum error norm on the ring regions of the concentric cylinders, between the numerical solution obtained and the exact solution respectively, Eq. (17), as well as the ratio of errors to grids NxN and 2Nx2N. Results of Table 2 show a convergence of order one. These results was expected, because of Immersed Boundary Method is a first order method, as discussed in Arruda (2004), Enriquez-Remigio (2005), Lai and Peskin (2001), and so on.

Table 2. Error norms on interesting regions: Region 12, Region 23. Case where the middle cylinder rotates and the other ones are stopped.

n	$\|erro(12)\|_\infty$	Ratio	$\|erro(23)\|_\infty$	Ratio
90	1.73041e-02		1.69802e-02	
180	1.25020e-02	1.384107	1.22614e-02	1.384850
360	5.26030e-03	2.376671	5.43021e-03	2.257997

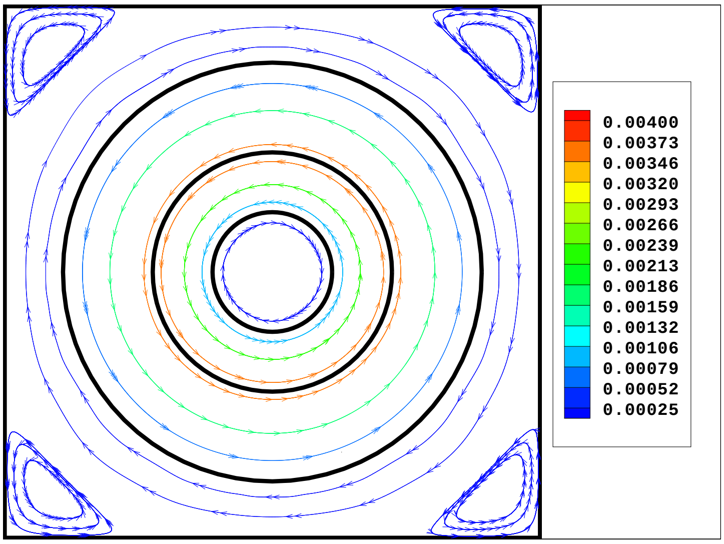


Figure 5. Ring region flow of three rotating concentric cylinders. The case in which the middle cylinder rotates counter-clockwise. Streamlines in function of the velocity norm (black lines represent the circular concentric cylinders).

The streamlines on the Figura 5 show three important behaviour obtained by the methodology: (1) streamlines around the middle cylinder follows its rotation counter-clockwise direction; (2) streamlines on the internal region around the internal cylinder and external region around the external cylinder are counter-rotative, reflecting the recuperation of non-movement boundary condition that was virtually modelled over this cylinder; and (3) streamlines around the Eulerian boundary domain, $\gamma = \partial\Omega$, present recirculations on neighborhood corners of this domain. Note that the external flow to the ring channel is inducted purely by the force field calculated by VPM. It is interesting to perceive that secondary cells was capturated on four domain corners.

4.2 Flow over a circular cylinder

Flow around a circular cylinder is a classic problem that was used to test numerical methodologies. To this problem, depending on Reynolds number, drag and lift coeficients are known, between another. Moreover, this problem tests methodology flexibility to lead with problems where one of regions defined by the interface is interesting to study.

Following, it is presented results obtained from simulation of flow around circular cylinder by using the new proposal for several Reynolds numbers. Simulations were developed following the same configurations that was used by Lima and Silva et al. (2003), basically.

The flow moves itself from left to right side because of the imposition of constant velocity, U_∞, as a left boundary condition. In the other boundaries it was imposed homogeneous Neumann condition type. The domain $\Omega = [0, L_x] \times [0, L_y]$, where $L_x = 35D$ and $L_y = 15D$, and D is the diameter of the circular cylinder. The circular cylinder position have as coordinates $x = 16.5D$ and $y = 7.5D$ (Figure 6). The Reynolds number is defined as $R_e = \frac{\rho U_\infty D}{\mu}$. The initial condition to the velocity is $(U_\infty, 0)$. In this case, it was considered a circular cylinder with a diameter $D = 0.1$. The fluid density $\rho = 1$, $U_\infty = 1.0$ and the viscosity were imposed in such a way that it could be possible obtain the properly Reynolds number.. The Eulerian grid consists of 700×300 cells ($\Delta x = 0.005$) and the number of Lagrangian points of the $Npl = 256$ points.

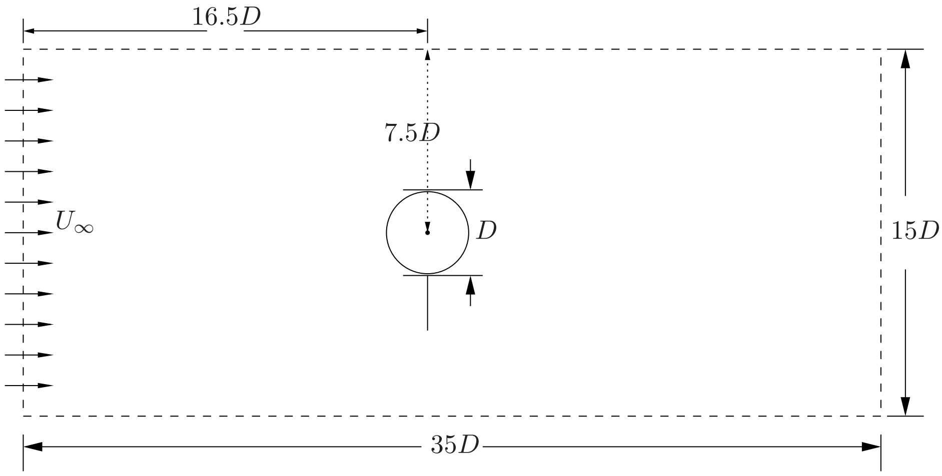


Figure 6. Geometry for flow passing a stationaty cylinder.

In Tables 3 e 4, it is presented a summarize of drag and lift coefficients to different Reynolds numbers, respectivelly, obtained by the actual methogology and registered on literature. You can see that the results present good concordance with those ones just obtained by another authors.

Table 3. Drag coefficients to different Reynolds numbers.

Re	Present work	Lima e Silva et al. (2003)	Triton	Liu et al. (1998)	De Tullio et al. (2007)	Le et al (2006)
40	1.537	1.54	1.48		1.55	1.56
80	1.380 ± 0.004	1.40	1.29			
100	1.357 ± 0.010	1.39		1.350 ± 0.012	1.32 ± 0.010	1.37 ± 0.009
150	1.329 ± 0.018	1.37		1.339 ± 0.030		
200	1.317 ± 0.037			1.310 ± 0.049	1.34 ± 0.045	1.34 ± 0.030
300	1.302 ± 0.063	1.27				

Table 4. Lift coefficients to different Reynolds numbers.

Re	Present work	Liu et al. (1998)	De Tullio et al. (2007)	Le et al. (2006)
40	0.0			
80	± 0.258			
100	± 0.321	± 0.390	± 0.331	± 0.323
150	± 0.486	± 0.530		
200	± 0.617	± 0.690	± 0.680	± 0.430
300	± 0.748			

5. Conclusion

In this work it was presented a new approach to use IBM/VPM, that presents a general feature, in the sense of it serves to model problems where one or both regions divided by the interface are interesting to study. Results presented for two problems show that:

- First-order spatial convergence;
- Applicability to solver problems where one of the interface regions is interesting to study.

Some future steps could be focused on studying the use of this methodology to solver more complex problems.

6. Acknowledgments

To CNPQ by the financial help. To the Laboratory Heat and Mass Transference and Fluid Dynamics people, specially to Lima and Silva and Tiago da Silva by discussions that provide development conditions to this work, as well as to Professor Alexandre M. Roma by putting important questions referring to the mathematical modeling.

7. References

Arruda, J. M., 2004, “ Modelagem Matemática de Escoamentos Internos Forçados Utilizando o Método da Fronteira Imersa e o Modelo Físico Virtual”, PhD thesis, Universidade Federal de Uberlândia.

De Tullio, M.D., De Palma, P., Iaccarino, G., Pascazio, G., and Napolitano, M., 2007, “An immersed boundary method for compressible flows”, Computers and Fluids, Vol. 35, pp. 693-702.

Enriquez-Remigio, S. A., 2005, “Interação Fluido Estrutura atráves do Método Físico Virtual”, Ph. D. Thesis, Instituto de Matemática e Estatística, Universidade de São Paulo, São Paulo, Brazil.

Enriquez-Remigio, S.A., Roma, A.M., 2005, “Incompressible flows in elastic domains: an immersed boundary method approach”, Applied Mathematical Modelling, Vol. 29, pp. 35-54.

Fadlun, E.A., Verzicco, R., Orlandi, P., Mohd-Yusof, J., 2000, "Combined immersed-boundary finite-difference methods for three-dimensional complex flows simulations", Journal of Computational Physics, Vol. 161, pp. 35-60..

Oliveira, A. F., 2000, “Técnicas Computacionais para Dinâmica dos Fluidos: Conceitos Básicos e Aplicações”, EDUSP.

Hu, H.H., 1996, “Direct Simulation of Flows of Solid-Liquid Mixture”, Int. J. Multiphase Flow, Vol. 22, pp. 335-352

Lai, M.-C., Peskin, C.S., 2001, “The Immersed Interface Method for the Navier-Stokes Equations with Singular Forces”, Journal of Computational Physics, Vol. 171, pp. 822-842

Le, D.V., Khoo, B.C., Peraire, J., 2006, “An immersed interface method for viscous incompressible flows involving rigid and flexible boundaries”, Journal of Computational Physics, Vol. 220, pp. 109-138.

Lima and Silva, A.L.F., Silveira-Neto, A., Damasceno, J.J.R., 2003, “Numerical simulation of two-dimensional flows over a circular cylinder using the immersed boundary method”, Journal of Computational Physics, Vol. 189, pp. 351-370.

Liu, C., Zheng, X., Sung, H., 1998, “Preconditioned Multigrid Methods for Unsteady Incompressible Flows”, Journal of Computational Physics, Vol. 139, pp. 35-57

Mittal, R., Iaccarino, G., 2005, “Immersed Boundary Methods”, Annual Review Fluid Mechanic, Vol. 37, pp. 239-261.

Peskin, C.S. 1972, “Flow patterns around heart valves: a numerical method”,Journal of Computational Physics, Vol. 10, pp. 252-271.

Peskin, C.S. 2002, “The Immersed Boundary Method”, Acta Numerical, pp. 1-39.

Xu, S., Wang, Z.J., 2006, “An immersed interface method for simulating the interaction of a fluid with moving boundaries”, Journal of Computational Physics, Vol. 216, pp. 454-493.

8. Responsibility notice

The authors are the only responsible for the printed material included in this paper