

A TWO PRESSURE-VELOCITY APPROACH FOR IMMERSED BOUNDARY METHODS IN THREE DIMENSIONAL INCOMPRESSIBLE FLOWS

T.M.Y.S Tuan Ya¹, O. Sabir^{2*}, Norhafizan Ahmed² and Y. Nukman²

¹Faculty of Mechanical Engineering, Universiti Malaysia Pahang,
26600 Pekan, Pahang, Malaysia

²Faculty of Engineering, University of Malaya,
50603 Kuala Lumpur, Malaysia

*Email: osama_sabir@siswa.um.edu.my

ABSTRACT

This paper describes innovative method for computing fluid solid interaction using Immersed boundary methods with two stage pressure-velocity corrections. The algorithm calculates the interactions between incompressible viscous flows and a solid shape in three-dimensional domain. The fractional step method is used to solve the Navier-Stokes equations in finite difference schemes. Most of IBMs are concern about exchange of the momentum between the Eulerian variables (fluid) and the Lagrangian nodes (solid). To address that concern, a new algorithm to correct the pressure and the velocity using Simplified Marker and Cell method is added. This scheme is applied on staggered grid to simulate the flow past a circular cylinder and study the effect of the new stage on calculations cost. To evaluate the accuracy of the computations the results are compared with the previous software results. The paper confirms the capacity of new algorithm for accurate and robust simulation of Fluid Solid Interaction with respect to pressure field.

Keywords: Immersed Boundary Methods; Two stage pressure-velocity correction approach; three dimension domain; incompressible flow.

INTRODUCTION

In many fluid-solid engineering the structure is in general flexible and interacts with the surrounding fluid. Commonly a body-fitted meshes techniques is used to model such systems since its consideration to both fluid and solid dynamics like The arbitrary Lagrangian–Eulerian method(Donea, Giuliani et al. 1982). The Immersed boundary methods (IBM) is a nonconforming methods to calculate the fluid solid interaction. Charles S, Peskin invented the methods(Charles S 1977) and since then it is become popular among the researcher because of simplicity and geometric flexibility. IBMs cancel the need of mesh regenerations and handle well the boundary deformations.

IBMs represent the immersed boundary by a set of moving Lagrangian points with a fixed Eulerian grid for the fluid. The group of material points linked with fibers (springs) shows the solid interface mode. This technique simplifies treating immersed boundaries of nearly any arbitrary shapes, sizes and configurations, without use of ALE descriptions or adaptive meshing(Wang 2010) . An efficient way to transform between the Eulerian fluid motion and the Lagrangian structure motion is to use the Dirac delta function. For numerical view the big advantage of IBM is the ability to employ a simple finite difference or finite volume scheme on a Cartesian mesh. Apply elastics boundary algorithm to simulate moving boundaries problems is easy, and it is work in three

dimension simulation. One of the major weakness that it cannot work efficiency when simulate a flow in high Reynolds numbers(Peskin 2002).

(Mohd-Yusof 1997) developed a method that extracts the forcing direct from the numerical solution for which a preliminary estimate can be determined(Mohd-Yusof 1998). The discrete approach is better suited for higher Reynolds numbers, because of imposing the velocity boundary conditions at the immersed boundary, without introducing or computing any forcing term. The governing equations are most of the time discretized as follows. A second-order Adams-Bash forth scheme is employed for the convective terms, while the diffusion terms are discretized using an implicit Crank-Nicolson scheme. This cancels the viscous stability constraint, which can be severe in simulation of viscous flows. The (spectral) method of Mohd-Yusof settle the force term by the variance between the interpolated velocities in the boundary points and the anticipated (physical) boundary velocities. This method reduces the errors between the calculated velocities and the desired velocity profile on the body surface. If the boundary is stationary, the force can be determined by pairing the velocity at the internal point to the velocity at the external point with a weighted linear interpolation (Zhang and Zheng 2007).

The obstacle of using a grid does not adapt to the body border is including the boundary conditions in the governing equations will involve enormous adjustments nearby the boundaries. Comparing with the traditional methods, it is challenging and had a bad impact on accuracy and conservation properties when treat the boundaries in IBM. In the boundary conforming grid methods, the control of grid resolutions on the body surrounding areas is much better. IBM has effects on grid resolutions to achieve the certain amount of control which leads to increase of the grid size with increasing Reynolds numbers. The alignment between the body surface and the grid lines places some grid points to be inside the body boundary where solving fluid flow equations is not essential (Ya, Takeuchi et al. 2007).

To enforce Dirichlet conditions at the boundary, interpolations are performed between Lagrangian velocities at the solid surface and Eulerian velocities at neighboring grid points. These interpolations have proved to be un robust, which is undoubtedly as obstacle for this method to become widespread for practicable applications. In (Kajishima, Takiguchi et al. 2001) body force method, the force between the solid and fluid is modeled by a volume fraction function of the solid volumetric fraction and the relative velocities of the two phases. This method ensures no momentum leakage between the phases as both fluid and solid share the rectilinear Eulerian grid and, therefore, making interpolation unnecessary. However, continuity is affected because of changes in velocity field enforced by volume averaging of the local fluid velocity and local solid velocity. Using Kajishima's method, the fluid-solid interaction is coupled by the body force. To use pressure as means of coupling of new two-way fluid-solid interaction, additional correction of pressure and velocity is needed.

In this paper we develop a model for three dimensions arbitrary elastic solid in a viscos incompressible domain based on two stage pressure-velocity correction immersed boundary methods. Following the steps of (Ya, Takeuchi et al. 2007) Method, we develop a code for three dimensions domain.

MATHEMATICAL MODEL

The equations governing the flow around the immersed boundaries are the continuity and momentum equations for viscous incompressible flow past a body is describe by the non-dimensional Navier-Stokes equations, including the body force term are given by

$$\frac{\partial \vec{u}}{\partial t} + \vec{u} \cdot \nabla \vec{u} + \frac{1}{\rho} \nabla p - \frac{1}{Re} \Delta \vec{u} = \vec{f}, \quad (1)$$

$$\nabla \cdot \vec{u} = 0 \quad \text{in } \Omega_f \quad (2)$$

$$\vec{u} = \vec{u}_\Gamma \quad \text{on } \Gamma_b \quad (3)$$

Where $\vec{u}(\vec{x}, t)$ represent the velocity field of the flow, $p(\vec{x}, t)$ is the pressure and \vec{f} represents any external source terms. The quantities ρ and μ are constant fluid density and viscosity, respectively. The solid body occupies the domain Ω_b , with boundary showed by Γ_b , and the surrounding fluid domain noted by Ω_f . The governing equations were transformed into dimensionless forms on incorporating the following nondimensional variables.

This set of equations is solved with the immersed boundary condition, which constrains the velocity of the fluid to the velocity of the local IB, exactly at the IB. The immersed boundary condition (IBC) can be implemented in different ways that lead to different methods, which can be both of first- or second-order accurate. A second-order accurate method employs a piecewise linear representation of Eq. (3); that is, linear interpolation is employed to relate the IB to the surrounding points. In a first order accurate method, the IB is approximated to the nearest discrete velocity point.

For cells partially occupied by the solid structure, Kajishima et al (Kajishima, Takiguchi et al. 2001) proposed an immersed boundary method that solves the momentum exchange at the fluid structure boundary. This method is briefly described by Yuki et al (Yuki, Takeuchi et al. 2007) as follows. A velocity field \vec{u} is introduced by calculating the volume fraction and finding the local fluid velocity \vec{u}_{FLUID} and the local solid phase velocity \vec{u}_{SOLID} in each cell by:

$$\vec{u} = (1 - \alpha) \vec{u}_{FLUID} + \alpha \vec{u}_{SOLID} \quad (4)$$

where $(0 \leq \alpha \leq 1)$ is the local solid volumetric fraction in a cell. The fluid-structure interaction then can be solved at the interface by assuming the velocity field \vec{u} follows the adjusted Navier-Stokes equations:

$$\frac{\vec{u}^{n+1} - \vec{u}^n}{\Delta t} = -\vec{u} \cdot \nabla \vec{u} - \frac{1}{\rho} \nabla p + \frac{1}{Re} \Delta \vec{u} + \vec{f}, \quad (5)$$

$$\frac{\vec{u}^{n+1} - \vec{u}^n}{\Delta t} = -\frac{1}{\rho} \nabla p + \overline{RHS} + \vec{f}, \quad (6)$$

And by sing projection method

$$\vec{u}_i = \vec{u}^n + \Delta t \left(\frac{3}{2} \overline{RHS}^n - \frac{1}{2} \overline{RHS}^{n-1} \right) \quad (7)$$

where superscripts n represent time, Δt is the time increment and \vec{u}_i is intermediate velocity. Time advancement is then carried by second-order Adam-Bashforth method and Fractional Step Method as a single continuum. Then we solve the pressure A Poisson equation is then solved with diverging the intermediate velocity \vec{u}_i as a source term to provide a pressure p^{n+1} ,

$$\nabla^2 p^{n+1} = \frac{\nabla \cdot \bar{\mathbf{u}}_I}{\Delta t} \quad (8)$$

this is then used to correct the intermediate velocity, providing a divergence free velocity $\bar{\mathbf{u}}^{n+1}$ and integration continuing to the next time step.

$$\bar{\mathbf{u}}^{n+1} = \bar{\mathbf{u}}_I - \Delta t \nabla p^{n+1} \quad (9)$$

That finish the first correction stage.

The second correction for the velocity field $\bar{\mathbf{u}}_{2nd}^{n+1}$ is derived by the volume averaging the local fluid velocity $\bar{\mathbf{u}}^{n+1}_{FLUID}$ and the local solid phase velocity $\bar{\mathbf{u}}^{n+1}_{SOLID}$ in each fluid cell that is partially or fully occupied by solid cell,

$$\bar{\mathbf{u}}_{2nd}^{n+1} = (1 - \alpha)\bar{\mathbf{u}}^{n+1}_{FLUID} + \alpha \bar{\mathbf{u}}^{n+1}_{SOLID} \quad (10)$$

Since there is significant change of the velocity field, the value of pressure should also reflect these changes. Therefore developing the second-stage correction. This procedure is also applied to fluid cells that are partially or fully occupied by solid cells bounded by a region called small domain. First, the SMAC method is used to find the scalar potential, ϕ^{n+1} . ϕ^{n+1} then used as the velocity and pressure correction factor to get $\bar{\mathbf{u}}_{3rd}^{n+1}$ and p_{2nd}^{n+1}

$$\nabla^2 \phi^{n+1} = \frac{\nabla \cdot \bar{\mathbf{u}}_{2nd}^{n+1}}{\Delta t} \quad (11)$$

$$\bar{\mathbf{u}}_{3rd}^{n+1} = \bar{\mathbf{u}}_{2nd}^{n+1} - \Delta t \nabla \phi^{n+1} \quad (12)$$

$$p_{2nd}^{n+1} = p^{n+1} + \phi^{n+1} \quad (13)$$

where $\bar{\mathbf{u}}_{3rd}^{n+1}$ and p_{2nd}^{n+1} are velocity and pressure that satisfies continuity equation. The velocity and pressure fields are used for the next time step.

Discretization of the governing equations

The continuity and momentum equations are discretized on a staggered grid (Harlow and Welch 1965, Patankar 1980), which is more natural than a collocated variable arrangement for Cartesian problems, as there is no artificial boundary conditions are needed and false modes are prevented. On a staggered grid, the cell faces contain the normal velocity components. The remaining variables, such as density and pressure, are stored at cell centers. A pressure projection method is employed to obtain the correct pressure after an initial guess of the velocity field.

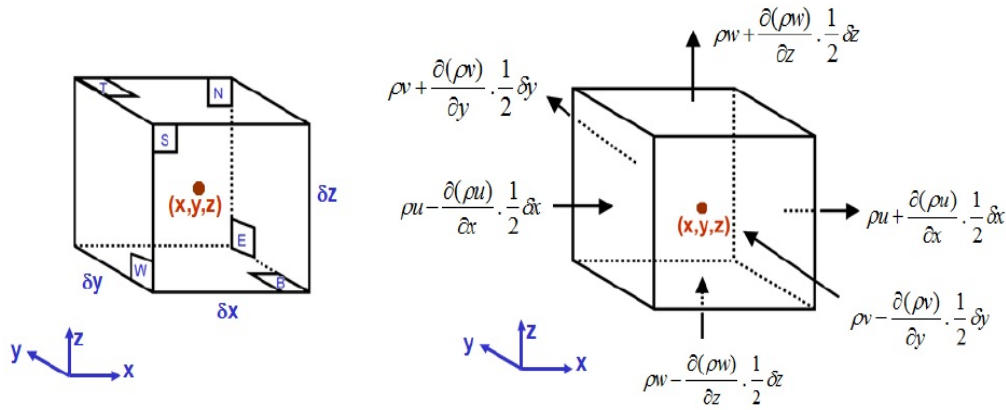


Figure 1. (a) Fluid element for conservation laws; (b) mass flows of fluid element

In Figure 1a, the six faces are labeled N, S, E, W, T and B, which stands for North, South, East, West, Top and Bottom. The pressure in the center of the element is found at position (x, y, z) . Deriving the mass conservation equation is to write down a mass balance for the element lead to :

$$\begin{aligned} \frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} &= -\frac{\partial p}{\partial x} + \frac{1}{Re} \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) + f_x \\ \frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} &= -\frac{\partial p}{\partial y} + \frac{1}{Re} \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) + f_y \\ \frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} &= -\frac{\partial p}{\partial z} + \frac{1}{Re} \left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right) + f_z \\ \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} &= 0 \text{ in } \Omega_f \end{aligned} \quad (14)$$

$$(15)$$

This equation will discretize using finite difference schemes with second order. By using the same local volumetric fraction it maintain secondorder accurate interpolation. This algorithm impose as:

1. Solve the Navier-Stokes equations using projection methods
2. Use second-order Adam-Bashforth method and Fractional Step Method to get the intermediate velocity component
3. Apply Successive of relaxation to calculate the pressure using the Poisson equation.
4. Correct the velocity.
5. Calculate the local solid volumetric fraction and update the velocity.
6. Use SMAC method to find the scalar potential ϕ then solve Poisson equation using SOR.
7. Update the velocity and pressure.
8. Go to step (1).

VALIDATION

Flow past a circular cylinder follow the shape of the body provided the velocity of the flow is slow; this is known as laminar flow. Validate our algorithm by Recreating the same bench mark test which suggested by Schäfer, Michael, et al. (Schäfer, Turek et al. 1996). For the 3D test cases the flows around a cylinder with circular cross-sections are considered.

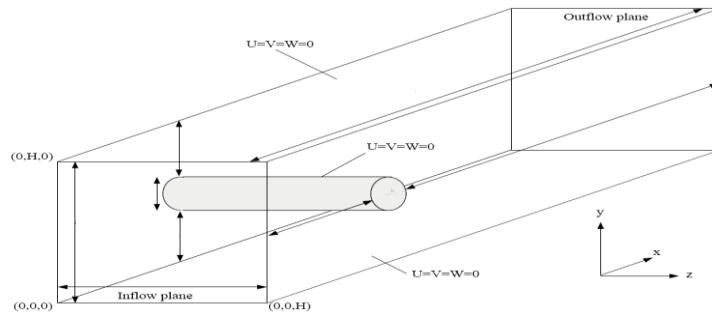


Figure 2 Configuration for flow around a cylinder (Schäfer, Turek et al. 1996).

RESULTS

Animated movies were created to foresee the results data from the model. The representation of the simulation of viscous Newtonian fluid flow around solid cylinder at definite time.

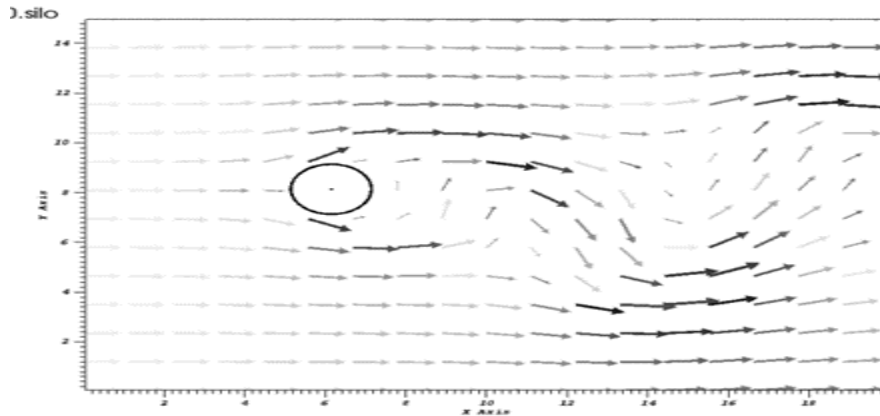


Figure 3 simulation flow around solid cylinder

The time step and grid size in the simulations are chosen with consideration to the stability coefficient. We calculate the drag and lift coefficients and make comparison with the bench mark data. In chart (4, 5) the simulated drag coefficient C_d and simulated lift coefficient C_l are plotted against the result obtained from the bench mark tests. It can be seen that the simulated drag coefficients lie very close to the established relations. The drag coefficient differs slightly from the test data because the domain size starts to affect the flow and the known drag relationships are determined for cylinder immersed in an infinite fluid.

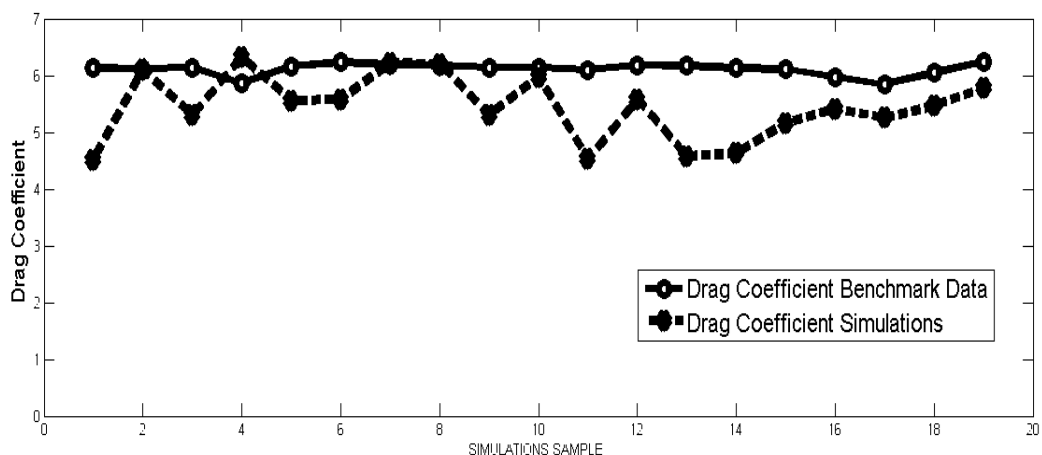


Figure 4 Drag Coefficient

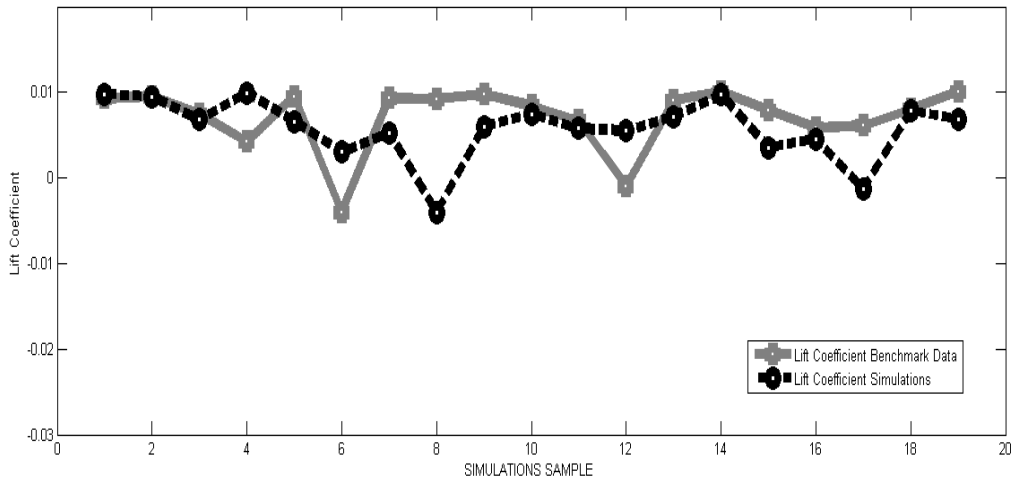


Figure 5 Lift Coefficient

We assume in our hypothesis that the cylinder is very rigid so there is no need to calculate the solid movement by using finite element methods. That leads to the different in the pressure as figure 6 shows. Although, it is not severely violate the standard deviation limits.

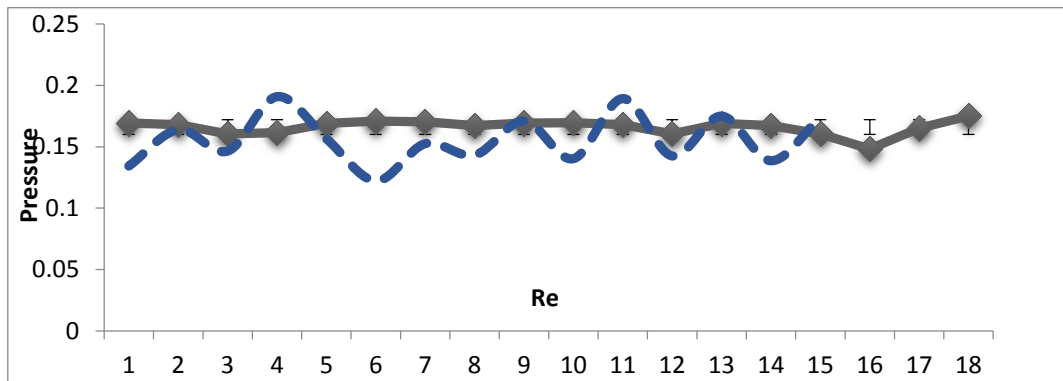


Figure 6 Pressure

CONCLUSION

In this paper, a novel implicit second-order accurate immersed boundary methods with two stage pressure-velocity corrections is developed, implemented and validated. The method is successfully simulating the fluid solid interaction phenomena in three dimensions. Those results encourage us to pursue our quest to upgrade the algorithm to deal with any arbitrary shape. For future work we intend to try new methods for calculating the local volume fraction and optimize the code for better computation cost.

ACKNOWLEDGEMENTS

The authors would like to acknowledge Ministry of Science, Technology and Innovation, Malaysia and University of Malaya for their financial support under: FRGS grant - code: FP064/2010B - Fundamental development and experimental validation of immersed boundary and finite element methods for interaction of a 3D elastic body and fluid by two-stage correction of velocity and pressure..

REFERENCES

- Charles S, P. (1977). "Numerical analysis of blood flow in the heart." *Journal of Computational Physics* **25**(3): 220-252.
- Donea, J., et al. (1982). "An arbitrary lagrangian-eulerian finite element method for transient dynamic fluid-structure interactions." *Computer Methods in Applied Mechanics and Engineering* **33**(1-3): 689-723.
- Harlow, F. H. and J. E. Welch (1965). "Numerical Calculation of Time-Dependent Viscous Incompressible Flow of Fluid with Free Surface." *Physics of Fluids* **8**: 2182.
- Kajishima, T., et al. (2001). "Turbulence Structure of Particle-Laden Flow in a Vertical Plane Channel Due to Vortex Shedding." *JSME International Journal Series B Fluids and Thermal Engineering* **44**(4): 526-535.
- Mohd-Yusof, J. (1997). Combined immersed-boundary/B-spline methods for simulations of flow in complex geometries. Center for Turbulence Research Annual Research Briefs.
- Mohd-Yusof, J. (1998). "Development of immersed boundary methods for complex geometries." Center for Turbulence Research Annual Research Briefs: 325-336.
- Patankar, S. (1980). *Numerical heat transfer and fluid flow*, Taylor & Francis.
- Peskin, C. S. (2002). "The immersed boundary method." *Acta Numerica* **11**: 479-517.
- Schäfer, M., et al. (1996). "Benchmark computations of laminar flow around a cylinder." *Notes on numerical fluid mechanics* **52**: 547-566.
- Wang, X. S. (2010). *Immersed Boundary/Continuum Methods Computational Modeling in Biomechanics*. S. De, F. Guilak and M. Mofrad R. K, Springer Netherlands: 3-48.
- Ya, T. M. Y. S. T., et al. (2007). *Immersed Boundary and Finite Element Methods Approach for Interaction of an Elastic Body and Fluid by Two-Stage Correction of Velocity and Pressure*, ASME.
- Yuki, Y., et al. (2007). "Efficient Immersed Boundary Method for Strong Interaction Problem of Arbitrary Shape Object with the Self-Induced Flow." *Journal of Fluid Science and Technology* **2**(1): 1-11.
- Zhang, N. and Z. C. Zheng (2007). "An improved direct-forcing immersed-boundary method for finite difference applications." *Journal of Computational Physics* **221**(1): 250-268.