Numerical Study of Flow Over a Cylinder Using an Immersed Boundary Finite Volume Method

Ranjith Maniyeri

Department of Mechanical Engineering, Symbiosis Institute of Technology (SIT), Symbiosis International University (SIU), Lavale, Pune-412115, Maharashtra State, India

ranjith.maniyeri@sitpune.edu.in

Abstract—Immersed boundary (IB) method has proved its efficacy in handling complex fluid-structure interaction problems in the field of Computational Fluid Dynamics (CFD). We present a two–dimensional computational model based on an immersed boundary finite volume method to study flow over a stationary circular cylinder. Lagrangian coordinates are used to describe the cylinder and Eulerian coordinates are employed to describe the fluid flow. The Navier-Stokes equations governing the fluid are solved on a staggered Cartesian grid system using the fractional-step based finite volume method. Numerical simulations are performed for four different Reynolds numbers to demonstrate the flow behavior.

Keywords— Immersed boundary method, Flow over cylinder, Finite volume method, Fractional-step method, Mean drag coefficient

I. Introduction

Fluid-structure interaction problems are most common in the field of computational fluid dynamics (CFD). In all the physical problems, the structure under consideration will be flexible or rigid. The flexible or rigid body exerts forces on the surrounding fluid and the fluid will exert forces through shear stresses and pressure differences. Hence, to develop a numerical model which can effectively describe such interaction to reveal the fluid dynamics involved needs careful attention. Conventional methods in CFD utilize body-fitted grid for analysing the flow around an arbitrary body. This methodology requires coordinate transformations and/or complex grid generation. For moving bodies, this technique further needs new mesh generation at each time, resulting in large computational effort. To overcome these difficulties, a new computational method has been developed, which is popularly known as the immersed boundary (IB) method. IB method uses a non-body conforming Cartesian grid. IB method is a Cartesian grid-based method in which the grid does not have to conform to the solid boundary. This method provides a modeling framework and numerical method for studying the interaction of incompressible fluid flow and immersed rigid or elastic structures. In this method, a momentum forcing function is incorporated in the Navier-Stokes equation to make the presence of the immersed boundary and the entire simulation can be carried out in a regular Cartesian grid. Hence the main advantage of IB method is that grid generation is much easier. Added advantages are less memory and CPU time. IB method was first developed by Peskin [1] to simulate blood flow in the human heart. Later, various modifications have been made to this method by different researchers emerging it as an efficient numerical tool for handling complex flow simulations in the field of CFD. As already mentioned, the IB method utilizes momentum forcing function and a Cartesian grid Navier-Stokes

solver. Based on how the momentum forcing is employed in the Navier-Stokes equation, IB method is classified into two: continuous forcing approach and discrete forcing approach. The continuous forcing approach of IB method utilizes Eulerian variables for the fluid region and Lagrangian variables for the solid region. The interaction between these two variables is linked by the Dirac delta function. The discrete version of IB method employs interpolation techniques to obtain the desired no-slip conditions at the boundary and the momentum forcing is obtained directly from the discretized Navier-Stokes equation. The continuous forcing approach is used for the simulation of fluid-flexible body interaction. For the simulation of elastic bodies interacting with the fluid, the Lagrangian force is the elastic force which can be obtained by applying Hooke's law. But when dealing with rigid bodies, this law is not well posed. Hence in such cases methods like the virtual boundary method proposed by Goldstein et al. [2] are employed. This method is based on the feedback forcing scheme which can enforce the no-slip boundary condition on the rigid boundary immersed in the fluid. The basic difference between the virtual boundary method and that of Lai and Peskin [3] is that, the boundary points are exactly prescribed in the former method, but allowed to move slightly from their equilibrium positions in the latter. Recently, Shin et al. [4] proposed a new version of the IB method which combines the feedback forcing scheme of virtual boundary method along with Peskin's regularized delta function. From the literature review on IB method, it is investigated that discrete forcing approach of IB method is suited for simulating rigid body interaction with the fluid whereas continuous forcing approach is suited for both rigid and elastic bodies. Also, for moving body problems, the continuous forcing approach of IB method is far superior because of the simplicity in computing the momentum forcing function. The present work is based on the continuous forcing approach of IB method. The two–dimensional flow past a stationary circular cylinder is a bench mark problem in fluid dynamics and has been widely investigated experimentally and numerically. Also, this problem has been used as the validation problem for most of the IB methods. With this perspective, a numerical model based on the Peskin's IB method [5] is developed to simulate the two – dimensional flow over a stationary cylinder. Here, the cylinder is modeled by discrete number of IB points. The Navier-Stokes equations governing the fluid flow over and surrounding the cylinder are solved on a staggered Cartesian grid system using the fractional-step based finite-volume method. We develop a FORTRAN code to perform numerical simulations to demonstrate the fluid dynamics of flow over a stationary circular cylinder. Numerical simulations are carried out for different values of Reynolds numbers to capture the flow behaviour over and surrounding the cylinder. The variation of mean drag coefficient with Reynolds number is also investigated.

The paper is arranged as follows: the solid model of the cylinder based on IB points and fluid model based on the momentum and continuity equations and the numerical procedure are explained in Section II. The numerical results generated by employing it are discussed in Section III. Finally, the concluding remarks of the present study appear in Section IV.

II. Mathematical Modeling and Numerical Procedure

The general form of mathematical model based on the continuous forcing approach of IB method can be described as follows:

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla p + \frac{1}{\text{Re}} \nabla^2 \mathbf{u} + \mathbf{f}(\mathbf{x},t),$$

(1)

$$\nabla \cdot \mathbf{u} = 0,$$

(2)

$$\mathbf{f}(\mathbf{x},t) = \int \mathbf{F}(s,t) \delta(\mathbf{x} - \mathbf{X}(s,t)) ds,$$

(3)

$$\mathbf{U}(\mathbf{X}(s,t)) = \int \mathbf{u}(\mathbf{x},t) \delta(\mathbf{x} - \mathbf{X}(s,t)) d\mathbf{x},$$

(4)

$$\frac{\partial \mathbf{X}}{\partial t} = \mathbf{U}(\mathbf{X}(s,t)),$$

(5)

where $\mathbf{u}$ is the fluid velocity, p the fluid pressure, Re the Reynolds number, $\mathbf{f}(\mathbf{x},t)$ the momentum forcing function or the Eulerian force density, t the time, $\mathbf{x}$ and s the Eulerian and Lagrangian variables, and $\delta(\mathbf{x})$ the Dirac delta function. Equations (1) and (2) are the Navier-Stokes and continuity equations for an incompressible viscous fluid in the dimensionless form. An elastic structure or organism is represented as an immersed boundary at time t denoted by $\mathbf{X}(s,t)$. Generally, the structure is assumed to have the same density as the fluid and hence the immersed boundary is assumed to be massless. The Lagrangian force $\mathbf{F}(s,t)$ on a particular segment s at time t is determined by the variational derivative of an elastic energy functional $E[\quad]$

$$\mathbf{F}(s,t) = -\frac{\delta E}{\delta \mathbf{X}}.$$

(6)

The Lagrangian forces due to the structure are applied to the Navier-Stokes equation by means of Eq. (3). The presence of the structure is considered in the Navier-Stokes equation as the additional force $\mathbf{f}(\mathbf{x},t)$. Finally, the immersed boundary $\mathbf{X}(s,t)$ is moved according to Eq. (5) using the local fluid velocity computed in Eq. (4). More details can be found in [1, 3, 5].

The schematic view of the physical domain in the dimensionless form is shown in Fig. 1. Note that the non-dimensionalization is achieved by introducing the diameter of the cylinder as the characteristic length $l_c^* = d^*$ and the inlet velocity of the fluid as the characteristic speed U_c^*. The

boundary conditions are imposed in such a way that the flow is from the left to right of the channel. The following boundary conditions are used:

At the inlet: $\quad u = U, v = 0$

(7)

At the outlet: $\quad \frac{\partial u}{\partial x} = 0, \frac{\partial v}{\partial x} = 0$

(8)

At the top wall: $\quad \frac{\partial u}{\partial y} = 0, v = 0$

(9)

At the bottom wall: $\quad \frac{\partial u}{\partial y} = 0, v = 0$

(10)

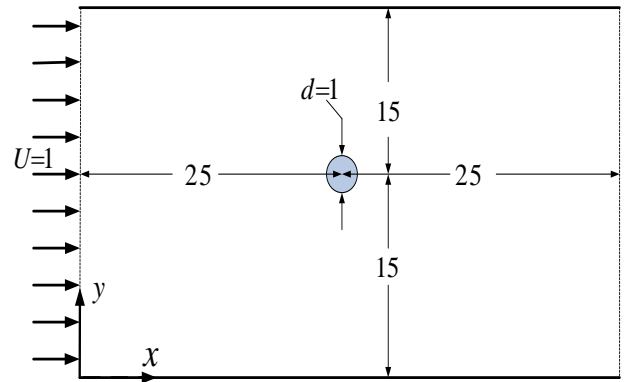At the solid surface of cylinder: $\quad u = 0, v = 0$ (11)

Fig. 1. Schematic diagram of the dimensionless physical domain showing the position of the cylinder.

Here the stationary cylinder, the immersed boundary is considered to be elastic but extremely stiff. The cylinder is divided into discrete number of IB points. The Lagrangian force is computed using a feedback forcing method described in the following way,

$$\mathbf{F}(s,t) = S(\mathbf{X}_E(s,t) - \mathbf{X}(s,t)),$$

(12)

where $\mathbf{X}(s,t)$ and $\mathbf{X}_E(s,t)$ denote respectively the IB and its equilibrium location, and the stiffness constant $S \gg 1$. The above equation interprets that the IB points are connected to their corresponding fixed equilibrium points with a very stiff spring whose stiffness constant is S. So if the IB points fall away from the desired locations, the force on the spring will pull these IB points back. Thus, as the time goes on, it is expected that the IB points will always be close to their desired configurations.

A fractional-step based finite-volume method is employed to solve the equations (1) and (2) to obtain the fluid velocity $\mathbf{u}^{n+1}(\mathbf{x})$ and the fluid pressure $p^{n+1}(\mathbf{x})$ at the next time level $n+1$. In this method a pseudo-pressure is used to correct the

velocity field so that the continuity equation is well satisfied at each computational time step. The governing equations are discretized using a third-order Runge-Kutta method (RK3) for the convection terms and a second-order Crank-Nicolson method for the diffusion terms. The step-by-step procedures for solving the governing equations are depicted below:

$$\frac{\hat{\mathbf{u}} - \mathbf{u}^k}{\Delta t} = \alpha_k \left(L(\hat{\mathbf{u}}) + L(\mathbf{u}^k) \right) - 2\alpha_k \nabla p^k$$
$$- \gamma_k N(\mathbf{u}^k) - \rho_k N(\mathbf{u}^{k-1}) + \mathbf{f}^k ,$$

(13)

$$\nabla^2 \phi = \frac{1}{2\alpha_k \Delta t} (\nabla \cdot \hat{\mathbf{u}}) ,$$

(14)

$$\mathbf{u}^{k+1} = \hat{\mathbf{u}} - 2\alpha_k \Delta t \nabla \phi ,$$

(15)

$$p^{k+1} = p^k + \phi - \frac{\alpha_k \Delta t}{\text{Re}} \nabla^2 \phi ,$$

(16)

where α_k, γ_k and ρ_k are the coefficients of the third-order RK3 method, $\hat{\mathbf{u}}$ the intermediate velocity, Δt the time step-size, ϕ the pseudo-pressure introduced to satisfy the flow-incompressibility constraint, k the substep index and $L(\mathbf{u})$ and $N(\mathbf{u})$ the linear diffusion and non-linear convection terms of the Navier-Stokes equation [6].

III. Results and Discussions

The governing equations outlined in Section II are solved in a two – dimensional rectangular dimensionless domain of 50×30. The cylinder is placed at the centre of the domain. The Eulerian grid size is 281×221 in the streamwise (x) and transverse (y) directions, respectively. Eighty grid points are uniformly distributed inside and near to the cylinder in both the x- and y-directions and the grid is stretched away from the cylinder. The smallest grid size is 0.025. The cylinder is divided into 288 Lagrangian (IB) points.

The numerical results pertinent to the flow over a stationary cylinder are obtained by developing a code in FORTRAN. Numerical simulations are performed for four different Reynolds numbers, Re=10,20,40 and 100. The Reynolds number is based on the cylinder diameter and the inlet velocity. Figure 2 shows the streamlines of the flow when it reaches its final steady state for the three different Reynolds numbers (Re=10,20,40). In all cases, a pair of stationary recirculating vortices is formed behind the cylinder. At a very low Reynolds number, flow around a circular cylinder is steady and symmetrical upstream and downstream.

As the Reynolds number increases, the upstream-downstream symmetry disappears and two attached vortices appear behind the cylinder. These vortices become bigger with increasing Reynolds number. As the Reynolds number becomes

higher than 40, alternating eddies are formed and convected in the wake. This generates the alternating separation of vortices, which are convected and diffused away from the cylinder, known as Karman vortex shedding phenomena. The instantaneous streamline pattern of unsteady laminar flow at Re=100 is presented in Fig.3. The spanwise vorticity contours near the cylinder for Re=100 is shown in Fig.4. The vortex shedding phenomena behind the cylinder is well captured in Fig.4.

Table I represents the comparison of the mean drag coefficient obtained in the present study with those from the literature. In the present work, the drag forces are obtained by directly integrating all the momentum forcing values applied inside the body. A useful discussion on how to evaluate the drag force on a solid body in conjunction with the immersed boundary method can be found in Lai and Peskin [3]. From Table I, it is clear that the present values of mean drag coefficient and those of previous research results are in good agreement. This proves the reliability of the developed numerical model.

IV. Conclusion

This paper present a two–dimensional computational model to study the fluid dynamics of flow over a stationary circular cylinder based on an immersed boundary finite volume method. Accordingly, the cylinder is modeled by discrete number of immersed boundary points. The Lagrangian forces are computed using a feedback forcing method. The fluid dynamics of the fluid flow over and surrounding the cylinder is modeled using the momentum and continuity equations and the set of governing equations are solved on a staggered Cartesian grid using the fractional-step based finite-volume method.

Numerical simulations are performed using the developed numerical model to explore the flow behavior over the cylinder for four different Reynolds numbers. It is found that at low Reynolds numbers, flow around a circular cylinder is steady and symmetrical upstream and downstream. For high Reynolds number, alternate separation of vortices, which are convected and diffused away from the cylinder are observed. The instantaneous streamline pattern for four different Reynolds numbers is well captured. Further, mean drag coefficients are computed for each case and the values are compared with the previous researcher's results. It is interesting to note that a good agreement between the present numerical results and the previous researcher's results is achieved. This proves the validity of the developed numerical model.
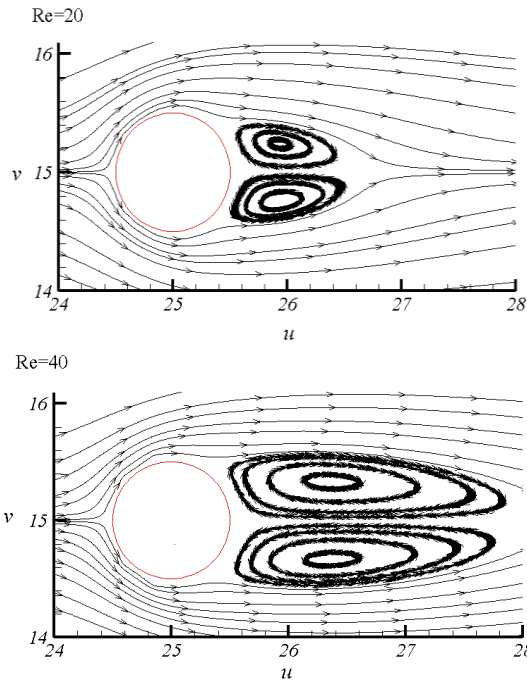
Re=20

Re=40
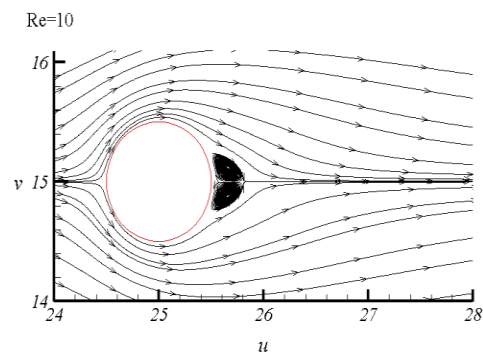
Fig. 2. Streamline visualization of the steady flow over a circular cylinder at Re=10, 20 and 40
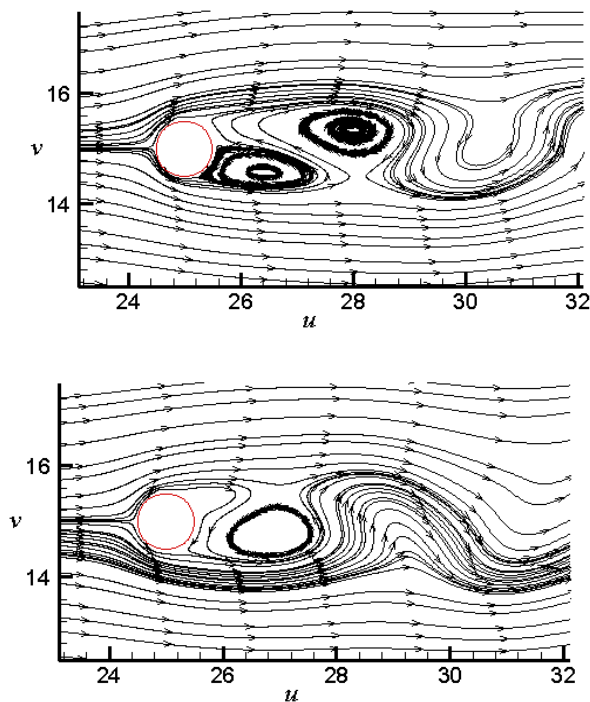

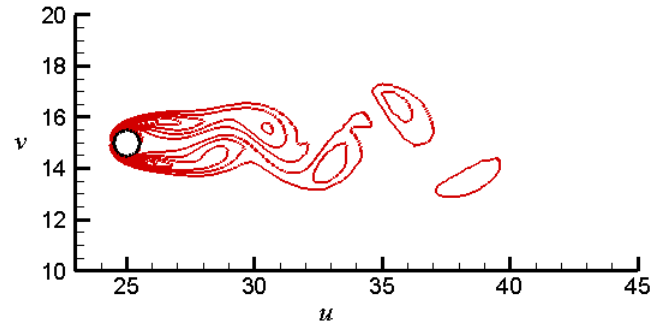Fig. 3. Instantaneous streamline pattern of the unsteady flow observed at Re=100

Fig. 4. Vorticity contours near the circular cylinder for Re=100

TABLE I
Comparison of mean drag coefficient for different Reynolds numbers

Reynolds Number	Su et al. [7]	Silva et al. [8]	Deng et al. [9]	**Present**
10	-----	2.81	2.98	**2.98**
20	2.20	2.04	2.06	**2.16**
40	1.63	1.54	1.52	**1.67**
100	1.40	1.39	1.30	**1.32**

Acknowledgement

References

i. Peskin, C.S., "Numerical analysis of blood flow in the heart," J Comput Phys., Vol.25, pp.221–249, 1977.

ii. Goldstein, D., Handler, R. and Sirovich, L., "Modeling a no-slip flow boundary with an external force field," J Comput Phys., Vol.105, pp.354–366, 1993.

iii. Lai, M.C. and Peskin, C.S., "An immersed boundary method with formal second-order accuracy and reduced numerical viscosity", J Comput Phys., Vol. 160, pp. 705-719, 2000.

iv. Shin, S.J., Huang, W.H. and Sung, H.J., "Assessment of regularized delta functions and feedback forcing schemes for an immersed boundary method," Int J Numer Methods Fluids., Vol.58, pp.263–286, 2008.

v. Peskin, C.S., "The immersed boundary method," Acta Numer., Vol.11, pp.1– 39, 2002.

vi. Kim, J., Kim, D. and Choi, H., "An immersed-boundary finite-volume method for simulations of flow in complex geometries, J Comput Phys.,Vol. 171, pp.132–150, 2001.

vii. Su, S.W., Lai, M.C. and Lin, C.A., "An immersed boundary technique for simulating complex flows with rigid boundary," Comput Fluids., Vol.36, pp.313–324, 2007.

viii. Lima E Silva, A.L.F., Silveira-Neto, A. and Damasceno, J.J.R., "Numerical simulation of two–dimensional flows over a circular cylinder using the immersed boundary method," J Comput Phys., Vol. 189, pp.351–370, 2003.

ix. Deng, J., Shao, X.M. and Ren, A.L., "A new modification of the immersed boundary method for simulating flows with complex moving boundaries," Int J Numer Methods Fluids., Vol. 52, pp.1195–1213, 2006.