Discrete-forcing immersed boundary method for fluid-structure interaction

Injae Lee\textsuperscript{1} and Haecheon Choi\textsuperscript{1}

1) School of Mechanical and Aerospace Engineering, Seoul National University, Seoul 151-744, KOREA

Corresponding Author: Haecheon Choi, choi@snu.ac.kr

ABSTRACT

In this study, a discrete-forcing immersed boundary (IB) method is proposed for the simulation of flow around a thin flexible body. The present method is based on the IB method for a stationary body proposed by Kim, Kim and Choi (J. Comput. Phys., 2001) and the dynamic equation for a thin flexible body is coupled with the incompressible Navier-Stokes equations. The incompressible Navier-Stokes equations are solved in an Eulerian grid system, and the motion of a thin flexible body is described in a Lagrangian grid system. The thin flexible body is modeled as a slender filament (2-dimensional case) or plate (3-dimensional case), and is segmented by finite number of blocks to express the shape deformation due to its elastic characteristics. Each block is then moved by the external and internal forces such as the hydrodynamic force from ambient fluid, elastic forces from shape deformation, and buoyancy force. With the present IB method, we simulate a few flow problems including a flexible filament and a flapping flag in a free stream, which are two- and three-dimensional fluid-structure interaction problems. The results from the flows around a flexible filament and a flapping flag with fixed leading edge show good agreement with those from previous studies, indicating the accuracy of the present IB method. We also show that the present IB method does not impose a severe limitation on the size of computational time step owing to the discrete-forcing approach.

NUMERICAL DETAILS

Navier-Stokes Equations

The governing equations are solved in the Eulerian coordinates using an IB method for a stationary body proposed by Kim et al. [1]:

\[
\frac{\partial u}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{1}{Re} \frac{\partial^2 u_i}{\partial x_j \partial x_j} + f_i, \quad (1)
\]

\[
\frac{\partial u_i}{\partial x_i} = q = 0, \quad (2)
\]

where \(x_i\) is the Cartesian coordinates, \(u_i\) is the corresponding velocity, \(p\) is the pressure, \(f_i\) is the momentum forcing, and \(q\) is the mass source/sink. All the variables are non-dimensionalized by the characteristic velocity and length scales, and \(Re\) is the Reynolds number. We use a semi-implicit fractional step method: Crank-Nicolson (CN2) method for
the viscous term and third-order Runge-Kutta (RK3) method for the convection term. The second-order central difference scheme is used for all the spatial derivative terms.

Dynamic Equation of Thin Flexible Body

The thin flexible body is described in the Lagrangian coordinates and is segmented by finite number of blocks. Each block is moved by the tension, bending, buoyancy and hydrodynamic forces:

\[
\frac{\rho}{2} \frac{\partial^2 X_i}{\partial t^2} = \frac{\partial}{\partial s_m} \left[ K^T_{nm} \left( \frac{\partial X_i}{\partial s_n} \frac{\partial X_j}{\partial s_m} - \frac{\partial X_j}{\partial s_n} \frac{\partial X_i}{\partial s_m} \right) \right] - \frac{\partial^2}{\partial s_m \partial s_n} \left( K^B_{nm} \frac{\partial^2 X_i}{\partial s_n \partial s_m} \right) - (\rho - 1) \frac{g}{g} F_r + \frac{F_i}{V},
\]

where \( \rho \) is the density ratio, \( X_i \) is the central position of each block, \( V \) is the volume of each block, \( K^T_{nm} \) is the tension coefficients, \( K^B_{nm} \) is the bending coefficients, \( F_r \) is the Froude number, and \( F_i \) is the hydrodynamic force.

Hydrodynamics Force

The hydrodynamic force acts as a coupling force between the fluid and thin flexible body, and is obtained directly from the integration of Navier-Stokes equations [2, 3]:

\[
F_i = \int_{V_b} \left( \frac{\partial u}{\partial t} + \frac{\partial u}{\partial x_i} \right) dV - \int_{V_b} f_i dV,
\]

where \( V_b \) is the volume of each block. As shown in Eq. (4), the hydrodynamic force on each block can be obtained directly from saved values of previous procedure. Therefore, Eq. (4) is efficient because it does not need any interpolation scheme on the surface of each block.

REFERENCES