1 Introduction

The phenomena of fluid-structure interaction are ubiquitous in nature such as flapping flags interacting with ambient fluid and swimming fish in water. The problems involving the coupled response of structures and flows are also of interest in various engineering areas such as aeronautical engineering, coastal engineering, and biomedical engineering. In such systems, the structures deform due to inertial, hydrodynamic, and internal forces; at the same time, they also exert forces on the surrounding fluid.

From a computational viewpoint, FSI simulations are challenging due to the following factors: (a) numerical issues (such as numerical instability) in handling two-way coupling between fluid and structure and (b) large mesh deformation when body-fitted mesh is used. The IB method overcomes the latter difficulty by adding a body force to the mesh. The IB method is used to mimic the effects of the immersed body on the flow. Numerical instabilities in handling two-way coupling between fluid and structure are also of interest in various engineering areas such as flapping flags interacting with ambient fluid and swimming fish in water. The problems involving the coupled response of structures and flows are also of interest in various engineering areas such as aeronautical engineering, coastal engineering, and biomedical engineering. In such systems, the structures deform due to inertial, hydrodynamic, and internal forces; at the same time, they also exert forces on the surrounding fluid.

In this work, we present an improved version of the direct-forcing immersed boundary (IB) method based on discrete stream function formulation for two- and three-dimensional incompressible flows [6]. In order to obtain an accurate prediction of local surface force, measures have been taken to suppress the unphysical spatial oscillations in the Lagrangian forcing. A fluid-structure interaction (FSI) solver has been developed by using the improved IB method for the fluid and the finite difference method for the structure. Several flow problems are simulated to validate our method. The testing cases include flows over a stationary cylinder and a stationary flat plate, two-dimensional flow past an inextensible flexible filament, and three-dimensional flow past a flapping flag. The results obtained in the present work agree well with those from the literature. [DOI: 10.1115/1.4026197]
motion of the flag are written as

The second case of FSI is the interaction of a flag (flexible plate) with a two-dimensional flow. The flexible filament can be regarded as a one-dimensional filament. The inextensible condition. Figure 1 shows the Lagrangian coordinate system $s$ on the filament. The length of the filament is $L$.

$$\text{Re} = \frac{UL}{\nu}$$

where $U$, $L$, and $\nu$ are the reference length, reference velocity, and kinematic viscosity, respectively. In our simulations, $U$ equals the inflow velocity, and $L$ is the diameter of the cylinder or the length of a filament (flag).

In this paper, two FSI simulations are performed. The first case is the interaction of an inextensible flexible filament with a two-dimensional flow. The flexible filament can be regarded as a one-dimensional flag. The governing equations for the motion of the filament are

$$\beta \frac{\partial^2 \mathbf{X}}{\partial t^2} - \frac{\partial}{\partial s} \left( T \frac{\partial \mathbf{X}}{\partial s} \right) + \frac{\partial^2}{\partial s^2} \left( \gamma \frac{\partial^2 \mathbf{X}}{\partial s^2} \right) = \beta F_r \frac{\mathbf{g}}{g} - \mathbf{F}$$

$$\frac{\partial \mathbf{X}}{\partial s} \frac{\partial \mathbf{X}}{\partial s} = 1$$

where $s$ is the Lagrangian coordinate along the arc length, $\mathbf{X}$ is the displacement vector, and $\beta$, $T$, and $\gamma$ are the mass ratio, dimensionless tension coefficient, and dimensionless bending rigidity, respectively. $\mathbf{F}$ is the Lagrangian force, and $\mathbf{g}$ is the acceleration of gravity and $g = |\mathbf{g}|$. $F_r$ is the Froude number. Equation (4) is equivalent to those used by Zhu and Peskin [9], Connell and Yue [10], and Huang et al. [11] for flexible structures. Equation (5) is the inextensible condition. Figure 1 shows the Lagrangian coordinates used in the simulation of a flapping filament.

The boundary condition at the free end is

$$T = 0, \quad \frac{\partial^2 \mathbf{X}}{\partial t^2} = (0,0), \quad \frac{\partial^3 \mathbf{X}}{\partial s^3} = (0,0)$$

At the fixed end, the simply supported boundary condition is applied.

$$\mathbf{X} = \mathbf{X}_0, \quad \frac{\partial \mathbf{X}}{\partial s} = (0,0)$$

The second case of FSI is the interaction of a flag (flexible plate) with a three-dimensional flow. The governing equations of the motion of the flag are written as

$$\beta \frac{\partial^2 \mathbf{X}}{\partial t^2} = \sum_{i,j=1}^{2} \left[ \frac{\partial}{\partial s_i} \left( \sigma_{ij} \frac{\partial \mathbf{X}}{\partial s_j} \right) - \frac{\partial^2}{\partial s_i \partial s_j} \left( \gamma \frac{\partial^2 \mathbf{X}}{\partial s_i \partial s_j} \right) \right] + \beta F_r \frac{\mathbf{g}}{g} - \mathbf{F}$$

$$\sigma_{ij} = \varphi_{ij} \left( \frac{\partial \mathbf{X}}{\partial s_i} \frac{\partial \mathbf{X}}{\partial s_j} - \delta_{ij} \right)$$

where $\varphi_{ij}$ and $\sigma_{ij}$ are the stretching and shearing coefficients, and $\sigma_{ij}$ are the stretching and shearing forces, and $\gamma_{ij}$ are the bending and twisting coefficients. $\delta_{ij}$ is the Kronecker symbol. When $i = j$, $\delta_{ij} = 1$, and when $i \neq j$, $\delta_{ij} = 0$. In this study, we consider an inextensible flag by making the stretching coefficients $\varphi_{11}$ and $\varphi_{22}$ sufficiently large. Equations (8) and (9) are equivalent to those used by Huang and Sung [12]. Figure 2 shows the Lagrangian coordinates used in the simulation of a flapping flag.

At the fixed end, we consider the simply supported condition in the simulation, i.e.,

$$\mathbf{X} = (0,0,s_2), \quad \frac{\partial^2 \mathbf{X}}{\partial s_1^2} = 0 \quad \text{at} \quad s_1 = 0$$

At the free boundaries, the boundary conditions are

$$\frac{\partial^2 \mathbf{X}}{\partial s_1^2} = 0, \quad \frac{\partial^3 \mathbf{X}}{\partial s_1^3} = 0 \quad \text{at} \quad s_1 = L$$

$$\frac{\partial^2 \mathbf{X}}{\partial s_2^2} = 0, \quad \frac{\partial^3 \mathbf{X}}{\partial s_2^3} = 0 \quad \text{at} \quad s_1 = 0 \text{ or } H$$

and

$$\sigma_{ij} = 0, \quad \gamma_{ij} = 0 \quad (i,j = 1,2)$$

2.2 Direct-Forcing IB Method Based on Discrete Stream Function Formulation. We use the discrete stream function approach for solving the Navier–Stokes equations. For more details of this approach, please refer to Ref. [6]. The discretized form of Eqs. (1) and (2) can be expressed by a matrix form as

$$\begin{bmatrix} \mathbf{A} & \mathbf{G} \\ \mathbf{D} & 0 \end{bmatrix} \begin{bmatrix} \mathbf{q}^{n+1} \\ \mathbf{p} \end{bmatrix} = \begin{bmatrix} \mathbf{r}^n \\ 0 \end{bmatrix} + \begin{bmatrix} \mathbf{bc}_1 \\ \mathbf{bc}_2 \end{bmatrix} + \begin{bmatrix} f \\ 0 \end{bmatrix}$$

where $q$, $p$, and $f$ are the discrete velocity flux, pressure, and body force, respectively, $q$ is related to the discrete velocity $u$ by multiplying the cell face area. $\mathbf{A}$, $\mathbf{G}$, and $\mathbf{D}$ are the implicit operator, gradient operator, and divergence operator, respectively. $r^s$ is the explicit right-hand side term of the momentum equation. $bc_1$ and $bc_2$ are the boundary condition vectors for the momentum and continuity equations, respectively.

In the discrete stream function approach, a discrete streamfunction $\tilde{s}$ is defined as

$$\tilde{s} = C \tilde{s}$$

where $C$ is the curl operator. This matrix is constructed in such a way that $\mathbf{D}$ and $\mathbf{C}$ enjoy the following relation:

$$\mathbf{DC} = 0$$

The definition in Eq. (15), together with the relation in Eq. (16), guarantees the discrete incompressibility. In the discrete stream function approach, another type of curl operator, the rotational operator $\mathbf{R}$, is also defined such that matrix $\mathbf{R}$ and matrix $\mathbf{C}$ enjoy the following relation:

$$\mathbf{R} = \mathbf{C}^T$$

By premultiplying the momentum equation with $\mathbf{R}$, the pressure can be eliminated from the system. This can be easily seen in the identity equation

$$\mathbf{RG} = -\mathbf{C}^T \mathbf{D}^T = -\left( \mathbf{DC} \right)^T = 0$$

Thus, the system of Eq. (14) is reduced to a single equation for $\mathbf{R}$ at each time step.

Fig. 1 Schematic representation of the Lagrangian coordinate system $s$ on the filament. The length of the filament is $L$. 

Transactions of the ASME
The representations of these operators are given in Ref. [6]. As to the time advancement, the diffusion term is implicit, the convection term is treated explicitly, and a three-step, second-order, low storage, Runge–Kutta scheme is used [6].

The matrix C^TAC is symmetric, positive-definite, and thus can be solved using the conjugate gradient (CG) method. A brief introduction of this iterative method, including the preconditioner and convergence criterion, is given in Ref. [6]. After solving Eq. (19), the velocity components can be recovered through Eq. (15). Although pressure is eliminated in this approach, if it is required, it can still be obtained through a postprocessing step, which is independent of the solution procedure for the velocity field.

The flow solver is based on the direct-forcing IB method in discrete stream function formulation. The original IB method in Ref. [6] can be summarized as follows:

\[ \mathbf{RAC}\tilde{s} = \mathbf{R} \tilde{r} + \mathbf{R} \tilde{f} \]
\[ \tilde{u} = \mathbf{C}\tilde{s} \]
\[ \tilde{U}^n(X_i) = \sum_x \tilde{u}(x)\delta_x(x - X_i)h^3 \]
\[ \sum_j \left( \delta_h(x - X_j)\delta_h(x - X_j)h^3 \right) F^2(X_i) = \frac{\tilde{U}^n(X_i) - \tilde{U}^n(X_i)}{\Delta t} \]
\[ \tilde{F}^{n+1} = \tilde{F}^{n-f} \]
\[ \tilde{f}^{n+1}(x) = \sum_j \tilde{F}^{n+1}(X_j)\delta_h(x - X_j)h^3 \]
\[ \mathbf{RAC}\tilde{s}^{n+1} = \mathbf{R}\tilde{r}^{n} + \mathbf{R}\tilde{f}^{n+1} \]

Fig. 2 Schematic representation of the Lagrangian coordinate system s₁ and s₂ on the flag. The longitudinal coordinate s₁ ranges from 0 to L, and the spanwise coordinate s₂ ranges from 0 to H, where L and H denote the length and width of the flag, respectively.

![Graph](image-url)
and trailing-edge of the zero-thickness plate is another source of oscillation in the temporal advancement.

Fourthly, for a slender body of zero-thickness, the discrete $\delta$-function [13] is adopted for the interpolation at endpoints. The mathematical form of this $\delta$-function [13] is

$$\phi(r) = \begin{cases} 1 - (1/2)|r| + (1/2)|r|^3, & |r| \leq 1.0 \\ 1 - (11/6)|r| + |r|^2 - (1/6)|r|^3, & 1.0 \leq |r| \leq 2.0 \\ 0, & |r| \geq 2.0 \end{cases}$$

(31)

It is found that the strong (unphysical) backflow at the leading- and trailing-edge of the zero-thickness plate is another source of spatial oscillation. The use of this type of kernel function can effectively eliminate the backflow at the endpoints. For the rest of the Lagrangian points (other than the endpoints), the regular four-point $\delta$ function (Eq. (29)) is used.

2.3 Discretization of the Structural Equations for Flexible Filament and Flag. The first case is a flexible filament. A finite difference method on staggered grid [11] is used to discretize Eq. (4) and Eq. (5). The displacement $X$ is defined at grid nodes while the tension $T$ is defined at the centroids of grid cells. Let $D_s$ denote the central difference operator with respect to $s$, for an arbitrary variable $\psi$, $D_s\psi$ means

$$D_s\psi = (\psi(s + \Delta s/2) - \psi(s - \Delta s/2))/\Delta s$$

(32)

$F_b$ denotes the bending force (i.e., the second term on the right-hand side of Eq. (4)). The solution procedure can be summarized as follows:

$$X' = 2X'' - X''''$$

(33)

$$\begin{align*}
(D_s(D_s(T^{1/2}D_sX')))_k^{1/2} & = 1 - 2(D_sX' - D_sD_sX')^{1/2} - (D_sX' - D_sD_sX')^{1/2} \\
& = \beta (1 - 2(D_sX' - D_sD_sX')^{1/2} - (D_sX' - D_sD_sX')^{1/2}) \\
& \beta (D_s\psi_{k} - (D_s\psi_{k})')_{k+1}^{1/2} - (D_s\psi_{k})_{k+1}^{1/2} - (D_s\psi_{k})_{k+1}^{1/2} - (D_s\psi_{k})_{k+1}^{1/2} \\
& \beta \frac{X_{k}^{n+1} - X_{k}^{n}}{\Delta t} = (D_s(T^{1/2}D_sX'^{n+1}))_{k} + (F_b)_{k}^{n} - F_b + \beta Fr \frac{g}{g} \\
& \beta \frac{X_{k}^{n+1} - X_{k}^{n}}{\Delta t} = (D_s(T^{1/2}D_sX'^{n+1}))_{k} + (F_b)_{k}^{n} - F_b + \beta Fr \frac{g}{g} \\
\end{align*}$$

(34)

(35)

---

Table 1 Influence of the ratio of Lagrangian grid width to Eulerian grid width on drag coefficients and oscillation in $U^*$ by using the four-point delta function in the simulation of flow over a cylinder at $Re = 40$. $N_l$ is the number of Lagrangian grid points.

<table>
<thead>
<tr>
<th>$\Delta s/h$</th>
<th>$N_l$</th>
<th>$C_d$</th>
<th>Oscillations in $U^*$</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.5</td>
<td>63</td>
<td>1.56</td>
<td>No</td>
</tr>
<tr>
<td>2</td>
<td>79</td>
<td>1.56</td>
<td>No</td>
</tr>
<tr>
<td>1.5</td>
<td>105</td>
<td>1.56</td>
<td>Yes</td>
</tr>
<tr>
<td>1</td>
<td>157</td>
<td>1.56</td>
<td>Yes</td>
</tr>
</tbody>
</table>

---

Fig. 4 Two ways of computing forcing component on a staggered mesh: (a) original way used in Ref. [6] and (b) more consistent way of computing forcing component. In (a), the Eulerian forcing (vector) is defined at cell centers, and each forcing component is interpolated (individually) to cell edges via simple average. In (b), each Eulerian forcing component is defined at cell edges, and no extra interpolation is needed.
Table 2 Comparisons of lift and drag coefficients, Strouhal numbers for flows over a cylinder at different Reynolds numbers

<table>
<thead>
<tr>
<th>Case</th>
<th>Re</th>
<th>$C_f$</th>
<th>$C_l$</th>
<th>$St$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Present</td>
<td>40</td>
<td>1.55</td>
<td>±0.00</td>
<td>–</td>
</tr>
<tr>
<td>Linnick and Fasel [14]</td>
<td>40</td>
<td>1.54</td>
<td>±0.00</td>
<td>–</td>
</tr>
<tr>
<td>Taira and Colonius [15]</td>
<td>40</td>
<td>1.54</td>
<td>±0.00</td>
<td>–</td>
</tr>
<tr>
<td>Wang and Zhang [6]</td>
<td>40</td>
<td>1.54</td>
<td>±0.00</td>
<td>–</td>
</tr>
<tr>
<td>Present</td>
<td>100</td>
<td>1.35</td>
<td>±0.33</td>
<td>0.166</td>
</tr>
<tr>
<td>Linnick and Fasel [14]</td>
<td>100</td>
<td>1.38</td>
<td>±0.33</td>
<td>0.169</td>
</tr>
<tr>
<td>Uhlmann [8]</td>
<td>100</td>
<td>1.45</td>
<td>±0.34</td>
<td>0.169</td>
</tr>
<tr>
<td>Wang and Zhang [6]</td>
<td>100</td>
<td>1.33</td>
<td>±0.32</td>
<td>0.166</td>
</tr>
<tr>
<td>Present</td>
<td>200</td>
<td>1.34</td>
<td>±0.69</td>
<td>0.197</td>
</tr>
<tr>
<td>Linnick and Fasel [14]</td>
<td>200</td>
<td>1.34</td>
<td>±0.69</td>
<td>0.197</td>
</tr>
<tr>
<td>Taira and Colonius [15]</td>
<td>200</td>
<td>1.35</td>
<td>±0.68</td>
<td>0.196</td>
</tr>
<tr>
<td>Wang and Zhang [6]</td>
<td>200</td>
<td>1.32</td>
<td>±0.69</td>
<td>0.197</td>
</tr>
</tbody>
</table>

Table 3 Grid refinement results and convergence order obtained by using flow past a cylinder at Re = 40

<table>
<thead>
<tr>
<th>$h/D$</th>
<th>$C_f^{(0)}/C_f^{(1)}$</th>
<th>$C_l^{(2)}$</th>
<th>$O_{in}$</th>
<th>$C_f^{(0)}$</th>
<th>$C_l^{(2)}$</th>
<th>$O_{in}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1/16</td>
<td>1.04e-1</td>
<td>–</td>
<td>–</td>
<td>7.24e-2</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>1/32</td>
<td>3.50e-2</td>
<td>2.97</td>
<td>1.57</td>
<td>2.36e-2</td>
<td>3.07</td>
<td>1.62</td>
</tr>
<tr>
<td>1/64</td>
<td>1.21e-2</td>
<td>2.91</td>
<td>1.54</td>
<td>7.98e-3</td>
<td>2.99</td>
<td>1.58</td>
</tr>
<tr>
<td>1/128</td>
<td>4.19e-3</td>
<td>2.87</td>
<td>1.52</td>
<td>2.65e-3</td>
<td>2.95</td>
<td>1.56</td>
</tr>
<tr>
<td>1/256</td>
<td>1.26e-3</td>
<td>3.32</td>
<td>1.73</td>
<td>8.20e-4</td>
<td>3.25</td>
<td>1.70</td>
</tr>
</tbody>
</table>

3 Numerical Validations and Results

3.1 Flows Over a Cylinder. To validate the solver, first the numerical study of flow over a stationary cylinder at Re = 40, 100, 200 is conducted. The simulation is performed in a rectangular domain of 30D × 40D, where D is the diameter of the cylinder. The cylinder is placed on the centerline with its center being 10D away from the inlet. The grid size in the vicinity of the cylinder (a region of 2D × 2D) is 0.02D. The grids are stretched to the boundaries with an expansion factor of 1.05, and the maximum grid size is 0.2D. The Lagrangian points are evenly distributed along the circumference of the circular cylinder and $\Delta/s = 2.0$. The time step is chosen such that the Courant–Friedrichs–Lewy (CFL) number never exceeds 0.5.

First, we compute lift and drag by the summation of the two components of the Lagrangian forcing, respectively. As listed in Table 2, the lift and drag coefficients obtained in the present study agree well with those from the references.

A refinement study is also performed employing the improved IB method to assess its convergence behavior. We choose the flow over a stationary cylinder at Re = 40 as a benchmark problem. We use uniform grids in this test and the computational domain is the same as that aforementioned. Spatial and temporal step sizes are reduced simultaneously by fixing the CFL number to 0.5 in all cases. The Lagrangian points are evenly distributed along the circumference of the circular cylinder and $\Delta/s = 2.0$. Since there is no analytical solution for this problem, the numerical results are compared to the reference solution obtained on a very fine grid with the grid width of $(1/152)D$ (where D is the diameter of the cylinder). Thus, the $L_{\infty}$ -error and $L_2$ -error can be defined as

$$e_{\infty} = \max \{ u_{1i} - u_{2i}^{ref} \}$$

$$e_2 = \sqrt{\frac{1}{N_x N_y} \sum_{i=1}^{N_x} \sum_{j=1}^{N_y} (u_{ij} - u_{ij}^{ref})^2}$$

The order of convergence $O$ is computed by using different errors obtained on successively refined grids, i.e.,

$$O = \frac{\log (e_2^{(b)}) - \log (e_2^{(0)})}{\log 2}$$

Table 3 shows that the convergence order of this method is between 1 and 2 (around 1.55). This is consistent with the results for many variants of IB methods, which use a second-order accurate basic N–S solver.

Next, we compute the pressure and skin-friction for the case of Re = 40 by projecting the Lagrangian forcing on the local normal and tangential direction, respectively. The distributions of the

---

Journal of Fluids Engineering


---

Equation (34) is the discretized form of the Poisson equation, which can be derived from Eq. (4) and the inextensibility condition Eq. (5) [11]. With the boundary condition in Eq. (6) and Eq. (7), the filament’s motion can be solved directly by using the CG method.

The second case is a flapping flag. A finite difference method is used to discretize Eqs. (8) and (9). We use a uniform mesh with mesh width $\Delta s$ in both $s_1$ and $s_2$ directions. For an arbitrary variable $\psi$, the first-order central differences of $\psi$ along $s_1$ and $s_2$ are

$$\frac{(D_1 \psi)_{i+\frac{1}{2}j}}{\Delta s} = \frac{\psi_{i+1,j} - \psi_{i,j}}{\Delta s}$$

$$\frac{(D_2 \psi)_{i+\frac{1}{2}j}}{\Delta s} = \frac{\psi_{i+1,j} - \psi_{i,j}}{\Delta s}$$

The second-order central difference and cross difference are expressed as

$$\frac{(D_{11} \psi)_{ij}}{\Delta s} = \frac{(D_1 \psi)_{i+\frac{1}{2}j} - (D_1 \psi)_{i-\frac{1}{2}j}}{\Delta s}$$

$$\frac{(D_{22} \psi)_{ij}}{\Delta s} = \frac{(D_2 \psi)_{i+\frac{1}{2}j} - (D_2 \psi)_{i-\frac{1}{2}j}}{\Delta s}$$

$$\frac{(D_{12} \psi)_{ij}}{\Delta s} = \frac{(D_1 \psi)_{i+\frac{1}{2}j} + (D_2 \psi)_{i+\frac{1}{2}j} + (D_1 \psi)_{i-\frac{1}{2}j} - (D_2 \psi)_{i-\frac{1}{2}j}}{\Delta s}$$

$$\frac{(D_{21} \psi)_{ij}}{\Delta s} = \frac{(D_1 \psi)_{i+\frac{1}{2}j} + (D_2 \psi)_{i+\frac{1}{2}j} - (D_1 \psi)_{i-\frac{1}{2}j} - (D_2 \psi)_{i-\frac{1}{2}j}}{\Delta s}$$

where $D_{11}$ and $D_{22}$ denote the second-order difference along the $s_1$ and $s_2$ direction, $D_{12}$ and $D_{21}$ are cross differences.

Using the definitions above, the discretized form of the flag’s equations (Eqs. (8) and (9)) can be written as

$$\sigma_{ij} = \phi_{ij} (D \mathbf{X} D \mathbf{X} - \delta_{ij})$$

$$\beta \frac{X_{i+1} - 2X_i + X_{i-1}}{\Delta t} = \sum_{j=1}^{N_x} \left[ \left( \sigma_{ij} D \mathbf{X} - D_{ij} \phi X \right) [n+1] - \mathbf{F} \right]$$

Together with the boundary conditions (Eqs. (10)–(13)), the flag’s motion can be solved by using the CG method.

For the coupling of the fluid and structure, we use a staggered (or loosely-coupled) method, in which the flow solver and structure solver are alternatively advanced by one step in time. In the framework of the direct-forcing IB method, the velocity of the flitament obtained in the structural solver provides the boundary condition for the fluid solver, i.e., $\mathbf{u}^d = \mathbf{X}$, while the Lagrangian force $\mathbf{F}$ determined in the flow solver acts as the source term in the structural equation.
pressure coefficient \(C_p\) and skin-friction coefficient \(C_f\) obtained by using the improved method are compared with the reference solutions in Ref. [16] and the solution by using the original method proposed in Ref. [6] (see Fig. 6). It is seen that a good agreement between the present result and the reference solution in Ref. [16] has been achieved. From this figure, it is also seen that the result obtained by using the method in Ref. [6] exhibits large spatial oscillations in both pressure and skin-friction.

3.2 Flows Over a Flat Plate. We then simulate the flow over a flat plate at \(Re = 200\) and two angles of attack (0 deg and 10 deg). The purpose of this validation is to test the accuracy of force distribution prediction for a slender body of zero-thickness. In this simulation, we use a rectangular domain of \(30L \times 20L\). The flat-plate is placed on the centerline with its center being 10L away from the inlet. The grid size in the vicinity of the plate (a region of \(2L \times 2L\)) is 0.01L. The time step is chosen such that the CFL number never exceeds 0.5. For reference purpose, we seek the solution of this problem by using the commercial computational fluid dynamics software-FLUENT. A body-fitted unstructured mesh with 140,000 cells is used in the computation by FLUENT. The mesh resolution used in FLUENT is comparable to that in the in-house flow solver (with the thickness of the plate represented by three mesh points in FLUENT). Other numerical settings in FLUENT are: second-order upwind scheme for convection, second-order central scheme for diffusion, and first-order Euler scheme for time advancing.

The distributions of pressure (difference) and skin-friction, obtained by using the improved method, by using the improved method but without the “negative-tailed” \(\delta\) function and by using FLUENT, are plotted in Fig. 7. It is seen that the agreement between the result obtained using the improved method and the one using FLUENT is reasonably well. The result using the improved method but without the “negative-tailed” \(\delta\) function exhibits some oscillations near the leading- and trailing-edge of the flat plate.

3.3 Flapping of Flexible Filament. Before performing any FSI simulation, we first validate the stand-alone structure solver
by simulating a flexible filament moving under gravity in vacuum. The simply supported (pinned) boundary condition is used at one end, and the free-end boundary condition is used at the other (see Fig. 1). The initial position condition of the filament is given by \[ X(s,0) = X_0 + (L - s)(\cos k, \sin k) \] \[ \frac{\partial X(s,0)}{\partial t} = (0,0) \] where \( k \) is a constant and \( X_0 = (0,0) \). At \( t = 0 \), the filament is released and starts swinging due to the gravitational force.

We use \( \beta = 1.0, L = 1.0, Fr = 10.0, \gamma = 0, \) and \( k = 0.1\pi \) as the control parameters. As shown in Fig. 8, the numerically predicted free-end position agrees well with the analytical solution in Ref.[11].

We then simulate the interaction of a flexible filament with a free stream at \( Re = 200 \). We use a computational domain of \( 16L \times 10L \). The distance between the leading edge of the filament and the inlet is \( 6L \). The mesh size is \( 0.01L \) in the vicinity of the filament (a region of \( 6L \times 2L \)). The number of the Lagrangian points representing the immersed filament is 50. The parameters used here are \( \beta = 1.5, Fr = 0.5 \), and \( L = 1.0 \). To trigger the instability, the filament is initially placed inclined at an angle of \( 0.1\pi \) with respect to the flow direction. Figure 9 shows the vorticity distribution in the wake for \( \gamma = 0.0015 \). Figure 10 shows the time histories of the \( y \)-position of the trailing edge for two different bending rigidities. It is seen that the present results agree well with those from Ref. [11] for both cases. Some discrepancies in phase and amplitude are due to the following two factors. First, although in this work we use the same algorithm as that in Ref. [11] in solving the structure equation, the basic solver for solving the N–S equation is different. Second, in Ref. [11] very small time steps are used (CFL number is only around 0.11) due to the stability restriction. In the present method, the stability restriction on time step is much looser, and we use larger time steps (CFL is around 0.5).

3.4 Three-Dimensional Simulation of a Flapping Flag. The 3D simulation of a flapping flag is also performed in this paper.
control parameters, with a free stream of Re = 100. The computational domain is $8L \times 8L \times 4L$, with the mesh size being 0.02L in the vicinity of the flag (a domain of $2L \times 2L \times 2L$). The total number of cells is $2.2 \times 10^6$. Other control parameters are $\beta = 1.0$, $Fr = 2.0$, and $L = H = 1.0$. The flag is initially held at an angle of $k = 0.1\pi$ from the XZ plane, as expressed by $X(s_1, s_2)|_{t=0} = (s_1 \cos k, s_1 \sin k, s_2 - H/2)$.

Figure 11 shows the instantaneous shape of a flapping flag in the three-dimensional simulation. The flag sags down slightly due to the gravitational force. The rolling motion of the upper corner is also seen. These observations are consistent with the report in Ref. [12]. Figure 12 shows the time histories of the transverse displacement of point A in Fig. 2 for Fr = 0.0. Both the result of the present study and that of Huang and Sung [12] are plotted in the figure. An excellent agreement between the two results is clearly seen.

4 Conclusions

In this study, an FSI solver is developed for the study of slender structures interacting with fluid. The present solver couples a direct-forcing IB method based on discrete stream function formulation for fluid flow and a staggered-grid finite difference method for the structural motion. Modifications to the original IB method are made to suppress the unphysical spatial oscillations in the force distribution on the surface of the structure. The solver is validated by a series of problems, including flows over stationary circular cylinder and flat plate. FSI simulations performed in this paper include 2D flow over an inextensible filament and 3D flow over a flapping flag. The results obtained in the present study agree well with those in the literature.

Acknowledgment

This work is supported by the Chinese Academy of Sciences under Project Nos. KJCX-SW-L08 and KJCX3-SYW-S01, and the National Natural Science Foundation of China under Project Nos. KJCX-SW-L08 and KJCX3-SYW-S01, and the National Supercomputing Center in Tianjin (NSCC-TJ) for the allocation of computing time.

Nomenclature

- $A$: implicit operator in discretization
- $bc$: boundary condition vectors in discretization
- $C$: curl operator
- $C_d$: drag coefficient
- $C_f$: skin friction coefficient
- $C_l$: lift coefficient
- $C_p$: pressure coefficient
- $D$: diameter of the circular cylinder
- $f$: Eulerian forcing in fluid momentum equation
- $F$: Lagrangian forcing in structure equation
- $F_c$: discrete Lagrangian force component
- $Fr$: Froude number
- $g$: magnitude of gravitational acceleration
- $G$: gravitational acceleration
- $H$: width of the flag
- $k$: initial inclined angle of the filament or flag
- $L$: length of the flat-plat, filament, or flag
- $O$: order of convergence
- $p$: fluid pressure
- $q$: discrete velocity flux
- $r$: explicit term in the discretization of the momentum equation
- $R$: rotational operator
- $Re$: Reynolds number
- $s$: Lagrangian coordinate
- $\dot{s}$: discrete stream function
- $T$: tension coefficient of the filament
- $u$: fluid velocity
- $u_d$: discrete fluid velocity component
- $U$: Lagrangian velocity at the boundary
- $x$: Eulerian coordinate
- $X$: displacement of the structure
- $\alpha$: angle of attack
- $\beta$: mass ratio
- $\gamma$: bending coefficient of the filament or bending and twisting coefficients of the flag
- $\Gamma$: boundary of the flexible body
- $\delta$: regularized $\delta$ function
- $\epsilon$: error
- $\theta$: angle between radical direction and horizontal direction
- $\nu$: kinematic viscosity
- $\sigma$: stretching and shearing coefficients of the flag
- $\Phi$: fluid domain

Subscripts

- $i,j$: index in tensorial material coefficients of the structure
- $i,j$: index of node in finite different discretization
- $k$: index of node in finite different discretization
- $n$: index of time step

References


