# FEDSM2013-16472

## AN IMPROVED DIRECT-FORCING IMMERSED BOUNDARY METHOD FOR FLUID-STRUCTURE INTERACTION SIMULATIONS

Xing Zhang<sup>\*</sup> LNM, Institute of Mechanics, Chinese Academy of Sciences, Beijing 100190 China Xiaojue Zhu LNM, Institute of Mechanics, Chinese Academy of Sciences, Beijing 100190 China Guowei He LNM, Institute of Mechanics, Chinese Academy of Sciences, Beijing 100190 China

## ABSTRACT

Simulation of fluid-structure interaction (FSI) of flexible bodies are challenging due to complex geometries and freely moving boundaries. Immersed boundary method has found to be an efficient technique for dealing with FSI problems because of the use of non-body-fitted mesh and simple implementation. In the present work, we developed a FSI solver by coupling a direct forcing immersed boundary method for the fluid with a finite difference method of the structure. Several flow problems are simulated to validate our method. The testing cases include flow over a stationary cylinder and flat plate, two-dimensional flow past an inextensible flexible filament and threedimensional flow past a flag. The results obtained agree well with those from previously published literatures.

## INTRODUCTION

The phenomena of fluid-structure interaction (FSI) are ubiquitous in nature such as flapping flags interacting with ambient fluid and fish swimming in water. The problems involving the coupled response of structures and flows are 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 facts: a) numerical issues (such as instability) in handling two-way coupling between fluid and structure; b) large mesh deformation when body-fitted mesh is used. The immersed boundary (IB) method overcomes the latter difficulty by using a non-body-fitted mesh and adding a body force to the momentum equation to enforce the no-slip boundary condition [1]. The IB method can be further classified into two types: continuous forcing and direct forcing [2]. In the continuous forcing approach, the forcing is incorporated into the continuous equations before discretization, whereas in the direct forcing approach, the forcing is introduced after the equations are discretized. The continuous forcing approach is often used for treating elastic boundaries whereas the direct forcing approach is originally designed for rigid-boundary problems.

In this paper, we developed a FSI solver by coupling a direct forcing IB method based on discrete stream function formulation [3] for the fluid and a finite different method for the structure. By using the original method proposed in [3], although an accurate prediction of total force can be achieved, unphysical spatial oscillation is observed in the force distribution. This oscillation is detrimental to the prediction of structure response in FSI. In this work, several modifications are made to improve this method. Firstly, the implicit forcing is replaced by an explicit forcing. Secondly, a more consistent way of computing each component of the forcing on a staggered mesh is proposed. Thirdly, for a slender body of zero thickness, the discrete delta-function with a 'negative-tail' is adopted for the interpolation at the endpoints. Numerical simulations (including FSI) are performed to test the efficacy of the modifications. It is found that the measures taken can successfully reduce the oscillation and the results obtained agree well with those from the literatures.

The rest of the paper is arranged as follows. The numerical method are briefly introduced in Section 2. Three testing cases, including flow over a stationary cylinder, vortex-induced

<sup>\*</sup> Corresponding author: zhangx@lnm.imech.ac.cn

vibration of elastically mounted cylinder and flexible filament in free stream are presented in section 3. Finally, conclusions are drawn in section 4.

### NUMERICAL METHODS

The fluid motion is governed by the incompressible Navier-Stokes equations, which in dimensionless form are written as

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

$$\nabla \cdot \mathbf{u} = 0.$$
(1)

where u is the velocity vector, p the pressure and Re the Reynolds number. f is the Eulerian body-force that is used to mimic the effects of the immersed body on the flow. The Reynolds number is defined as Re = UL/v, where U, L and v are the reference length, reference velocity and kinematic viscosity, respectively.

The flow solver is based on discrete stream function formulation and direct forcing IB method, for more details, please refer to [3]. Although an accurate prediction of total force can be achieved by using this method, unphysical spatial oscillation is observed in the force distribution on the surface of the immersed body. This oscillation is detrimental to the prediction of structure response in FSI. In this work, several modifications are made to improve this method.

Firstly, the implicit forcing is replaced by the explicit forcing proposed by Uhlmann [4]. Secondly, a more consistent way of computing the momentum forcing on a staggered mesh is proposed (see Figure 1). Instead of interpolating at cell centers to obtain the forcing vectors and then averaging them to face centers to obtain individual components, the x- and ycomponents of the forcing vectors are computed separately and directly at face centers by interpolation. It is well-known that the interpolation of non-smooth function (such as velocity) tends to create oscillation and reduce accuracy. These two measures reduce the number of velocity interpolation required in computing the momentum forcing component and thus reduce the spatial oscillations produced. Thirdly, for a slender body of zero thickness, the discrete delta-function with a 'negative-tail' is adopted for the interpolation at endpoints. The mathematical form of this delta function is

$$\delta(r) = \begin{cases} 1 - (1/2)|r| - |r|^2 + (1/2)|r|^3, & |r| \le 1.0\\ 1 - (11/6)|r| + |r|^2 - (1/6)|r|^3, & 1.0 \le |r| \le 2.0\\ 0, & |r| \ge 2.0. \end{cases}$$
(2)



Figure 1. Two ways of computing forcing component on a staggered mesh: (a) original way used in Wang and Zhang [3]; (b) more consistent way of computing forcing component. In (a), the Eulerian forcing (vector) are 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.

It is found that the strong (unphysical) backflow at the leadingand trailing-edge is another source of spatial oscillation. The use of this type of kernel function can effectively eliminate the backflows at endpoints. For the rest of the Lagrangian points, a regular 3-point delta function [1] is used.

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 governing equations for the motion of the filament is written as

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

$$\frac{\partial \mathbf{X}}{\partial s} \cdot \frac{\partial \mathbf{X}}{\partial s} = 1,$$
(3-2)

where  $\rho$  is density ratio, T is the tension force,  $\gamma$  is the bending rigidity and Fr is the Froude number. g is the acceleration of gravity and F is the force exerted on the structure by the fluid.

A finite difference method on staggered grid [5] is used to discretize Eq. (3). 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* and  $\mathbf{F}_b$  denote the bending force (i.e., the second term on the righthand-side of Eq. (3-1)). The solution procedure can be summarized as follows.

$$\mathbf{X}^* = 2\mathbf{X}^n - \mathbf{X}^{n-1},\tag{4-1}$$

$$(D_{s}(D_{s}(T^{n+l/2}D_{s}\mathbf{X}^{*})))_{k+l/2} \cdot (D_{s}\mathbf{X}^{*})_{k+l/2} = \beta \frac{1-2(D_{s}\mathbf{X}^{*} \cdot D_{s}\mathbf{X}^{*})^{n} + (D_{s}\mathbf{X}^{*} \cdot D_{s}\mathbf{X}^{*})^{n-1}}{2\Delta t^{2}} - \beta (D_{s}\mathbf{U} \cdot D_{s}\mathbf{U})_{k+l/2}^{n} - (D_{s}\mathbf{X}^{*})_{k+l/2} \cdot (D_{s}(\mathbf{F}_{b}^{n} - \mathbf{F}^{n}))_{k+l/2},$$

$$(4-2)$$

$$\beta \frac{\mathbf{X}_{k}^{n+1} - \mathbf{X}_{k}^{*}}{\Delta t^{2}} = \left(D_{s}(T^{n+1/2}D_{s}\mathbf{X}^{n+1})\right)_{k} + \left(\mathbf{F}_{b}\right)_{k}^{n} - \mathbf{F}_{k}^{n} + \beta Fr \frac{\mathbf{g}}{g}.$$
(4-3)

Eq. (4-2) is the discretized form of the Poison equation which can be derived from Eq. (3-1) and the inextensibility condition (Eq. (3-2)).

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

$$\beta \frac{\partial \mathbf{X}}{\partial t^2} = \sum_{i,j=1}^2 \left[ \frac{\partial}{\partial s_i} (\sigma_{ij} \frac{\partial \mathbf{X}}{\partial s_j}) - \frac{\partial^2}{\partial s_i \partial s_j} (\gamma_{ij} \frac{\partial^2 \mathbf{X}}{\partial s_i \partial s_j}) \right] + \beta Fr \frac{\mathbf{g}}{g} - \mathbf{F},$$
(5)

where  $\sigma_{ij}$  (i, j = 1 or 2) are the stretching and shearing coefficients;  $\gamma_{ij}$  (i, j = 1 or 2) are the bending and twisting coefficients. In this study, we assume that the flag is inextensible by making the stretching coefficients  $\sigma_{11}$  and  $\sigma_{22}$  sufficiently large. Moreover, large  $\sigma_{12}$  and  $\sigma_{21}$  are also used to resist in-plane shear strain.

In the structure solver, Eq. (5) is discretized by using the finite different scheme proposed in [6], which is very similar to Eq.(4-2).

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 immersed boundary method, the velocity of the filament obtained in the structural solver provides one boundary condition for the fluid solver, i.e.,  $\mathbf{U}_{b} = \dot{\mathbf{X}}$ ; while the Lagrangian force **F** determined in the flow solver acts as the source term in the structural equation.

#### NUMERICAL VALIDATIONS AND RESULTS

#### I. Flows over a cylinder and a flat plate

To validate the solver, first the numerical study of flow over a stationary cylinder at Re = 40 is conducted. The simulation is performed in a rectangular domain of  $60D \times 40D$ , where D is the diameter of the cylinder. The grid size in the vicinity of the cylinder (a region of  $2D \times 2D$ ) is 0.04D. The grids are stretched to the boundaries with an expansion factor of 1.05 and the maximum grid size is 0.5D. The Lagrangian points are evenly distributed along the circumference of the circular cylinder such that the inter-distance equals the local size of the Eulerian grid approximately. As that listed in Table 1, the mean drag coefficient C<sub>d</sub> obtained in the present study agrees well with the those from the references.

Table 1. Comparisons of drag coefficients for Re=40.

	Taira and Colonius [7]	Linnick and Fasel [8]	Present	-
$C_d$	1.54	1.54	1.55	

The distributions of the pressure coefficient  $C_p$  and skinfriction coefficient  $C_f$  obtained by using the improved method are also compared with reference solutions in [8] and the solutions using the original method proposed in [3]. It is seen that a good agreement between the present result and the reference solution in [8] has been achieved (see Figure 2). The slight discrepancy in the skin-friction coefficient near the shoulder is attributed to the insufficient grid resolution near this region (the grid size is 0.05*D*). From this figure, it is also seen that the result obtained by using the method in [3] exhibits large spatial oscillations in both the pressure and skin-friction coefficients.

We then simulate the flow over a flat plate at Re = 200 and two angles of attack (0° and 10°). 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  $30D \times 20D$ . The grid size in the vicinity of the plate (a region of  $2D \times 2D$ ) is 0.02*D*. For reference purpose, we also seek the solution of this problem by using the commercial CFD 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 3 mesh points). Other numerical settings in FLUENT are: second order upwind scheme for convection; second order central scheme for diffusion; first order Euler scheme for time advancing.

The distributions of pressure (difference) and skin-friction, obtained by using the improved method, the improved method but without the 'negative-tailed' delta function and FLUENT are plotted in Figure 3. 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.



Figure2. Distribution of (a) pressure coefficient  $C_{\rm p}$  and (b) skin-friction coefficient  $C_{\rm f}.$ 





Figure 3. Distributions of pressure coefficient difference  $\Delta c_p$  and skin-friction coefficient (C<sub>f</sub>) along the flat-plate surface at Re = 200 and two angles of attack. (a) C<sub>f</sub> for  $\alpha = 0^\circ$ ; (b)  $\Delta c_p$  for  $\alpha = 10^\circ$ ; (c) C<sub>f</sub> for  $\alpha = 10^\circ$ .

#### II. 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  $(\partial^2 \mathbf{X}/\partial s^2 = (0,0), \partial^3 \mathbf{X}/\partial s^3 = (0,0))$  is used at the other (see Figure 4). The initial position condition of the filament is given by  $\mathbf{X}(s,0) = \mathbf{X}_0 + (L-s)(\cos k, \sin k), \partial \mathbf{X}(s,0)/\partial t = (0,0)$ , 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.



Figure 4. Schematic of the Lagrangian coordinate system s on the filament. The length of the filament is *L*.

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 figure 5, the numerically predicted free-end position agrees well with the analytical solution in [5].



Figure 5. Comparison of the predicted free-end position with the analytical result.

We then simulate the interaction of a flexible filament with a free stream at Re = 200. We use a computational domain of  $16L \times 10$  L. The distance between the leading edge of the filament and the inlet is 6 L. The mesh size is 0.02 L in the vicinity of the filament (a region of  $6 L \times 2 L$ ). The number of the Lagrangian points representing the immersed filament is 50. The parameters used here are  $\beta = 1.5$ , Fr = 0.5, 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 6 shows the vorticity distribution in the wake for  $\gamma = 0.0015$ . Figure 7 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 [5] for both cases.



Figure 6. Vorticity contours in wake of a flapping flexible filament.



Figure 7. Time history of y-position of the trailing edge: (a)  $\gamma = 0.0015$ ; (b)  $\gamma = 0.0$ . Solid line denotes the result of the present study; square denotes the result from Huang, Jing & Sung [5].

#### III. Three-dimensional simulation of a flapping flag

The 3D simulation of a flapping flag is also performed in this paper. The simply supported (pinned) boundary condition  $\mathbf{X} = (0, 0, s_2), \ \partial^2 \mathbf{X} / \partial s_1^2 = \mathbf{0}$  is used at the pole ( $s_1 = 0$ ) (see Figure 8). The conditions at the free boundaries are:  $\partial^2 \mathbf{X} / \partial s_1^2 = \mathbf{0}, \ \partial^3 \mathbf{X} / \partial s_1^3 = \mathbf{0}$ , at the free end  $s_1 = L$ ;

 $\partial^2 \mathbf{X} / \partial s_2^2 = \mathbf{0}, \partial^3 \mathbf{X} / \partial s_2^3 = \mathbf{0}, \text{ at other two free ends } s_2 = 0 \text{ and } s_2 = \mathbf{H}.$ 

In the simulation of the flapping flag, we use  $\sigma_{12} = \sigma_{21} = 10.0$ ,  $\sigma_{11} = \sigma_{22} = 10.0$ ,  $\gamma_{12} = \gamma_{21} = \gamma_{11} = \gamma_{22} = 0.0001$  and as the control parameters, with a free stream of Re = 500. 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 million. Other control parameters are  $\beta = 1.0$ , Fr = 2.0 and L = H = 1.0. The flag is initially held at an angle of  $\alpha = 0.1\pi$  from the X<sub>1</sub>X<sub>3</sub> plane, as expressed by  $\mathbf{X}(s_1, s_2)|_{t=0} = (s_1 \cos \alpha, s_1 \sin \alpha, s_2 - H/2)$ .



Figure 8. Schematic of the Lagrangian coordinate system  $(s_1, s_2)$  on the flag. The width and length of the flag are H and L, respectively.

Figure 9 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 [10]. Figure 10 shows the time histories of the transverse displacements of points A in Figure 8 for Fr = 0.0. Both the result of present study and that of Huang & Sung [10] are plotted in the figure. An excellent agreement between the two results is clearly seen.



Figure 9. The instantaneous shape of a flapping flag in the three-dimensional simulation.



Figure 10. Time histories of the transverse displacements of points A in figure 8 for Fr = 0.0. Solid line denotes the result of the present study; square denotes the result from Huang & Sung [10].

#### CONCLUSIONS

In this study, a FSI solver is developed or the study of slender structures interacting with fluid. The present solver couples a direct forcing immersed boundary 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 immersed boundary 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 flow 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 literatures.

#### NOMENCLATURE

- $C_d$  drag coefficient
- $C_f$  skin friction coefficient
- $\dot{C_p}$  pressure coefficient
- *D* diameter of the circular cylinder
- *Fr* Froude number
- g magnitude of gravitational acceleration
- g gravitational acceleration
- **f** Eulerian forcing in fluid momentum equation
- **F** Lagrangian forcing in structure equation
- H width of the flag
- *k* initial inclined angle of the filament
- *L* length of the filament (or flag)
- *p* fluid pressure
- *r* independent variable in the delta function
- *Re* Reynolds number
- *s* Lagrangian coordinate
- *T* tension coefficient of the filament
- u fluid velocity
- **x** Eulerian coordinate
- **X** displacement of the structure

#### **Greek Symbols**

- $\alpha$  angle of attack
- $\beta$  mass ratio
- $\delta$  Regularized delta function

 $\gamma$  bending coefficient of the filament or also bending and twisting coefficients of the flag

 $\sigma$  stretching and shearing coefficients of the flag

## Superscripts

*n* index of time steps

## Subscripts

- *i*, *j* index in the tensorial material coefficients of structure
- k index of node in finite different discretization

## ACKNOWLEDGMENTS

This work was supported by Chinese Academy of Sciences under the Innovative Projects (KJCX-SW-L08) and (KJCX3-SYW-S01); National Natural Science Foundation of China under Project Nos.10325211, 10628206, 10732090, 10872201 and 11023001. The authors would like to thank the National Supercomputing Center in Tianjin (NSCC-TJ) for the allocation of computing time.

## REFERENCES

- [1] Peskin, C.S., 2002, "The immersed boundary method", Acta Numerica, **11**, pp. 479-517.
- [2] Mittal, R. and Iaccarino, G., 2005, "Immersed boundary methods", Annual Review of Fluid Mechanics, 37, pp. 239-261.
- [3] Wang, S.Z., and Zhang, X., 2011, "An immersed boundary method based on discrete stream function formulation for two- and three-dimensional incompressible flows", Journal of Computational Physics, 230, pp. 3479-3499.
- [4] Uhlmann, M., 2005, "An immersed boundary method with direct forcing for the simulation of particulate flows", Journal of Computational Physics, **209**, pp. 448-476.
- [5] Huang, W.X., Shin, S.J., and Sung, H.J., 2007, "Simulation of flexible filaments in a uniform flow by the immersed boundary method", Journal of Computational Physics, **226**, pp. 206-2228.
- [6] Huang, W.X. and Sung, H.J., 2009, "An immersed boundary method for fluid-flexible structure interaction", Computer Methods in Applied Mechanics and Engineering, 198, 2650-2661.
- [7] Taira, K. and Colonius, T., 2007, "The immersed boundary method: a projection approach", Journal of Computational Physics, 225, pp. 2118-2137.
- [8] Linnick, M.N., and Fasel, H.F., 2005, "A high-order immersed interface method for simulating unsteady incompressible flows on irregular domains", Journal of Computational Physics, 204, pp. 157-192.
- [9] Majumdar, S., Iaccarino, G. and Durbin, P., 2001, "RANS solvers with adaptive structured boundary nonconforming grids", Annual Research Briefs, Center for Turbulence Research, Stanford University, pp, 353-466.
- [10] Huang, W.X. and Sung, H.J., 2010, "Three dimensional simulation of a flapping flag in a uniform flow", Journal of

f time store