



HAL
open science

Moving immersed boundary method for fluid–solid interaction

Shang-Gui Cai, Abdellatif Ouahsine, Yannick Hoarau

► **To cite this version:**

Shang-Gui Cai, Abdellatif Ouahsine, Yannick Hoarau. Moving immersed boundary method for fluid–solid interaction. *Physics of Fluids*, 2022, 34 (5), pp.053307. 10.1063/5.0088302 . hal-03664341

HAL Id: hal-03664341

<https://hal.science/hal-03664341>

Submitted on 10 May 2022

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.

1 Moving immersed boundary method for fluid-solid interaction

2 Shang-Gui Cai (蔡尚旻),^{1,2} Abdellatif Ouahsine,¹ and Yannick Hoarau³

3 ¹*Sorbonne Universités, Université de Technologie de Compiègne,*
4 *CNRS, UMR 7337 Roberval, Centre de Recherche Royallieu, CS 60319,*
5 *60203 Compiègne Cedex, France*

6 ²*Aix Marseille Univ, CNRS, Centrale Marseille, M2P2 UMR 7340, Marseille 13451,*
7 *France*

8 ³*Université de Strasbourg, CNRS, ICUBE UMR 7357 Strasbourg, 67000,*
9 *France*

10 (*Electronic mail: shanggui.cai@univ-amu.fr)

11 (Dated: 2 April 2022)

12 A strongly coupled algorithm is presented for the incompressible fluid-rigid body interac-
13 tion using the moving immersed boundary method. The pressure and the boundary force
14 are treated as Lagrange multipliers to enforce the incompressibility and no-slip wall con-
15 straints. To compute the two unknowns from the velocity field, we adopt the fractional
16 step algorithm and successively apply the two constraints. A Poisson equation and a mov-
17 ing force equation are derived for the pressure and the boundary force respectively. As
18 both coefficient matrices are formulated to be symmetric and positive-definite, the result-
19 ing linear systems are solved efficiently with the conjugate gradient solver. The strongly-
20 coupled nonlinear fluid-solid system is achieved by a fixed point iteration. To improve the
21 computational efficiency, we only iterate the moving force equation with the rigid body
22 motion equation and the time-consuming pressure Poisson equation is solved once the
23 sub-iteration is finished. The proposed method is validated with various benchmark tests
24 and the results compare well with the literature.

25 I. INTRODUCTION

26 One primary challenge of fluid-solid interaction (FSI) simulation is dealing of complicated and
27 time-dependent interface between two physical fields. Body-fitted mesh methods, such as finite
28 volume or finite element discretization of fluid equations in arbitrary Lagrangian–Eulerian for-
29 mulation, need mesh-deforming or re-meshing regularly to accommodate the time-varying solid
30 boundaries. It is very costly for large scale problems and prone to mesh interpolation errors and
31 divergence due to severe mesh distortion. The Chimera/overset grid method¹ circumvents these
32 issues by employing multiple computational fluid domains covered by independent meshes. One
33 background mesh, usually a Cartesian mesh, is utilized to discretize the global domain, while the
34 others are body-fitted meshes surrounding the objects. The Chimera/overset grid method is ca-
35 pable to capture the boundary layer well at moderate and high Reynolds number, comparable to
36 classical body-fitted mesh methods, since grids can be allocated efficiently when aligning to solid
37 boundaries. However constructing a body-fitted mesh of high quality for complex geometries is
38 still non-trivial. Simple discretization method like finite difference may require a coordinate trans-
39 formation to work on a curvilinear mesh. In such circumstance finite volume or finite element
40 method is usually used and the resulting system is solved with sophisticated solvers, such as pre-
41 conditioned Krylov subspace methods (e.g. conjugate gradient) or algebraic multigrid method.
42 Moreover since the fluid equations are computed on different domains, it is difficult to ensure
43 conservation.

44 Alternatively, the immersed boundary method (IBM)^{2–8} adopts a surface mesh for the solid
45 boundary, thereby the fluid is only simulated on the background mesh. Using a simple fluid
46 mesh is advantageous not only for the ease of mesh generation but also for the accuracy mat-
47 ter. The Cartesian mesh produces the best mesh quality and hence high order accuracy can be
48 achieved. The matrix assembly is also simplified and the matrix condition is well preserved. Most
49 importantly, it facilitates the use of many simple and efficient methods developed for Cartesian
50 mesh. The finite difference discretization can be constructed into high order accuracy easily and
51 highly efficient solvers, such as geometrical multigrid method and fast Poisson solver, can be used
52 straightforward. Parallelization of program is relatively simple to a good efficiency. However as
53 no underlying mesh will conform to the solid boundary, the imposition of boundary condition at
54 the interface can be tricky. Interpolation methods are often selected to overcome this problem,
55 either by locally reconstructing the velocity^{9,10} or by adopting a source forcing function in the

56 momentum equation ⁵⁻⁸. In fact, the two approaches are essentially the same as they both rectify
57 the velocity in the vicinity of immersed boundary to produce solid effect¹¹.

58 For the source forcing approach, it is crucial to compute the boundary force accurately to
59 ensure a physical boundary condition. Uhlmann³ developed an efficient direct forcing IBM for
60 simulating particle suspensions, where an explicit formulation is used for the boundary force. As
61 a result, the no-slip wall condition is not satisfied and large errors occur at the interface. Kempe
62 and Fröhlich⁴ reduced the error through an additional forcing loop, which is solved with only a
63 few number of iterations without convergence. Therefore the no-slip wall condition will not be
64 fully verified. Full convergence however requires considerable computational efforts, thus it is
65 often relaxed in practice¹². Actually, the boundary force can be viewed as a Lagrange multiplier
66 for the no-slip constraint. an implicit immersed boundary projection method (IBPM) is proposed
67 by Taira and Colonius¹³ by integrating the boundary force and the pressure into a modified Pois-
68 son equation. The IBPM is very accurate since it enforces the divergence-free and the no-slip
69 constraints simultaneously. In spite of the mathematical rigor and completeness, the IBPM is less
70 efficient and stable than the explicit IBMs, as an enlarged system is formulated and evolves in time
71 for moving bodies. Instead of solving the pressure and the boundary force in one single system,
72 the moving immersed boundary method (MIBM) was proposed by Cai et al.⁵⁻⁸ to decouple the
73 pressure and the boundary force into two independent systems. The pressure Poisson coefficient
74 matrix is kept unchanged and hence preconditioners only need to be constructed in the initial step.
75 The boundary force is computed from a moving force equation after operator splitting. The size
76 of the corresponding matrix is considerably reduced as it is only proportional to the number of
77 Lagrangian markers defining the immersed boundary, which is much smaller than the fluid mesh
78 numbers. The MIBM is found very efficient and accurate for moving boundary problems. In this
79 work, we will extend the MIBM to fluid-solid interaction applications by incorporating the rigid
80 body dynamics.

81 Generally, two approaches can be used for the fluid-solid interaction time coupling: the mono-
82 lithic (direct) approach and the partitioned (segregated) approach. For the monolithic method, all
83 the unknowns from the fluid and solid equations are formulated into a single large system along
84 with interface conditions. While the partitioned method formulates the two subproblems into dif-
85 ferent systems and solve them separately. The monolithic approach is advantageous from the
86 stability point of view, but it requires considerable efforts on code modifications. The partitioned
87 approach can be further classified into two subgroups, namely the explicit coupling (weakly or

loosely coupled) and the implicit coupling (strongly or tightly coupled). Weak coupling is very efficient since it solves the two subproblems in a sequential manner, but it leaves the interface conditions unfulfilled. As a result, spurious energy is generated at the interface, which leads to unstable solutions, which is often called the added mass effect^{14,15}. The stability criterion mainly depends on the density ratio between the solid and the fluid. Below a certain threshold the calculation diverges immediately for any chosen time step for incompressible fluids¹⁶. The strongly coupled method iterates the two subproblems until the interface conditions are satisfied, thus it is comparable to the monolithic method. In the present work, we will design a partitioned but strongly coupled scheme with the moving immersed boundary method.

The organization of this paper is as follows. First we present the general fluid-solid interaction problem in the immersed boundary formulation. In the following part the MIBM and the strongly coupled partitioned scheme are presented in detail. Numerical simulations are then performed with the proposed method and validated with benchmark tests. Conclusions are drawn in the final section.

II. METHODOLOGY

Figure 1 illustrates a general description of a rigid body motion in an incompressible fluid, where the fluid and the rigid body occupy the physical domains Ω_f and Ω_s respectively. The fluid-solid interaction takes place at their common boundary $\partial\Omega_i = \Omega_f \cap \Omega_s$. The whole system is subjected to the gravitational acceleration \mathbf{g} .

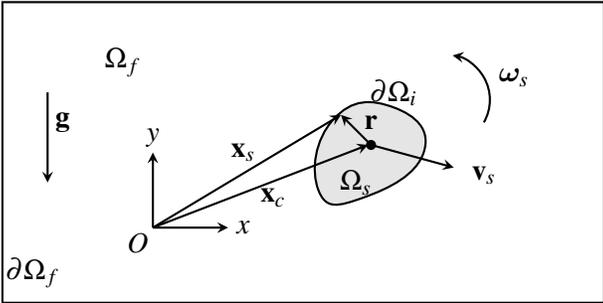


FIG. 1: Schematic representation of the fluid-solid interaction problem.

The fluid subproblem is usually described in Eulerian coordinates, which is governed by the

108 incompressible viscous Navier-Stokes equations

$$109 \quad \frac{\partial \mathbf{v}_f}{\partial t} + \nabla \cdot (\mathbf{v}_f \otimes \mathbf{v}_f) = \nabla \cdot \boldsymbol{\sigma}_f + \mathbf{g}, \quad (1a)$$

$$110 \quad \nabla \cdot \mathbf{v}_f = 0, \quad (1b)$$

112 where \mathbf{v}_f represents the fluid velocity vector and the fluid stress tensor $\boldsymbol{\sigma}_f$ (divided by ρ_f) is given
113 by

$$114 \quad \boldsymbol{\sigma}_f = -\frac{p}{\rho_f} \mathbf{I} + \nu (\nabla \mathbf{v}_f + (\nabla \mathbf{v}_f)^T), \quad (1c)$$

116 where p designates the fluid pressure, ρ_f the fluid density, ν the fluid kinematic viscosity. Approp-
117 priate initial and boundary conditions are assumed to the fluid Navier-Stokes equations to ensure
118 that the problem is well posed.

119 Contrarily the solid subproblem is often treated in a Lagrangian framework. The motion of a
120 rigid body is governed by the Newton-Euler equations, which can be expressed as

$$121 \quad m_s \frac{d\mathbf{v}_s}{dt} = \rho_f \int_{\partial\Omega_i} \boldsymbol{\sigma}_f \cdot \mathbf{n} ds + m_s \left(1 - \frac{\rho_f}{\rho_s}\right) \mathbf{g}, \quad (2a)$$

$$122 \quad I_s \frac{d\boldsymbol{\omega}_s}{dt} = \rho_f \int_{\partial\Omega_i} \mathbf{r} \times (\boldsymbol{\sigma}_f \cdot \mathbf{n}) ds, \quad (2b)$$

124 where m_s , ρ_s , I_s represent the solid mass, density and the moment of inertia respectively. \mathbf{v}_s , $\boldsymbol{\omega}_s$
125 designate the translational velocity and the angular velocity of the solid. $\mathbf{r} = \mathbf{x}_s - \mathbf{x}_c$ is the position
126 vector of any surface point with respect to the solid mass center \mathbf{x}_c . \mathbf{n} represents the outward-
127 pointing normal vector to the surface $\partial\Omega_i$. The position of the rigid body can be obtained by
128 integrating the kinematic equations

$$129 \quad \frac{d\mathbf{x}_c}{dt} = \mathbf{v}_s, \quad (2c)$$

$$130 \quad \frac{d\boldsymbol{\theta}_c}{dt} = \boldsymbol{\omega}_s, \quad (2d)$$

132 where $\boldsymbol{\theta}_c$ designates the rotation angle around the solid mass center.

133 On the fluid-solid interface $\partial\Omega_i$ the continuity of velocity

$$134 \quad \mathbf{v}_f = \mathbf{v}_s + \boldsymbol{\omega}_s \times \mathbf{r}, \quad (3)$$

135 needs to be satisfied in order to take the fluid-solid interaction into account.

136 The immersed boundary method approximates the above fluid-solid interaction problem by
137 replacing the solid domain with the surrounding fluid. To account for the presence of the immersed

138 solid, a boundary force \mathbf{f} is introduced and added into the fluid momentum equation. Therefore
 139 the fluid is simply simulated in a fixed domain $\bar{\Omega} = \Omega_f(t) \cup \Omega_s(t)$ irrespective to the movement
 140 of the immersed solid. The fluid-solid interaction problem in the immersed boundary formulation
 141 becomes

$$142 \quad \frac{\partial \mathbf{v}_f}{\partial t} + \nabla \cdot (\mathbf{v}_f \otimes \mathbf{v}_f) = -\frac{1}{\rho_f} \nabla p + \nu \nabla^2 \mathbf{v}_f + \mathbf{f} \quad \text{in } \bar{\Omega}, \quad (4a)$$

$$143 \quad \nabla \cdot \mathbf{v}_f = 0 \quad \text{in } \bar{\Omega}, \quad (4b)$$

$$144 \quad m_s \frac{d\mathbf{v}_s}{dt} = -\rho_f \int_{\Omega_s} \mathbf{f} dV + m_s \left(1 - \frac{\rho_f}{\rho_s}\right) \mathbf{g}, \quad (4c)$$

$$145 \quad I_s \frac{d\boldsymbol{\omega}_s}{dt} = -\rho_f \int_{\Omega_s} \mathbf{r} \times \mathbf{f} dV, \quad (4d)$$

$$146 \quad \mathbf{v}_f = \mathbf{v}_s + \boldsymbol{\omega}_s \times \mathbf{r} \quad \text{on } \partial\Omega_i, \quad (4e)$$

$$147 \quad \mathbf{f}(\mathbf{x}, t) = \int_{\Gamma_s} \mathbf{F}(s, t) \delta(\mathbf{x} - \mathbf{X}(s, t)) ds, \quad (4f)$$

149 where $\mathbf{F}(s, t)$ represents the boundary force defined on the Lagrangian position $\mathbf{X}(s, t)$ and $\mathbf{f}(\mathbf{x}, t)$
 150 on the Eulerian frame respectively. δ designates the Dirac delta function. Note that the effect of
 151 gravity in the fluid momentum equation is from now on incorporated into the pressure.

152 The fluid governing equations (4) are discretized in time as

$$153 \quad \frac{\mathbf{v}_f^{n+1} - \mathbf{v}_f^n}{\Delta t} + \frac{3}{2} \mathcal{N}(\mathbf{v}_f^n) - \frac{1}{2} \mathcal{N}(\mathbf{v}_f^{n-1}) = -\frac{1}{\rho_f} \mathcal{G} p^{n+1} + \frac{\nu}{2} \mathcal{L}(\mathbf{v}_f^{n+1} + \mathbf{v}_f^n) + \mathcal{I} \mathbf{F}^{n+1}, \quad (5a)$$

$$154 \quad \mathcal{D} \mathbf{v}_f^{n+1} = 0, \quad (5b)$$

$$155 \quad \mathcal{T} \mathbf{v}_f^{n+1} = \mathbf{v}_s^{n+1} + \boldsymbol{\omega}_s^{n+1} \times \mathbf{r}^{n+1} \quad \text{on } \partial\Omega_i^{n+1}, \quad (5c)$$

157 where second order semi-implicit time stepping schemes are utilized for the velocity, namely
 158 the explicit Adams-Bashforth scheme for the convection and the implicit Crank-Nicholson for the
 159 diffusion. Euler implicit scheme is applied to the two Lagrange multipliers, that is, the pressure and
 160 the boundary force. \mathcal{N} , \mathcal{L} , \mathcal{D} , \mathcal{G} represent the convection, Laplacian, divergence, and gradient
 161 operators respectively. Since the fluid grid is not coincident with the solid node, \mathcal{T} and \mathcal{I} are
 162 the interpolation and spreading operators that are responsible for the field exchange between two
 163 grids (see Figure 2). The discrete delta function is employed for the construction of \mathcal{T} and \mathcal{I}

$$164 \quad \delta_h(\mathbf{x} - \mathbf{X}) = \frac{1}{h^2} \phi\left(\frac{x-X}{h}\right) \phi\left(\frac{y-Y}{h}\right). \quad (6)$$

165 In this work, we select the three-point kernel proposed in Ref. 17 aimed for staggered grid, whose

166 one-dimensional form is

$$167 \quad \phi(r) = \begin{cases} \frac{1}{3} \left(1 + \sqrt{-3r^2 + 1} \right), & |r| < 0.5, \\ \frac{1}{6} \left(5 - 3|r| - \sqrt{-3(1 - |r|)^2 + 1} \right), & 0.5 \leq |r| < 1.5, \\ 0, & \text{otherwise.} \end{cases} \quad (7)$$

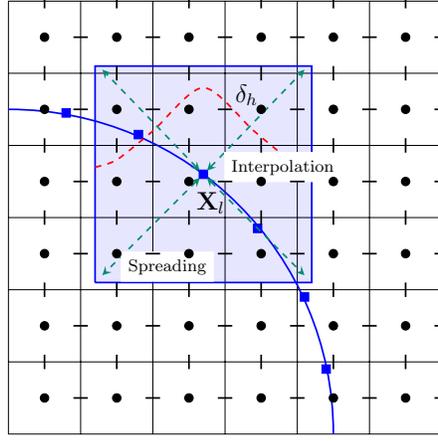


FIG. 2: Illustration of interpolation and spreading procedures with a discrete delta function.

168 With respect to the solid equation, we choose a simple Euler implicit scheme

$$169 \quad m_s \frac{\mathbf{v}_s^{n+1} - \mathbf{v}_s^n}{\Delta t} = -\rho_f \int_{\Omega_s} \mathbf{f}^{n+1} dV + m_s \left(1 - \frac{\rho_f}{\rho_s} \right) \mathbf{g}, \quad (8a)$$

$$170 \quad I_s \frac{\boldsymbol{\omega}_s^{n+1} - \boldsymbol{\omega}_s^n}{\Delta t} = -\rho_f \int_{\Omega_s} \mathbf{r} \times \mathbf{f}^{n+1} dV, \quad (8b)$$

$$171 \quad \frac{\mathbf{x}_c^{n+1} - \mathbf{x}_c^n}{\Delta t} = \mathbf{v}_s^{n+1}, \quad (8c)$$

$$172 \quad \frac{\boldsymbol{\theta}_c^{n+1} - \boldsymbol{\theta}_c^n}{\Delta t} = \boldsymbol{\omega}_s^{n+1}. \quad (8d)$$

173

174 The total amount of force and torque are not changed during the transfer between Lagrangian
 175 and Eulerian locations, owing to the partition of unity of the discrete delta function. The solid
 176 equations can be solved as follows

$$177 \quad m_s \frac{\mathbf{v}_s^{n+1} - \mathbf{v}_s^n}{\Delta t} = -\rho_f \sum_{l=1}^{n_b} \mathbf{F}^{n+1} \Delta V_l + m_s \left(1 - \frac{\rho_f}{\rho_s} \right) \mathbf{g}, \quad (9a)$$

$$178 \quad I_s \frac{\boldsymbol{\omega}_s^{n+1} - \boldsymbol{\omega}_s^n}{\Delta t} = -\rho_f \sum_{l=1}^{n_b} \mathbf{r} \times \mathbf{F}^{n+1} \Delta V_l, \quad (9b)$$

179

180 where ΔV_l represents the area/volume associated with the Lagrangian marker. Therefore, provided
 181 the boundary force, the solid equations can be easily solved. For the fluid part, we employ an
 182 operator splitting scheme as

183 (1) Prediction step for $\hat{\mathbf{v}}_f^{n+1}$

$$184 \quad \frac{\hat{\mathbf{v}}_f^{n+1} - \mathbf{v}_f^n}{\Delta t} + \frac{3}{2} \mathcal{N}(\mathbf{v}_f^n) - \frac{1}{2} \mathcal{N}(\mathbf{v}_f^{n-1}) = -\frac{1}{\rho_f} \mathcal{G} p^n + \frac{\mathbf{v}}{2} \mathcal{L}(\hat{\mathbf{v}}_f^{n+1} + \mathbf{v}_f^n). \quad (10)$$

185 (2) Immersed boundary forcing step for imposing the no-slip wall condition at the interface

$$186 \quad \frac{\tilde{\mathbf{v}}_f^{n+1} - \hat{\mathbf{v}}_f^{n+1}}{\Delta t} = \mathcal{S} \mathbf{F}^{n+1}, \quad (11a)$$

$$187 \quad \mathcal{T} \tilde{\mathbf{v}}_f^{n+1} = \mathbf{v}_s^{n+1} + \boldsymbol{\omega}_s^{n+1} \times \mathbf{r}^{n+1} \quad \text{on } \partial\Omega_i^{n+1}. \quad (11b)$$

189 Applying the no-slip constraint, we obtain

$$190 \quad \mathcal{T} \mathcal{S} \mathbf{F}^{n+1} = \frac{\mathbf{v}_s^{n+1} + \boldsymbol{\omega}_s^{n+1} \times \mathbf{r}^{n+1} - \mathcal{T} \hat{\mathbf{v}}_f^{n+1}}{\Delta t}, \quad (12a)$$

$$191 \quad \tilde{\mathbf{v}}_f^{n+1} = \hat{\mathbf{v}}_f^{n+1} + \Delta t \mathcal{S} \mathbf{F}^{n+1}. \quad (12b)$$

193 Given $\mathcal{M} = \mathcal{T} \mathcal{S}$ the moving force coefficient matrix, then we obtain a compact form of the
 194 moving force equation

$$195 \quad \mathcal{M} \mathbf{F}^{n+1} = \mathbf{F}^e, \quad (13)$$

196 with

$$197 \quad \mathbf{F}^e = \frac{\mathbf{v}_s^{n+1} + \boldsymbol{\omega}_s^{n+1} \times \mathbf{r}^{n+1} - \mathcal{T} \tilde{\mathbf{v}}_f^{n+1}}{\Delta t}, \quad (14)$$

198 where \mathbf{F}^e is the explicit force used by Kempe and Fröhlich⁴. Given the solid velocity and position
 199 at time level $n + 1$, the moving force equation can be solved with various linear system solvers. In
 200 the present work the moving force coefficient matrix \mathcal{M} is found to be symmetric and positive-
 201 definite. Hence the moving force equation can be converged to the machine precision very quickly
 202 by using the conjugate gradient method, even without preconditioning. Moreover, the size of the
 203 moving force coefficient matrix is proportional to the number of nodes on the immersed boundary,
 204 which is in general much smaller than the size of the fluid matrix. Since the moving force coef-
 205 ficient matrix does not involve the fluid matrix, its update is computationally inexpensive in the
 206 case of moving boundaries.

207 (3) Projection step for obtaining a divergence free velocity \mathbf{v}_f^{n+1}

$$208 \quad \frac{\mathbf{v}_f^{n+1} - \tilde{\mathbf{v}}_f^{n+1}}{\Delta t} = -\mathcal{L}\phi^{n+1}, \quad (15a)$$

$$209 \quad \mathcal{D}\mathbf{v}_f^{n+1} = 0. \quad (15b)$$

211 Applying the divergence operator to (15a) along with the divergence free condition (15b), the
212 projection step is actually performed as

$$213 \quad \mathcal{L}\phi^{n+1} = \frac{1}{\Delta t} \mathcal{D}\tilde{\mathbf{v}}_f^{n+1}, \quad (16a)$$

$$214 \quad \mathbf{v}_f^{n+1} = \tilde{\mathbf{v}}_f^{n+1} - \Delta t \mathcal{L}\phi^{n+1}. \quad (16b)$$

216 The final pressure is advanced by

$$217 \quad p^{n+1} = p^n + \phi^{n+1} - \frac{V}{2} \mathcal{D}\hat{\mathbf{v}}_f^{n+1}. \quad (17)$$

218 Now we will present a novel coupling approach for the interaction of the two subproblems.
219 Mathematically speaking, the FSI problem of equation (4) is accomplished by the Lagrange multi-
220 plier method to joint two subdomains on a common boundary. The unknowns of the entire system
221 are $(\mathbf{v}_f^{n+1}, p^{n+1})$ for the fluid, $(\mathbf{x}_s^{n+1}, \dot{\mathbf{x}}_s^{n+1})$ for the solid, and \mathbf{F}^{n+1} for the interface condition.
222 The conventional strongly coupled method iterates those variables at each time step in order to
223 match the interface condition. Even if it leads to accurate and stable solutions, solving implicit
224 coupling usually exhibits a prohibitive computational cost. This becomes more unaffordable in
225 ALE formulation, since the mesh and the associated matrices have to be updated regularly.

226 Fernández et al.¹⁸ proposed an efficient coupling method by taking advantage of the projection
227 method. The prediction (ALE-advection-viscous) step is ruled out from the FSI loop to reduce
228 computational cost, so that the mesh and the associated matrices are computed once at each time
229 step. Finally only the projection step is coupled implicitly to ensure stability. It has been shown
230 that this semi-implicit scheme is stable for a reasonable range of the discretization parameters,
231 compared to the explicit coupling approach.

232 We extend this idea to the moving immersed boundary method. However the projection step
233 usually is the most time-consuming part in the projection method. In spite of various methods
234 (e.g. Aitken relaxation) are available to accelerate the coupling procedure, the computational cost
235 still remains high. We also notice that in equation (9) the solid is coupled to the fluid by the
236 boundary force but not the pressure, and in equation (12) the boundary force is determined by

237 the solid velocity and position. Therefore, we can move out the time-consuming projection step
 238 from the FSI iteration, and replace it by the immersed boundary forcing step. Following Ref. 18,
 239 the prediction step is not included in the FSI iteration either, as it is performed on a stationary
 240 combined domain in MIBM and the boundary force is not incorporated in this step for the solid
 241 effects. The moving force equation can be considered as an implicit equation for the no-slip
 242 boundary condition for the fluid at the interface. Therefore the implicit coupling of the immersed
 243 boundary forcing step with the solid equations features a strongly coupled FSI method.

Algorithm 1: Novel implicit coupling scheme

```

1 Given:  $\mathbf{v}_f^n, p^n, \mathbf{x}_s^n, \dot{\mathbf{x}}_s^n$ ;
2 (Fluid) Predict the velocity  $\tilde{\mathbf{v}}_f^{n+1}$  using (10);
3 Initialize values:  $(\cdot)^{k=0,n+1} = (\cdot)^n$ , where  $(\cdot)$  includes  $\mathbf{x}_s, \dot{\mathbf{x}}_s, \mathbf{F}$ ;
4 for  $k = 0$  to  $k_{\max}$  do
5   (Fluid) Construct or update the interpolation operator matrix  $\mathcal{T}(\mathbf{x}_s^{k,n+1})$  and the moving force
   coefficient matrix  $\mathcal{M}(\mathbf{x}_s^{k,n+1})$ ;
6   (Fluid) Interpolate the fluid velocity  $\mathcal{T}(\mathbf{x}_s^{k,n+1})\tilde{\mathbf{v}}_f^{n+1}$ ;
7   (Interface) Solve the moving force equation (12a) for  $\mathbf{F}^{k+1,n+1}$  with  $\dot{\mathbf{x}}_s^{k,n+1}$ ;
8   (Solid) Compute the solid equations for  $(\mathbf{x}_s^{k+1,n+1}, \dot{\mathbf{x}}_s^{k+1,n+1})$  using (9);
9   if  $\|(\cdot)^{k+1,n+1} - (\cdot)^{k,n+1}\| / \|(\cdot)^{k+1,n+1}\| < \text{tolerance}$  then
10      $(\cdot)^{n+1} = (\cdot)^{k+1,n+1}$ ;
11     break;
12   else
13      $k = k + 1$ ;
14   end
15 end
16 (Fluid) Correct the fluid velocity to  $\hat{\mathbf{v}}_f^{n+1}$  with  $\mathbf{F}^{n+1}$  using (12b);
17 (Fluid) Solve the pressure Poisson equation and compute the final velocity  $\mathbf{v}_f^{n+1}$  and the pressure
 $p^{n+1}$  using (16a), (16b), (17).
```

244 Note that the moving force equation is non-linear, because the interpolation, spreading op-

245 erators and the boundary force are functions of solid position \mathbf{x}_s^{n+1} , namely $\mathcal{M}^{n+1}\mathbf{F}^{n+1} =$
 246 $\mathcal{T}(\mathbf{x}_s^{n+1})\mathcal{S}(\mathbf{x}_s^{n+1})\mathbf{F}(\mathbf{x}_s^{n+1})$. We can linearize this equation by treating the moving force coef-
 247 ficient matrix explicitly $\mathcal{M}(\mathbf{x}_s^n)$, but this will decrease the overall accuracy due to the time lag, as
 248 indicated in Ref. 19. In order to preserve a high accuracy, fully implicit implementation of the
 249 moving force equation is considered in the Algorithm 1.

250 III. NUMERICAL RESULTS

251 A. Freely falling cylinder in a confined channel

252 We first investigate the motion of a circular cylinder falling freely between two parallel walls
 253 in a quiescent fluid. The width and height of the computational domain are chosen to be 2 cm and
 254 6 cm. The circular cylinder with a diameter of $D = 0.25$ cm is released initially from the position
 255 (1 cm, 4 cm) and falls down because of gravity. The density of the cylinder and the surrounding
 256 fluid are $\rho_s = 1.25$ g/cm³ and $\rho_f = 1.0$ g/cm³ respectively. The fluid dynamic viscosity μ is set to
 257 0.1 g/(cm·s).

258 The calculations are carried out on two different uniform meshes, i.e. $h = 1/48$ cm and $h =$
 259 $1/96$ cm, to check the mesh sensitivity. The immersed cylinder surface is discretized evenly with
 260 a mesh size approximated to the size of surrounding fluid cells, due to the inherent limitation
 261 of the discrete delta function. Therefore the resulting Lagrangian marker numbers are 38 and 76
 262 respectively. The time step is chosen to be $\Delta t = 0.001$ s and the resulting maximum CFL number is
 263 reported to be 0.46. The vorticity around the falling cylinder is shown in Figure 3 at different times
 264 $t = 0.2$ s, 0.4 s, 0.6 s, 0.8 s. The cylinder quickly reaches a uniform falling velocity until it hits the
 265 bottom of the channel. We plot the flow quantities as a function of time in Figure 4, including the
 266 longitude position y_c of the cylinder center, the vertical velocity v_c , the Reynolds number Re and
 267 the translational kinetic energy E_T . Here Re and E_T are defined as $Re = (\rho_s D \sqrt{u_c^2 + v_c^2})/\mu$ and
 268 $E_T = 0.5m_s(u_c^2 + v_c^2)$ respectively, where u_c is the horizontal velocity component. For comparison,
 269 the results of Ref. 20 are included in Figure 4, taken from $h = 1/96$ cm. Good agreements have
 270 been obtained.

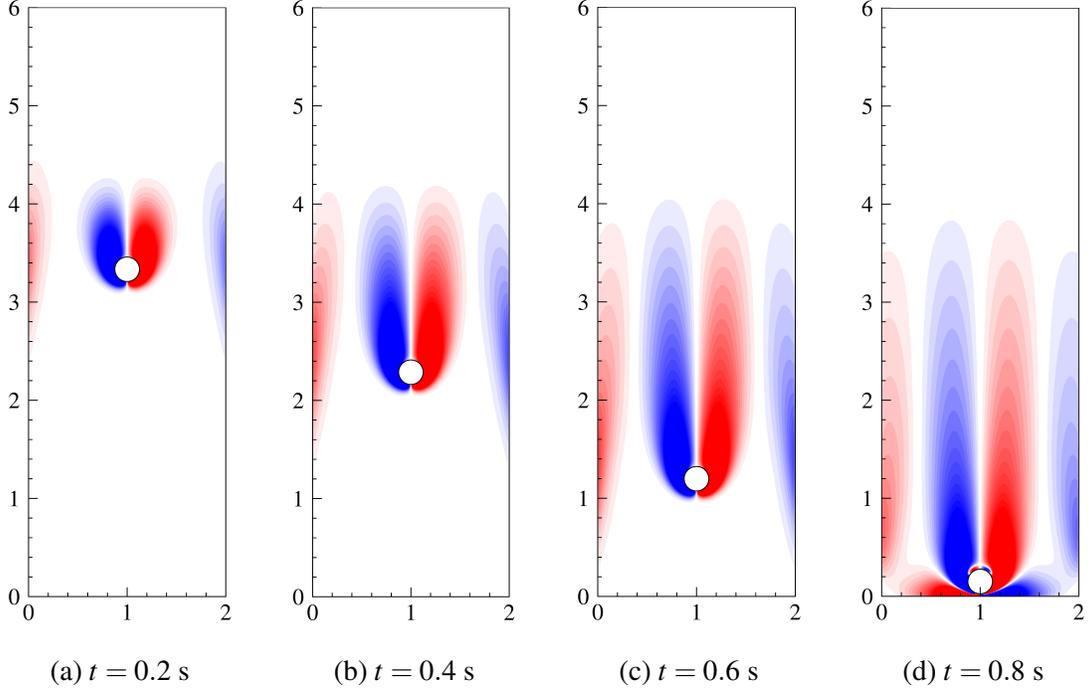


FIG. 3: Vorticity fields at different times $t = 0.2$ s, 0.4 s, 0.6 s, 0.8 s for the freely falling cylinder in a confined channel problem. The contour levels are set from -15 (blue) to 15 (red) with an increment of 1 .

271 B. Freely falling and rising cylinder in an open domain

272 Next we consider an object freely falling and rising in an open domain as another test. This
 273 phenomenon happens frequently in nature and a large amount of work can be found in the litera-
 274 ture. Here we compare to the numerical results of Namkoong et al.²¹ and Lacis et al.¹⁹, using the
 275 body-fitted ALE formulation and the immersed boundary projection method respectively.

276 Two density ratios are considered in this study, i.e. $\rho_s/\rho_f = 1.01$ for the falling case and
 277 $\rho_s/\rho_f = 0.99$ for the rising simulation. A large computational domain is taken as $[-5D, 5D] \times$
 278 $[-70D, 70D]$ with free-slip boundary conditions applied at all the exterior boundaries, where $D =$
 279 0.5 cm is the cylinder diameter. A uniform mesh is employed to cover the computational domain,
 280 and the mesh resolution is kept to $0.04D$ in order to compare with Lacis et al.¹⁹. Initially the
 281 cylinder is located at $\pm 65D$, depending on the situation ($65D$ for the falling case, $-65D$ for the
 282 rising case). The Reynolds number $Re = V_t D / \nu_f$ here is 156 , where V_t is the terminal velocity.
 283 Note that the Reynolds number depends on the Galileo number $G = (|\rho_s/\rho_f - 1| g D^3)^{1/2} / \nu_f$ (the
 284 gravity force divided by the viscous force, $G = 138$) and the density ratio ρ_s/ρ_f .

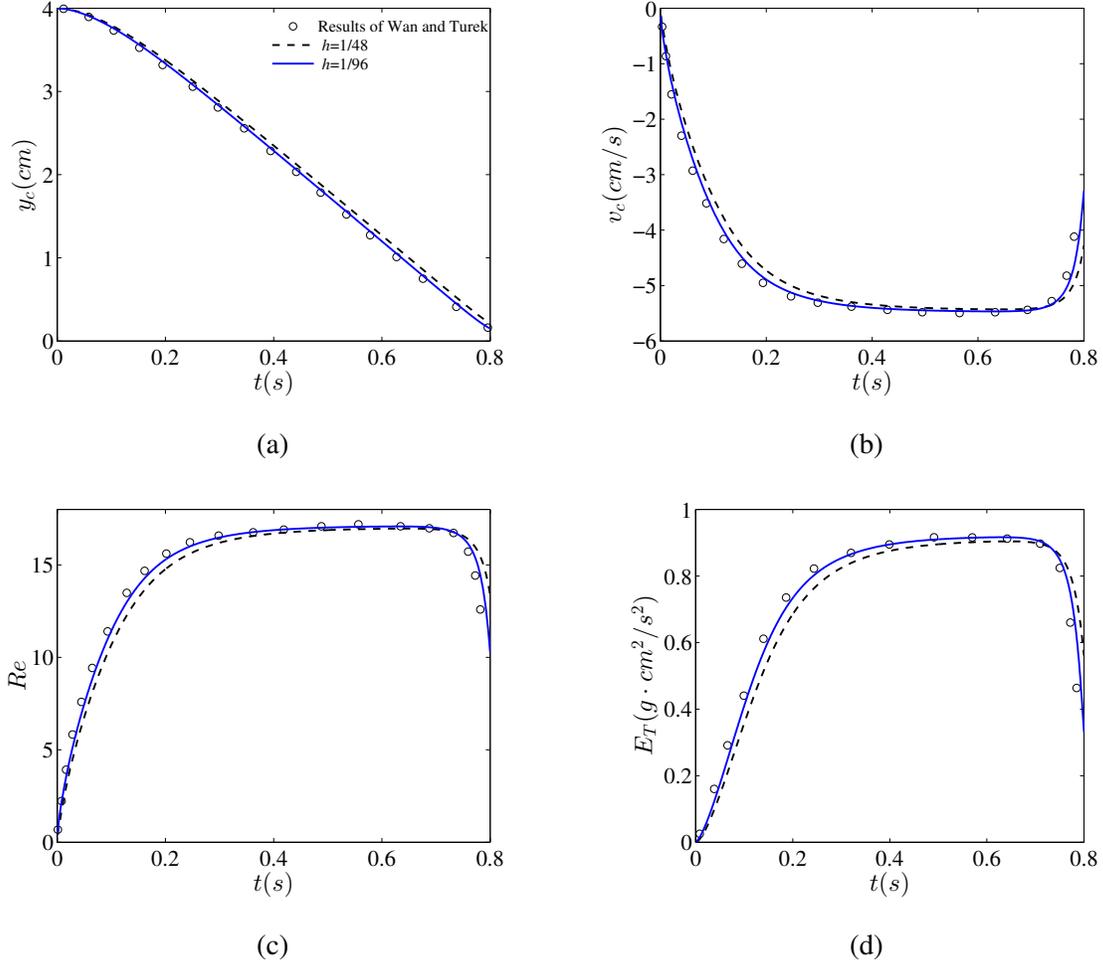


FIG. 4: Time evolution of longitude position y_c , velocity v_c of the cylinder center, the Reynolds number Re and the translational kinetic energy E_T for the freely falling cylinder in a confined channel problem. “○”, result of Ref. 20; “—”, present result with $h = 1/96$ cm; “- - -”, present result with $h = 1/48$ cm.

285 The instantaneous vorticity fields are presented in Figure 5 for the falling cylinder case. Ini-
 286 tially symmetric vortex pair forms behind the cylinder in the beginning of falling. After that the
 287 numerical error accumulates and breaks the symmetry. At around $tV_t/D = 40$, the flow becomes
 288 unsteady and periodic vortex shedding occurs. The time histories of the velocity of the cylinder are
 289 plotted in Figure 6. Table I shows the Strouhal number $St = fD/V_t$ (f is the shedding frequency)
 290 and the coefficients of drag and lift. Present results are compared to those of Namkoong et al.²¹
 291 and Lacis et al.¹⁹. Good agreements have been obtained.

292 To illustrate the efficiency of the proposed coupling method, different strong FSI coupling

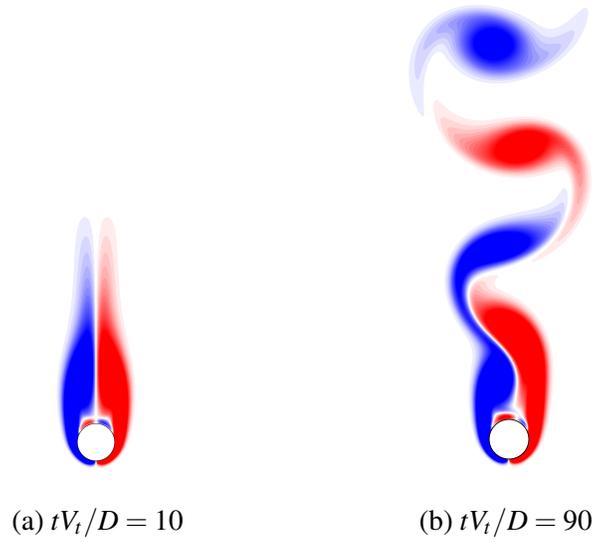


FIG. 5: Snapshots of vorticity fields for a freely falling cylinder in an open domain . The contour level is set from -6 (blue) to 6 (red) with an increment of 0.4.

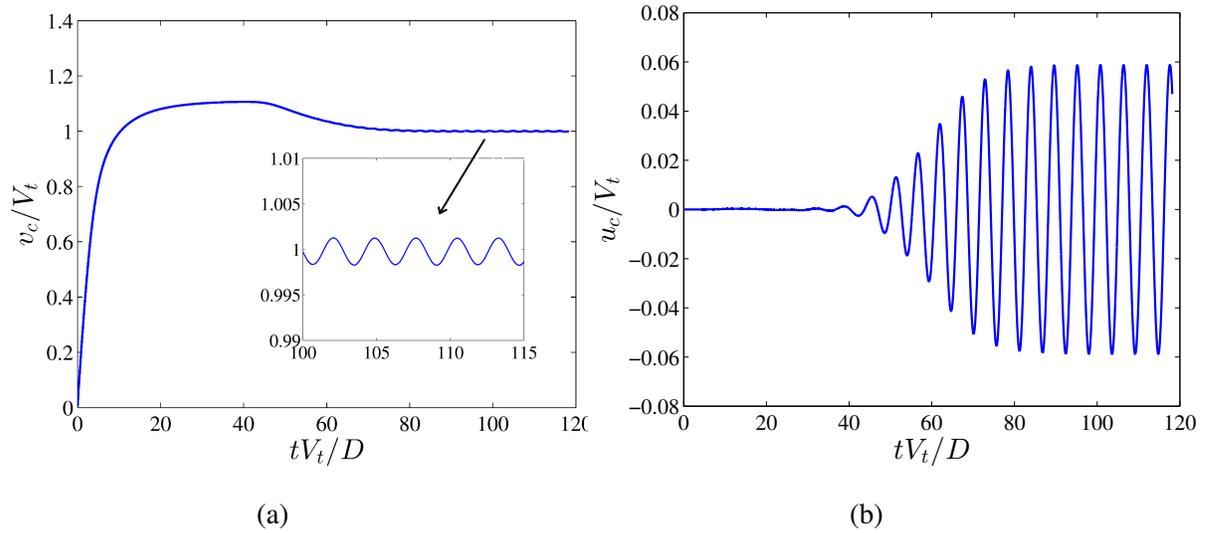


FIG. 6: Time histories of the vertical and horizontal velocity for the freely rising cylinder $\rho_s/\rho_f = 0.99$.

293 methods are compared in Figure 7. The time consumption of the traditional strong FSI coupling
 294 method, where the pressure Poisson equation is involved in the sub-iteration, almost scales linearly
 295 with the number of sub-iterations, which is due to the fact that the solution of the pressure Poisson
 296 equation is the most dominating part. It can be seen that the proposed method reduces up to 75%

ρ_s/ρ_f	Methods	C_D	$ C_L _{\max}$	St
1.01	Present	1.35	0.10	0.189
	Lacis et al. ¹⁹	1.29	0.14	0.17185
	Namkoong et al. ²¹	1.23	0.15	0.1684
0.99	Present	1.35	0.10	0.189
	Lacis et al. ¹⁹	1.29	0.14	0.17188
	Namkoong et al. ²¹	-	-	0.1687

TABLE I: The drag, lift coefficients and the Strouhal number for the freely falling and rising circular cylinder in an open domain.

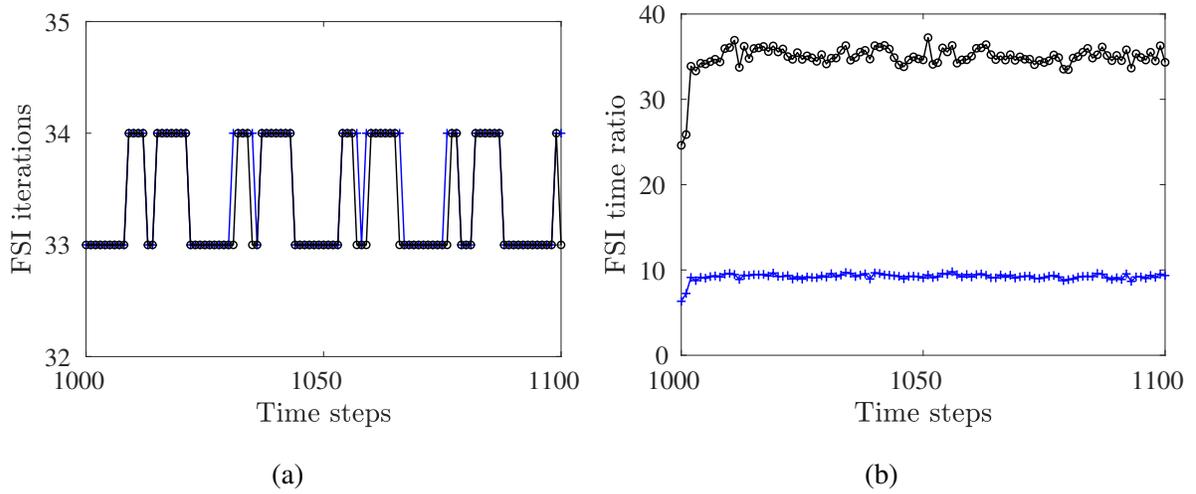


FIG. 7: Comparison of different strong FSI coupling strategies. (a) The number of sub-iterations in the strong FSI coupling; (b) Time consumption ratio of various strong coupling methods compared to the weak coupling method. “o”, strong coupling with pressure Poisson equation in the sub-iteration; “+”, the proposed strong coupling.

297 of the FSI coupling time within nearly the same number of sub-iteration.

298 C. Elliptical particle sedimentation in a confined channel

299 In this example, we consider the sedimentation of an elliptical particle in a narrow channel, to
300 demonstrate the ability of current FSI algorithm for non-circular object. This example was stud-

301 ied previously by Xia et al.²² for the boundary effects on the sedimentation mode. Five distinct
 302 modes of sedimentation have been found ranging from oscillating, tumbling along the wall, ver-
 303 tical sedimentation, horizontal sedimentation to an inclined mode. In their work, a multi-block
 304 lattice Boltzmann method is used and compared to the traditional ALE formulation.

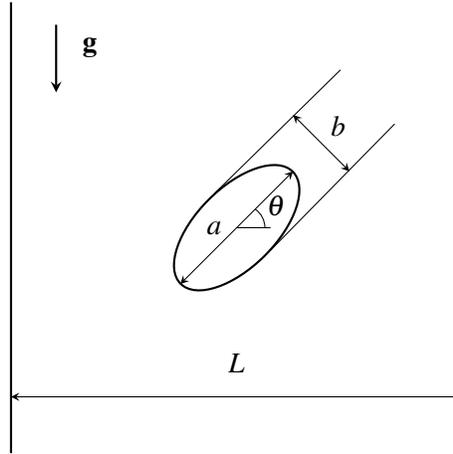


FIG. 8: Computational domain of the elliptical particle sedimentation problem.

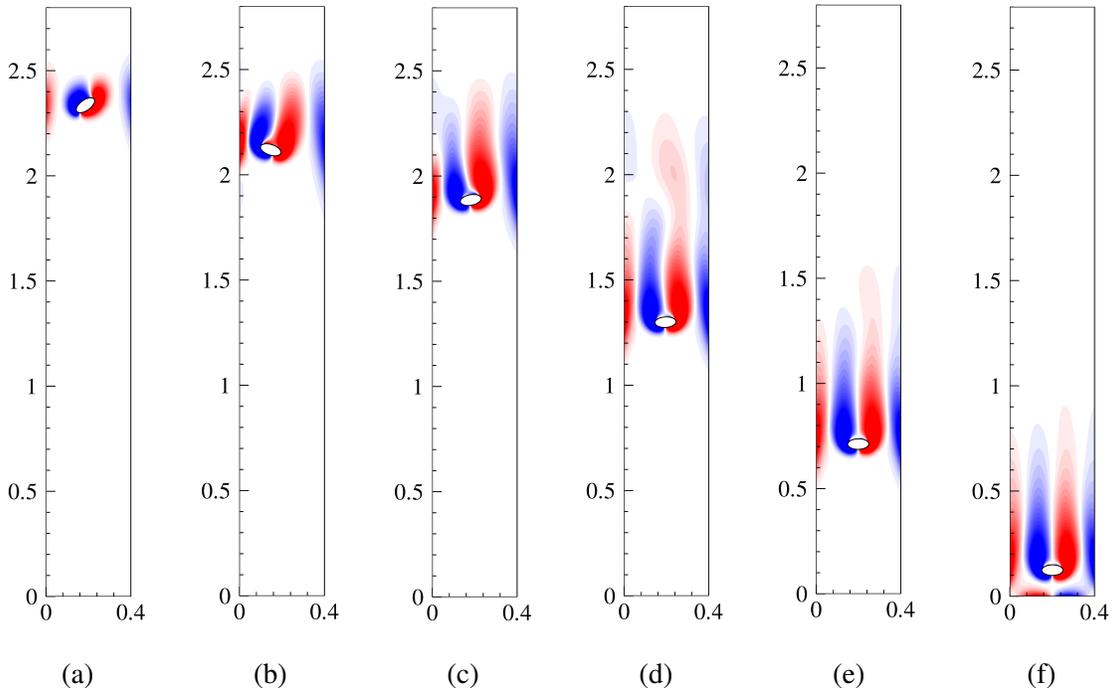


FIG. 9: Vorticity fields at different times: (a) $t = 0.1$ s ; (b) $t = 0.3$ s ; (c) $t = 0.5$ s ; (d) $t = 1.0$ s ;
 (e) $t = 1.5$ s ; (f) $t = 2.0$ s. The contour levels are set from -15 to 15.

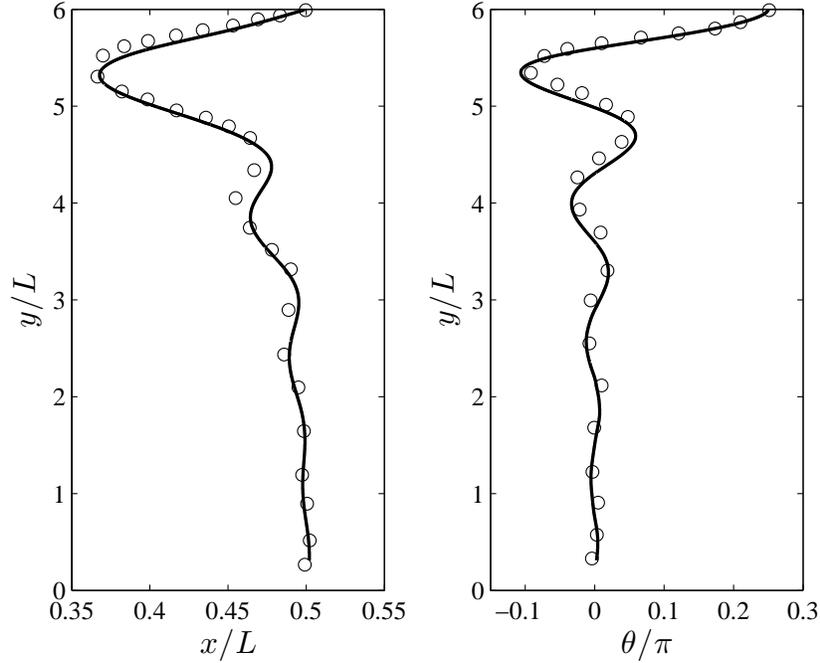


FIG. 10: Particle trajectory and orientation of the elliptical particle. “—”, present results; “o”, results of Xia et al.²².

305 To compare with Xia et al.²², the computational domain is selected to be $[0, L] \times [0, 7L]$ with
 306 $L = 0.4$ cm, as shown in Figure 8. The aspect ratio of the ellipse is $\alpha = a/b = 2$, where a and b are
 307 the major and minor axes respectively. The blockage ratio is defined as $\beta = L/a = 4$. The density
 308 ratio is $\rho_s/\rho_f = 1.1$. The kinematic viscosity of fluid is set to $\nu = 0.01$ cm²/s. The particle starts
 309 falling in a quiescent fluid from the centroid at $(0.5L, 6L)$ with an initial angle of $\pi/4$ to break the
 310 symmetry.

311 No-slip boundary conditions are applied at four boundaries. A uniform mesh is employed with
 312 a grid resolution of 0.0027 cm. The time step is chosen such that the CFL condition is satisfied.
 313 Figure 9 shows the vorticity fields at different times at $t = 0.1$ s, 0.3 s, 0.5 s, 1.0 s, 1.5 s, 2.0
 314 s. The trajectory and orientations are compared to the results of Xia et al.²² in Figure 10. Good
 315 agreements have been obtained.

316 D. Flow around a rotating NACA0012 airfoil

317 The rigid objects simulated in this section so far have been the circular or elliptical particles.
 318 In this example, we consider the incompressible viscous flow over a NACA0012 airfoil to demon-

319 strate the capacity of current MIBM for handling sharp geometries. The shape of the NACA0012
 320 airfoil is given by

$$321 \quad Y = \pm 0.6 \cdot (0.2969\sqrt{X} - 0.1260X - 0.3516X^2 + 0.2843X^3 - 0.1015X^4), \quad (18)$$

322 where $X \in (0, 1.009)$ cm. The characteristic length, i.e. the airfoil length, is 1.009 cm. Following
 323 [Glowinski et al.²³](#) and [Wan and Turek²⁰](#), we select a computational domain of $[-4 \text{ cm}, 16 \text{ cm}] \times$
 324 $[-2 \text{ cm}, 2 \text{ cm}]$ with the airfoil centered at $(0.42 \text{ cm}, 0)$, as shown in Figure 11.

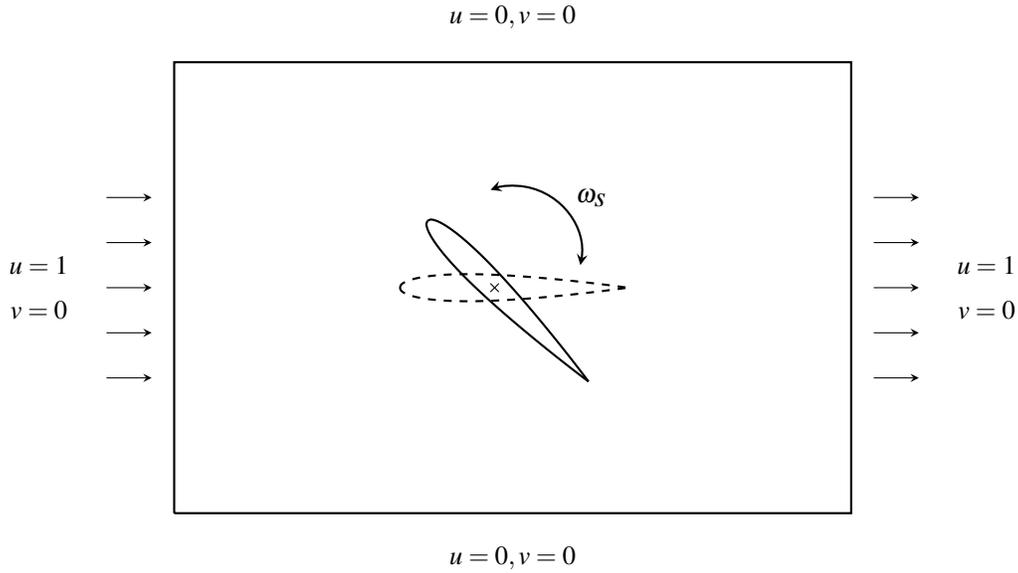


FIG. 11: Computational domain of the flow past a rotating NACA0012 airfoil. The dashed lines represent the initial position of the airfoil.

325 The airfoil is fixed at its mass center and is free to rotate due to hydrodynamic forces. The
 326 density of fluid is taken as $\rho_f = 1.0 \text{ g/cm}^3$ and the density of solid is $\rho_s = 1.1 \text{ g/cm}^3$ in this
 327 simulation. The viscosity of the fluid is $\nu_f = 0.01 \text{ cm}^2/\text{s}$. Initial angular velocity and incident
 328 angle of the airfoil are set to zero. The boundary conditions of flow are given as $\mathbf{u} = (0, 0)$ at
 329 $y = -2 \text{ cm}, 2 \text{ cm}$ and $\mathbf{u} = (1, 0) \text{ cm/s}$ at $x = -4 \text{ cm}, 16 \text{ cm}$. Those boundary conditions are used
 330 in [Glowinski et al.²³](#) and are adopted here in order to compare the results of two methods. The
 331 Reynolds number is around 101 based on airfoil length and the maximum inflow speed.

332 This flow is quite challenging as the leading edge of the airfoil has very small radius of curva-
 333 ture. To resolve the flow near the leading edge, a good resolution of the Cartesian mesh is required.
 334 Two sets of grids are chosen here to test the grid sensitivity, namely $h = 1/96 \text{ cm}$ and $h = 1/64$

335 cm. The same time step is used in both cases ($\Delta t = 0.002$ s). The resulting CFL numbers are 0.40
336 and 0.25 respectively.

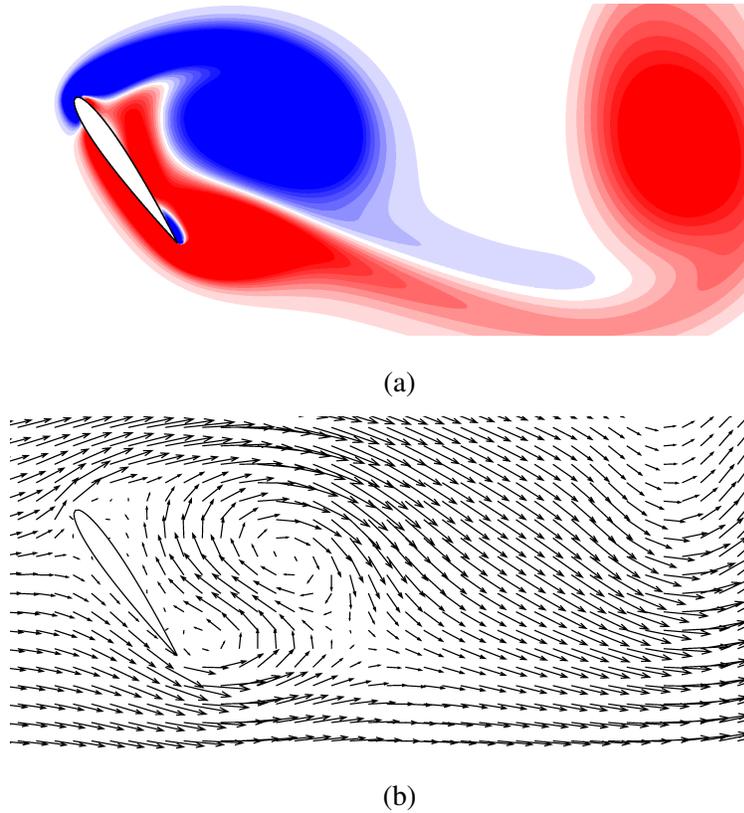
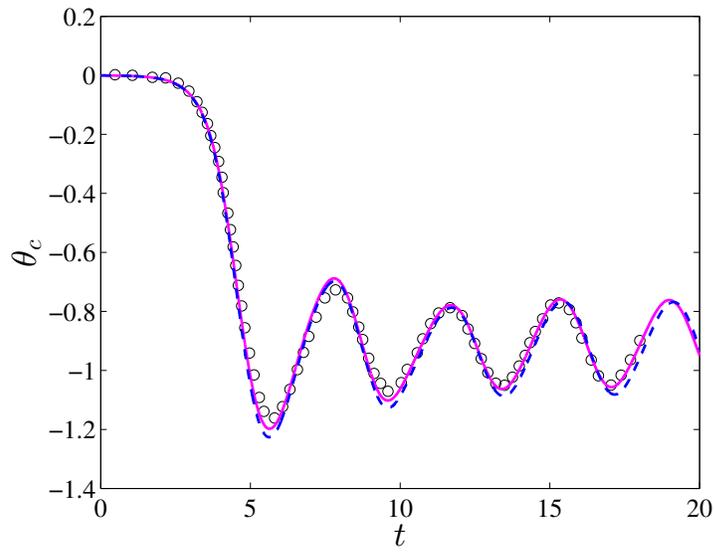
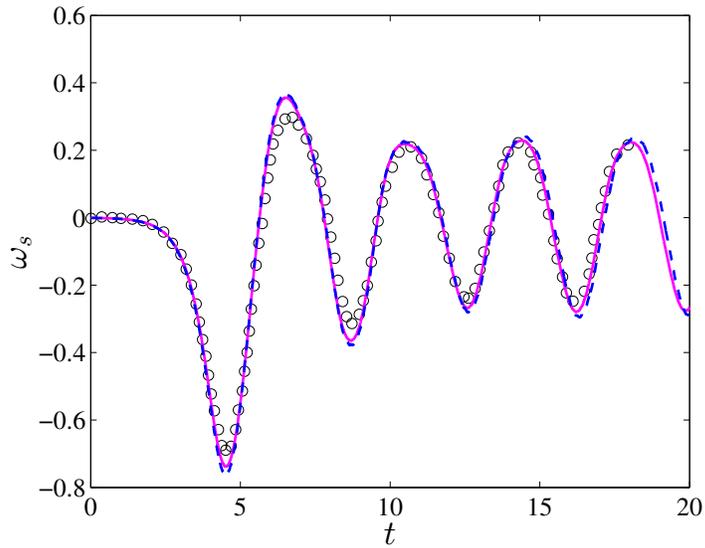


FIG. 12: Instantaneous vorticity (a) and velocity (b) of the flow over a rotating NACA0012 airfoil.

337 The flow fields are shown in Figure 12. The airfoil keeps a stable position with its broadside
338 perpendicular to the in-flow direction in the beginning and finally reaches a periodic motion of
339 oscillation. The time histories of the rotational angle and the angular velocity are plotted in Figure
340 13. The results are pretty smooth. For comparison, we also include the results of [Glowinski](#)
341 [et al.²³](#) in Figure 13 obtained by the distributed Lagrange multiplier method. Presents results
342 match well those of [Glowinski et al.²³](#). We observe that the direction of oscillation depends on
343 the numerical errors. The oscillating direction changes over different calculation parameters. An
344 opposite oscillating case can be found in [Wan and Turek²⁰](#). Present direction of oscillation is in
345 accord with [Glowinski et al.²³](#).



(a)



(b)

FIG. 13: Time histories of the angle (a) and the angular velocity (b) of the rotating NACA0012 airfoil. $h = 1/96$ cm, solid line; $h = 1/64$ cm, dashed line. The measures are in rad and rad/s respectively. The results of [Glowinski et al.²³](#) are marked with circles.

346 IV. CONCLUSIONS

347 A revised algorithm for the fluid-solid interaction was presented in this paper based on the
348 moving immersed boundary method. Operator splitting is successively employed to achieve com-
349 putational efficiency and modularity. First the pressure-velocity coupling is handled by a second
350 order fractional step method, breaking the Navier-Stokes equations into a Helmholtz-type equation
351 for the velocity and a Poisson equation for the pressure; Followed by the immersed boundary force
352 splitting, a moving force equation is derived to account for the solid effect; At last a partitioned
353 approach separates the fluid field from the solid field. By doing this, all the variables are decoupled
354 and computed effectively with their own favored solvers. To achieve a more stable FSI coupling,
355 a strongly coupled scheme is employed through a fixed point iteration. For better performance,
356 we have removed the time-consuming pressure Poisson solver out from the FSI iteration and the
357 computational time has been considerably reduced. Various cases have been tested, ranging from
358 circular to non-circular objects with large displacements and rotations. The numerical results are
359 compared to the benchmark, which demonstrates a good accuracy of the proposed method.

360 ACKNOWLEDGMENTS

361 This work was part of the PhD study of the first author, where the financial support of the China
362 Scholarship Council is acknowledged. The computation was performed on the platform PILCAM2
363 at the Université de Technologie de Compiègne and the HPC at the Université de Strasbourg.

364 REFERENCES

- 365 ¹T. Deloze, Y. Hoarau, and J. Dusek, “Transition scenario of a sphere freely falling in a vertical
366 tube,” *J. Fluid Mech.* **711**, 40–60 (2012).
- 367 ²C. S. Peskin, “Flow patterns around heart valves: A numerical method,” *J. Comput. Phys.* **10**,
368 252–271 (1972).
- 369 ³M. Uhlmann, “An immersed boundary method with direct forcing for the simulation of particu-
370 late flows,” *J. Comput. Phys.* **209**, 448–476 (2005).
- 371 ⁴T. Kempe and J. Fröhlich, “An improved immersed boundary method with direct forcing for the
372 simulation of particle laden flows,” *J. Comput. Phys.* **231**, 3663–3684 (2012).

- 373 ⁵S.-G. Cai, *Computational fluid-structure interaction with the moving immersed boundary*
374 *method*, Ph.D. thesis, Université de Technologie de Compiègne (2016).
- 375 ⁶S.-G. Cai, A. Ouahsine, J. Favier, and Y. Hoarau, “Improved implicit immersed boundary
376 method via operator splitting,” (Springer Verlag, 2016) Chap. 3, pp. 49–66.
- 377 ⁷S.-G. Cai, A. Ouahsine, J. Favier, and Y. Hoarau, “Implicit immersed boundary method for
378 fluid-structure interaction,” *La Houille Blanche* **1**, 33–36 (2017).
- 379 ⁸S.-G. Cai, A. Ouahsine, J. Favier, and Y. Hoarau, “Moving immersed boundary method,” *Int. J.*
380 *Numer. Meth. Fluids* **85**, 288–323 (2017).
- 381 ⁹S.-G. Cai, J. Degryny, J.-F. Boussuge, and P. Sagaut, “Coupling of turbulence wall models and
382 immersed boundaries on Cartesian grids,” *J. Comput. Phys.* **429**, 109995 (2021).
- 383 ¹⁰J. Degryny, S.-G. Cai, J.-F. Boussuge, and P. Sagaut, “Improved wall model treatment for
384 aerodynamic flows in LBM,” *Comput. Fluids*, 105041 (2021).
- 385 ¹¹Y.-H. Tseng and J. Ferziger, “A ghost-cell immersed boundary method for flow in complex
386 geometry,” *J. Comput. Phys.* **192**, 593–623 (2003).
- 387 ¹²W.-P. Breugem, “A second-order accurate immersed boundary method for fully resolved simu-
388 lations of particle-laden flows,” *J. Comput. Phys.* **231**, 4469–4498 (2012).
- 389 ¹³K. Taira and T. Colonius, “The immersed boundary method: A projection approach,” *J. Comput.*
390 *Phys.* **225**, 2118–2137 (2007).
- 391 ¹⁴P. Causin, J.-F. Gerbeau, and F. Nobile, “Added-mass effect in the design of partitioned al-
392 gorithms for fluid–structure problems,” *Comput. Methods Appl. Mech. Engrg.* **194**, 4506–4527
393 (2005).
- 394 ¹⁵C. Förster, W. A. Wall, and E. Ramm, “Artificial added mass instabilities in sequential staggered
395 coupling of nonlinear structures and incompressible viscous flows,” *Comput. Methods Appl.*
396 *Mech. Engrg.* **196**, 1278 – 1293 (2007).
- 397 ¹⁶C. Kassiotis, A. Ibrahimbegovic, R. Niekamp, and H. G. Matthies, “Nonlinear fluid–structure
398 interaction problem. Part I: implicit partitioned algorithm, nonlinear stability proof and valida-
399 tion examples,” *Comput. Mech.* **47**, 305–323 (2011).
- 400 ¹⁷A. M. Roma, C. S. Peskin, and M. J. Berger, “An adaptive version of the immersed boundary
401 method,” *J. Comput. Phys.* **153**, 509–534 (1999).
- 402 ¹⁸M. A. Fernández, J.-F. Gerbeau, and C. Grandmont, “A projection semi-implicit scheme for
403 the coupling of an elastic structure with an incompressible fluid,” *Int. J. Numer. Meth. Eng.* **69**,
404 794–821 (2007).

- 405 ¹⁹U. Lacia, K. Taira, and S. Bagheri, “A stable fluid–structure-interaction solver for low-density
406 rigid bodies using the immersed boundary projection method,” *J. Comput. Phys.* **305**, 300–318
407 (2016).
- 408 ²⁰D. Wan and S. Turek, “Direct numerical simulation of particulate flow via multigrid fem tech-
409 nique and the fictitious boundary method,” *Int. J. Numer. Meth. Fluids* **51**, 531–566 (2006).
- 410 ²¹K. Namkoong, J. Yoo, and H. Choi, “Numerical analysis of two-dimensional motion of a freely
411 falling circular cylinder in an infinite fluid,” *J. Fluid Mech.* **604**, 33–53 (2008).
- 412 ²²Z. Xia, K. Connington, S. Rapaka, P. Yue, J. Feng, and S. Chen, “Flow patterns in the sedimen-
413 tation of an elliptical particle,” *J. Fluid Mech.* **625**, 249–272 (2009).
- 414 ²³R. Glowinski, T. Pan, T. Hesla, D. Joseph, and J. Periaux, “A fictitious domain approach to the
415 direct numerical simulation of incompressible viscous flow past moving rigid bodies: application
416 to particulate flow,” *J. Comput. Phys.* **169**, 363–426 (2001).