An immersed boundary finite element method for flow around moving rigid bodies

Ilinca, Florin; Hétu, Jean-François

Publisher's version / Version de l'éditeur:

19th Annual Conference of the CFD Society of Canada [Proceedings], 2011-04-28

An immersed boundary finite element method for flow around moving rigid bodies

Florin Ilinca[1] and Jean-François Hétu[1]

National Research Council, 75 de Mortagne, Boucherville, QC, J4B 6Y4, Canada

Email: *florin.ilinca@cnrc-nrc.gc.ca*

ABSTRACT

This contribution presents applications of a recently proposed immersed boundary method to the solution of the flow around moving rigid bodies. The use of body-conforming meshes to solve the flow around rigid objects may involve extensive meshing work that has to be repeated each time a change in the position of immersed solids is needed. Mesh generation and solution interpolation between successive grids may be costly and introduce errors if the geometry changes significantly during the course of the computation. These drawbacks are avoided when the solution algorithm can tackle grids that do not fit the shape of immersed objects. We present here an extension of a recently developed Immersed Boundary (IB) finite element method to the computation of interaction forces between the fluid and immersed solid bodies. Solid objects immersed into the fluid are considered rigid and the fixed mesh covers both the fluid and solid regions. The boundary of immersed objects is defined using a time dependent level-set function and the proposed procedure is able to impose accurately the boundary conditions on the immersed solid surfaces and to recover the interacting forces. This is done by enriching the finite element discretization of interface elements with additional degrees of freedom which are latter eliminated at element level. The forces acting on the solid surfaces are then computed from the enriched finite element solution and if needed the solid movement is determined form the rigid solid momentum equation. Solutions are shown for flows around moving solid bodies.

1 INTRODUCTION

Most CFD and fluid-structure interaction solvers are based on body-conforming (BC) grids (i.e. the external boundary and surfaces of immersed bodies are represented by the mesh faces), but there is an increased interest in solution algorithms for non body-conforming grids. For these methods the spatial discretization is done over a single domain containing both fluid and solid regions and where mesh points are not necessarily located on the fluid-solid interface [1, 2, 3]. For simplicity, we will use in the present work the immersed boundary (IB) term to identify a non body-fitted method. IB methods have the main advantage of avoiding costly and sometimes very difficult meshing work on body-fitted geometries. Generally a regular parallelepiped is meshed with an uniform grid. The IB method results also in important algorithmic simplifications when immersed moving bodies are considered. One important drawback of such a method is that the boundary which has an important influence on the solution especially for fluid-solid interaction is also a place where distorted elements may be found once regular mesh elements are cut by the solid boundary. The imposition of the boundary conditions on the immersed boundary is also a point of concern.

In most IB methods, boundary conditions on immersed surfaces are handled either accurately by using dynamic data structures to add/remove grid points as needed, or in an approximate way by imposing the boundary conditions to the grid point closest to the surface or through least-squares. Our recently proposed approach [4, 5] achieves the level of accuracy of cut cell dynamic node addition techniques with none of their drawbacks (increased CPU time and costly dynamic data structures). The discretization of elements cut by the fluid/solid interface is enriched by the addition of degrees of freedom associated to interface nodes which are latter eliminated at element level. The proposed approach is verified on simple cases for which solutions on BC grids can be obtained and is applied to more complex flow problems in presence of moving solid bodies.

2 THE MODEL PROBLEM

We consider the transient incompressible fluid flow problem on a bounded computational domain Ω formed by the fluid region $\Omega_f(t)$ and the solid volume $\Omega_s(t)$. The fluid and solid volumes are time dependent but the total volume Ω formed by their reunion is not. The immersed interface $\Gamma_i(t) = \partial\Omega_f(t) \cap \partial\Omega_s(t)$ represents a boundary for the fluid flow.

2.1 Finite element solution

The flow is described by the incompressible Navier-Stokes equations:

$$\rho\left(\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u}\cdot\nabla\mathbf{u}\right) = -\nabla p + \nabla\cdot\left[\mu\left(\nabla\mathbf{u} + (\nabla\mathbf{u})^T\right)\right] + \mathbf{f} \quad (1)$$

$$\nabla\cdot\mathbf{u} = 0, \quad (2)$$

where ρ is the density, $\mathbf{u}$ the velocity vector, p the pressure, μ the viscosity, and $\mathbf{f}$ a volumetric force vector.

The interface Γ_i, between the fluid and solid regions, is specified using a level-set function ψ defined as a signed distance function from the immersed interface:

$$\psi(\mathbf{x},t) = \begin{cases} d(\mathbf{x},\mathbf{x}_i(t)), & \mathbf{x} \text{ in the fluid,} \\ 0, & \mathbf{x} \text{ on the interface,} \\ -d(\mathbf{x},\mathbf{x}_i(t)), & \mathbf{x} \text{ in the solid,} \end{cases} \quad (3)$$

The initial and boundary conditions associated to equations (1) and (2) are

$$\mathbf{u} = \mathbf{U}_0(\mathbf{x}), \text{ for } t = t_0, \quad (4)$$

$$\mathbf{u} = \mathbf{U}_D(\mathbf{x},t), \text{ for } \mathbf{x} \in \Gamma_D(t), \quad (5)$$

$$\mu\left(\nabla\mathbf{u} + \nabla\mathbf{u}^T\right)\cdot\hat{\mathbf{n}} - p\hat{\mathbf{n}} = \mathbf{t}(\mathbf{x},t), \text{ for } \mathbf{x} \in \Gamma_t(t), \quad (6)$$

where Γ_D is the portion of the fluid boundary $\partial\Omega_f$ where Dirichlet conditions are imposed, and $\mathbf{t}$ is the traction imposed on the remaining fluid boundary $\Gamma_t = \partial\Omega_f \backslash \Gamma_D$. Dirichlet boundary conditions are imposed at the interface between fluid and solid regions, i.e. $\Gamma_i \subset \Gamma_D$. Because Γ_i is not represented by the finite element discretization, a special procedure is used to enforce velocity boundary conditions on this surface. This approach will be discussed in Section 2.2.

Equations are discretized using the GLS method with linear continuous shape functions for both velocity and pressure [5]. Time derivatives are computed using an implicit Euler scheme. The nonlinear equations are solved with a few Picard steps followed by Newton-Raphson iterations. The resulting linear systems are solved using the bi-conjugate gradient stabilized (Bi-CGSTAB) iterative method with an ILU preconditioner.

2.2 The IB method

The algorithm used to treat the immersed boundary surface is the same as introduced by Ilinca and Hétu for the static fluid/solid interfaces [4] and for moving interfaces [5].

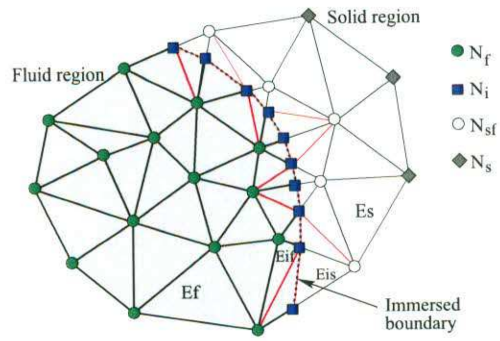
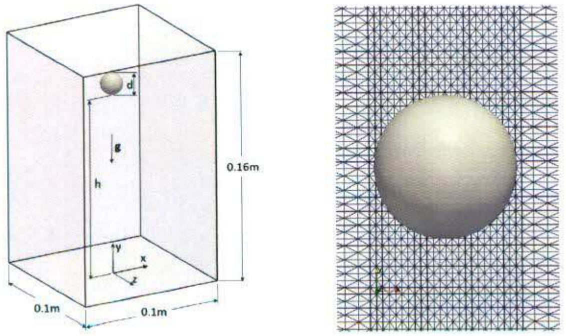


Figure 1: Decomposition of interface elements.

The mesh is intersected by the interface at the current time step t_n at points located along element edges (N_i in Figure 1) and we consider those points as additional degrees of freedom in the finite element formulation. While elements cut by the immersed boundary have nodes in both fluid and solid regions, they can be decomposed into sub-elements which are either entirely in the fluid region (E_{if} in Figure 1) or in the solid region (E_{is} in Figure 1).

When solving the fluid flow we consider that the solid embedded in the mesh has a prescribed velocity $\mathbf{u_s(t)}$. This velocity can also be computed from the forces applied on the solid body. The solid velocity is therefore imposed on the solid nodes including the additional interface nodes. By doing so no additional degrees of freedom need to be included for the interface nodes. Only the right hand side of equations associated to fluid nodes connected to them will change and by these means take into account the location of the interface.

The pressure degrees of freedom are associated to the continuity equations. In order to enforce mass conservation in the entire fluid region, the continuity equations are solved on all fluid elements, including the fluid sub-elements at the interface. The continuity equations are not solved in the solid elements and the pressure is set to a constant (say zero) on solid nodes. The pressure discretization is considered discontinuous between interface sub-elements and the additional pressure degrees of freedom corresponding to interface nodes are eliminated by static condensation. For more details the reader should consult [4, 5].



(a) Computational domain (b) Detail of the mesh

Figure 2: Computational domain and mesh.

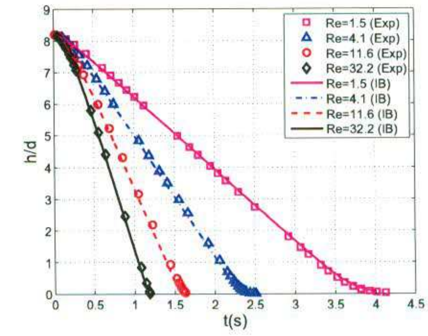


Figure 3: Evolution with time of the sphere position.

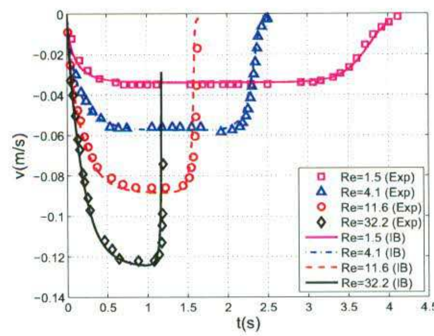


Figure 4: Evolution with time of the sphere velocity.

2.3 Computation of fluid/solid interaction forces

In this work the fluid and solid equations are solved separately. The fluid flow uses the solid velocity as boundary conditions and then provides the forces acting on the solid to serve in the computation of the solid body movement. One main aspect in the simulation of fluid/solid interaction is therefore the computation of the interaction forces between the fluid and solid. The force acting on the fluid at the solid interface is computed from:

$$\mathbf{F}_F(t) = \int_{\Gamma_i} \left[\mu\left(\nabla\mathbf{u} + \nabla\mathbf{u}^T\right)\cdot\hat{\mathbf{n}} - p\hat{\mathbf{n}}\right] \mathrm{d}\Gamma, \qquad (7)$$

where the fluid/solid interface Γ_i is considered as formed by the triangular faces of interface sub-elements having all three nodes on the interface and $\hat{\mathbf{n}}$ is the outward unit normal vector on these faces.

The force acting on the solid may be seen as a reaction force and has the same magnitude as the force on the fluid but acts in the opposite direction, $\mathbf{F}_S = -\mathbf{F}_F$.

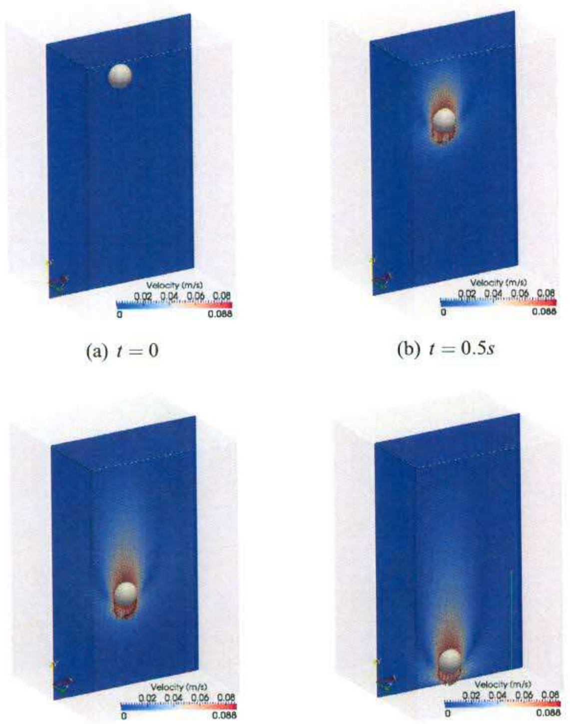
3 APPLICATION

A 3-D fluid-solid interaction problem for which experimental data are available is the falling of a sphere under gravity in an enclosure filled with a viscous fluid. In the experimental setup of ten Cate *et al.* [6] the sphere has a diameter $d = 0.015m$ and is placed inside a box of dimensions $0.1 \times 0.1 \times 0.16m^3$ as shown in Figure 2(a). The sphere is released at a initial height of $0.12m$ from the bottom of the box. The present IB method was tested for the same set of conditions used in the experiment of ten Cate *et al.* [6] and for which numerical results given by an immersed boundary method were presented by Liao *et al.* [7]. The density of the falling sphere is $\rho_s = 1120kg/m^3$ and the fluid properties for the four cases considered are summarized in Table 1. The IB method was used to determine the movement of the falling sphere inside the rectangular enclosure until it reaches the bottom wall. The mesh used for this simulation has $2,545,928$ elements and $533,022$ node. A detail of the mesh near the sphere is shown in Figure 2(b).

Table 1: Fluid properties for the falling sphere.

Case	Re	$\rho(kg/m^3)$	$\mu(Ns/m^3)$
E1	1.5	970	0.373
E2	4.1	965	0.212
E3	11.6	962	0.113
E4	31.9	960	0.058

The evolution with time of the sphere position and velocity is compared with the measured values of ten Cate *et al.* [6] in Figures 3 and 4. The agreement is excellent indicating that the forces acting on the solid and the transient flow solution are well captured by the IB method.



(a) $t = 0$ (b) $t = 0.5s$

(c) $t = 1.0s$ (d) $t = 1.5s$

Figure 5: Sphere falling for the conditions of case $E3$.

Figure 5 shows the position of the falling sphere at various times as well as the velocity distribution in a section across the sphere center for the case $E3$. At $t = 0$ (Figure 5(c)) the sphere is standing still and there is no flow around it. Once the sphere is released, it accelerates gradually and the fluid flow in the box evolves accordingly. At $t = 1.0s$ the sphere almost reached the maximum sedimentation velocity and then at $t = 1.5s$ the deceleration caused by the proximity of the bottom wall has started as confirmed by the results presented in Figure 4.

4 CONCLUSIONS

An accurate IB finite element method for computing the fluid/solid interaction forces is presented. The boundary conditions are imposed on the immersed interface by incorporating into the grid the points where the mesh intersects the fluid/solid boundary. The degrees of freedom associated with the additional grid points are eliminated either because the velocity is known or by static condensation in the case of the pressure. The velocity and pressure solution on interface sub-elements are then used to determine the forces acting on the solid surface. The three-dimensional IB finite element method was used to solve the settling of a single sphere in a viscous fluid. Simulations for different flow regimes indicate an excellent agreement with the experiment for both the position and velocity of the sphere.

REFERENCES

[1] R. Mittal and G. Iaccarino, "Immersed boundary methods," *Annu. Rev. Fluid Mech.*, vol. 37, pp. 239–261, 2005.

[2] R. Löhner, J.R. Cebral, F.E. Camelli, S. Appanaboyina, J.D. Baum, E.L. Mestreau and O.A. Soto, "Adaptive embedded and immersed unstructured grid techniques," *Comput. Methods Appl. Mech. Engrg.*, vol. 197, pp. 2173–2197, 2008.

[3] R. Löhner, S. Appanaboyina and J.R. Cebral, "Comparison of body-fitted, embedded and immersed solutions of low Reynolds-number 3-D incompressible flows," *International Journal for Numerical methods in Fluids*, vol. 57, pp. 13–30, 2008.

[4] F. Ilinca, J.-F. Hétu, "A finite element immersed boundary method for fluid flow around rigid objects," *Int. Journal for Numerical Methods in Fluids*, 2010, in press. doi:10.1002/fld.2222.

[5] F. Ilinca, and J.-F. Hétu, "A finite element immersed boundary method for fluid flow around moving objects," *Computers and Fluids*, vol. 39, pp. 1656–1671, 2010.

[6] A. ten Cate, C.H. Nieuwstad, J.J. Derksen and H.E.A. Van den Akker, "Particle imaging velocimetry experiments and lattice-Boltzmann simulations on a single sphere settling under gravity," *Physics of Fluids*, Vol. 14, pp. 4012–4025, 2002.

[7] C.-C. Liao, Y.-W. Chang, C.-A. Lin and J.M. McDonough, "Simulating flows with moving boundary using immersed-boundary method," *Computers & Fluids*, vol. 39, pp. 152–167, 2010.