Computers and Fluids 137 (2016) 15–35

Contents lists available at ScienceDirect

Computers and Fluids

journal homepage: www.elsevier.com/locate/compfluid

A sharp interface immersed boundary method for moving geometries with mass conservation and smooth pressure variation

Manish Kumar, Somnath Roy*

Department of Mechanical Engineering, Indian Institute of Technology Patna, Bihta, Bihar 801103, India

ARTICLE INFO

Article history:
Received 14 March 2016
Revised 11 June 2016
Accepted 13 July 2016
Available online 21 July 2016

Keywords:
Immersed boundary method
Moving boundaries
Mass conservation
Pressure fluctuations

ABSTRACT

Improper mass conservation and spurious pressure fluctuations are considered as two serious limitations of the immersed boundary method (IBM) for modeling flow over moving/deformable bodies. Earlier attempts to overcome these issues were usually mathematically involved and computationally expensive. In one of our recent work (Kumar M, Roy S, Ali MS. An efficient immersed boundary algorithm for simulation of flows in curved and moving geometries. Comput Fluids 2016; 129: 159–178.), a simple and robust methodology is demonstrated for the sharp interface immersed boundary method which imposes proper boundary conditions for pressure and velocity in the cells intercepted by the solid boundary. In the present paper the mass conservation and pressure fluctuations of the proposed scheme are investigated in detail and the results are presented. The proposed methodology is shown not to add any computational overhead for both fixed and moving boundary problems. An overall second order accuracy is maintained in the discretization and the interpolation schemes. Validation and verification studies have been presented. The achieved results show a second order accurate mass conservation and also exhibit smooth behavior of pressure near the moving surfaces. Species concentration equation is solved to quantify the accuracy in mixing calculations. The present IBM scheme is also used to predict the terminal velocity of objects falling in a quiescent fluid medium under the actions of gravity.

1. Introduction

Immersed boundary method (IBM) was first introduced by Peskin [2] while simulating cardiovascular flows with deformable heart walls. He attributed the superiority of this method over the body-fitted mesh approaches to the fact that IBM does not essentially need the gridlines to conform to the geometry of the flow boundary. In IBM, a rectilinear grid is deployed to discretize the flow domain; while the replication of the boundary condition is achieved through incorporating appropriate forcing terms in the governing equations. Here, the boundary is assumed to be immersed in the flow domain and is represented using Lagrangian descriptions. This indirect way of considering geometries/boundaries helps in avoiding the complexities related to body-fitted approaches for complex, moving and deforming bodies. Especially, the costly dynamic meshing or mesh deformation and solution interpolation steps are avoided as a fixed Cartesian mesh is utilized throughout the simulation. It also helps in achieving better computational efficiency as load balancing and domain decomposition for parallel processes become much simpler in a Cartesian mesh framework [3,4].

Implementing the exact boundary conditions is of utmost importance in IBM. Different IBM implementations have discussed strategies for that. In general, the boundary effects are mimicked by adding appropriate source terms in the governing equations. This can be implemented via continuous forcing approach (CFA) or discrete forcing approach (DFA). In CFA, the forcing function spreads over a band of cells in the vicinity of the actual boundary [2,5,6]. Smooth distributive functions are generally used for this purpose [7]. Peskin [2] used constitutive laws for spring-mass type of systems to obtain the force exerted by the elastic solid boundary on the fluid domain and thus incorporated the effects of boundary deformation. He used a Dirac-delta function for distributing this force over next two grid points on both sides of the boundary. For rigid bodies, Goldstein et al. [8] used a feedback-forcing mechanism to determine forcing function near the boundary. In CFA, this forcing function is incorporated in the governing differential equations before obtaining their discretized form. In CFA, the velocity boundary condition is not satisfied exactly at the interface rather it is satisfied over a distributed region near the boundary. One of the major drawbacks of this approach is its inability to maintain the sharpness of the interfaces as the boundary forces are diffused

* Corresponding author.
E-mail addresses: manish.aspiring@iitp.ac.in (M. Kumar), somnath@iitp.ac.in (S. Roy).

http://dx.doi.org/10.1016/j.compfluid.2016.07.008
0045-7930/

over the adjacent cells. So, determination of the local field variables is difficult in CFA. Moreover, as the forcing term is distributed at both sides of the solid wall, it becomes necessary to solve the equations in the fictitious domain inside the body, which leads to an increased computational overhead in this approach. The computational overheads further increase at higher Reynolds numbers due to the requirement of setting finer grids near boundary. DFA was introduced by Mohd-Yusof [9] and was later explored by Verzico et al. [10] and others. In this approach the forcing function is later added to the discretized set of equations. Smoother functions are used instead of Dirac delta functions. DFA also avoid user specified parameters for the forcing function and the associated stability issues. This formulation can be identified as an indirect boundary condition implementation as the forces are calculated by substituting the variables with boundary values in the discretized set of equations. Although DFA helps in achieving better results in case of low and moderate Reynolds numbers [11,12], it fails to achieve acceptable results in high Reynolds number cases due to the smooth distributive nature of the forcing function. This can be attributed to the fact that at higher Reynolds number regime, the local accuracy is of much importance within boundary layer without smearing its profile. To solve this issue, DFA is further modified to direct boundary implementation techniques and termed as sharp interface immersed boundary method. In most of the approaches of this class (excluding cut-cell methodology), polynomial velocity interpolation functions are used to impose the boundary conditions directly at the fluid cells which have been intercepted by the physical boundary. This is also known as solution reconstruction scheme (SRS). Approaches in sharp interface immersed boundary method can be classified as: (i) Cut-cell (or Cartesian) method, and (ii). Ghost Cell Immersed boundary method (GCIBM). Cut-cell method has similarity with boundary conforming approach, as the fluid-parts of the intercepted cells are re-structured to form a new cell. This restructuring allows having irregular-shaped cells that match well with the topology of the boundary. This was first introduced by Clarke et al. [13] for inviscid flow and later extended by Udaykumar et al. [14–16] and Ye et al. [17] to viscous flow calculations. Calculation of the pressure gradient and flux quantities through the irregular cells is a challenging task in this approach. Slow convergence rate due presence of the small cells is also one of the main limitations of this method [18]. In GCIBM, boundary conditions at the intercepted cells are enforced by extrapolating the field variables at the ghost nodes (nodes which are inside the solid body but have at least one neighboring node in the fluid domain) [19–22]. GCIBM is extended to a new variant called hybrid Cartesian immersed boundary method (HCIBM) [23,24]. The word 'hybrid' can be attributed to the fact that this method is conceptually inspired by Immersed interface method (IIM) [25–27] and Cartesian method. However this method does not incorporate discrete forcing terms into the momentum equations (like IIM) or merge the intercepted cell with fluid cells (like Cut cell method). Here, the solution at the immersed cell is reconstructed using known boundary conditions and local field variables. Overall, DFA is easy to implement and can well-resolve the sharp interfaces at the intersection of fluid and solid without smearing the boundary effects over the adjacent cells. DFA has also been demonstrated to show good accuracy and stability over a large family of problems [7].

Although good agreements with available experimental, analytical or numerical (body fitted approach) data were reported using these various IBM schemes, some of the critical issues are still not well addressed [18]. Mass conservation and spurious pressure fluctuations near the boundary deserve a very special attention especially for flows involving boundary movements. These issues are observed in all variants of discretely forced (DFA) immersed boundary method [18,28,29] and they restrict applicability of IBM at wider ranges of problems involving fluid structure interaction (FSI), mixing and heat transfer calculations. Mass loss at the immersed cell is accounted due to the violation of geometric conservation law [30] at that cell. Hou and Shi [31] noted that area loss is as large as 23% in the immersed cells. A large number of numerical techniques for incompressible flow simulations are based on pressure correction methods to obtain a divergence-free velocity field. Therefore, the violation of local mass conservation at the intercepted cell can lead to spurious pressure fluctuations near the boundary [18]. These issues are amplified when moving or deforming bodies are involved, as the violation of local conservation of mass is augmented by the continuous change of status of the nodes near the boundaries from dead nodes to fluid and vice versa. Liu and Hu [32] demonstrated the causality of spurious pressure fluctuations in great detail. They showed that as the moving boundary enters into a fresh Eulerian cell, an unphysical calculation of normal velocity derivative is obtained, which perturbs the growth of pressure field near boundary. One of the solutions for this problem is the use of Cartesian grid (cut-cell) method [15,17] or very high density of mesh [33]. Strict adherence to mass conservation is obeyed in the cut-cell method. Although cut-cell method eliminates the issue of improper mass conservation and spurious pressure fluctuation, it introduces matrix stiffness and also involves very high numerical complexities in treatment of geometrical irregularities, especially for the 3D bodies [15,17,18]. More difficulties arise as the forcing functions and interpolation stencils change continuously with changes in the cut cell shapes. Whereas, the use of higher grid density suppress the pressure fluctuations substantially increase the computational load. In the diffused interface method, control over spurious pressure fluctuation has been achieved by distributing and smoothing out of forcing terms using smoother delta function but it does so without correcting the conservation error [34,35]. In other discrete forcing techniques, spurious pressure oscillations are more prominent due to unavailability of smoothing scheme while reconstructing the solution at the immersed cell [35–37]. To address this issue, Muldoon and Acharya [38] constrained the velocities at the immersed cells so that the continuity equation is satisfied. They showed good global mass conservation for flows over two-dimensional obstacles. Kang et al. [39], proposed an immersed boundary approximation method in which mass conservation at the immersed cell was obtained by solving a divergence minimization equation and thus forcing the interpolation functions to satisfy overall incompressibility. They obtained noise-free wall-pressure spectra within turbulent boundary layers. However, their methodology was not been tested over moving boundary problems. Luo et al. [28] used the idea of reconstruction of interpolated solution at the immersed cell using another set of equations (called hybrid equations) to counter the temporal oscillations. Apart from giving a control over the pressure oscillations, it also helped in avoiding a lower CFL criteria (<0.02) and finer spatial resolution but introduced one extra set of equations. Seo and Mittal [18] proposed a scheme for sharp interface method utilizing the concepts of cut-cell method. Their method satisfied mass conservation in the immersed cell by considering the mass in/or out due to boundary movement. This formulation helped in achieving better results in terms of mass conservation, transient pressure fluctuation and also avoided stiff equations, but there were still complexities involved (lesser than cut-cell approach) and hence needs intricate coding logistics. Recently, Liu and Hu [32] used local grid refinement, higher time step and modified interpolation schemes coupled with dynamic weighted functions to suppress the temporal pressure fluctuation. However, it has been observed that most of the special treatments for mass conservation and smooth pressure behavior either reduce the sharpness of the boundary or involve complexity associated with higher computational and implementation efforts. This is to mention that the computational