Computers and Fluids 129 (2016) 159–178

Contents lists available at ScienceDirect

Computers and Fluids

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

An efficient immersed boundary algorithm for simulation of flows in curved and moving geometries

Manish Kumar, Somnath Roy*, Md Sujaat Ali

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

ARTICLE INFO

Article history:
Received 8 August 2015
Revised 5 February 2016
Accepted 8 February 2016
Available online 20 February 2016

Keywords:
Immersed boundary method
Moving geometry
Pressure fluctuation
Mass conservation
Computational efficiency
Search algorithm

ABSTRACT

The applicability of immersed boundary method (IBM) in solving flow over complex and moving geometries is often restricted due to the computational overheads associated with the dynamic identification of immersed nodes and the high ratio of solid to fluid nodes. Furthermore, improper mass conservation and the unphysical pressure fluctuations result in large errors in IBM based flow predictions. Earlier attempts to overcome these issues were usually mathematically involved and computationally expensive. In the present paper, a simple and robust IBM implementation has been demonstrated which addresses these issues. The present method preserves the simplicity of sharp interface immersed boundary method and hence avoids complex coding logistics. This method uses a dynamic search algorithm for tagging of the immersed nodes and reduces the computational overheads. Mass conservation is ensured in the intercepted cells through a hybrid SOLA–MAC algorithm. The proposed algorithm iteratively satisfies the mass conservation equation in the intercepted cell and also eliminates spurious fluctuations in the temporal behavior of pressure. An overall second order accuracy is maintained in the discretization and interpolation schemes. The effectiveness and accuracy of the present scheme in handling the mass loss and spurious pressure fluctuation are demonstrated through a number of test cases. Comparisons are made with available experimental and numerical data for different external and internal flow problems.

1. Introduction

Implementation of a numerical method in a complex geometry is usually a challenging task. However, most of the real situations (e.g. biological flow, flows over complex terrain, flow in bend tubes, etc.) involve flow domains which are quite complex in shape. This challenge is further aggravated when the boundaries undergo a movement or deformation. It is also well acknowledged that numerical methods have an edge over experimentation in these cases as the task of achieving similar physical environment is tough and sometimes impossible to accomplish.

Traditionally, the boundary conforming grid approaches (where the grid lines are aligned with the geometry) are used for simulation of flow over non-rectilinear geometries. Techniques for grid generation in a complex geometry (PDE based method or conformal mapping) are usually computationally expensive as they involve iterative solution of large equation systems [1]. Moreover, the simulation methods for moving boundary problems like Deformable-Spatial-Domain/Stabilized Space-Time (DSD/SST) formulation [2,3] or Arbitrary Lagrangian Eulerian (ALE) method [4] needs updating of the mesh (i.e. moving of mesh or remeshing) at every time-step during the movement of the boundary. A significant CPU time is needed to move/regenerate the mesh in order to accommodate the geometrical changes of the flow domain [5]. Further the remeshing is followed by projection of the solution from the old mesh to the newer one which demands computational efforts and also incurs errors [6]. In this context, an alternative approach, named Immersed boundary method (a non-conforming grid approach), has been reported to have an edge over the traditional boundary conforming grid approaches as it does not need costly grid generation and subsequent mapping of results after each computational time step [7,8] and thus offers simplicity and computational efficiency while handling the underline cases. In addition, this method also favors parallel paradigm without an increase in complexity [9,10].

IBM was first introduced by Peskin [11] for simulation of cardiovascular flows. In his proposed method, the entire simulation was performed using a fixed Cartesian grid framework irrespective of the geometry of the heart wall and its state of motion. The boundary effect was mimicked by adding appropriate source terms in the governing equations. Proper implementation of the exact boundary condition is of utmost importance in an IBM. Different implementation strategies have been proposed resulting in several

* Corresponding author. Tel.: +91-612-3028139.
E-mail addresses: manish.aspiring@iitp.ac.in (M. Kumar), somnath@iitp.ac.in (S. Roy), sujaat.me11@iitp.ac.in (M.S. Ali).

http://dx.doi.org/10.1016/j.compfluid.2016.02.009
0045-7930/

variants of IBM. A detailed discussion on this method can be found in Mittal and Iaccarino [12]. Recently, Picano et al. [13] simulated pressure driven turbulent channel flow in presence of 10,000 neutrally buoyant spherical particles using IBM. No other numerical method was reported to address the complexities and calculation efforts involved in resolving thousand spheres (particulates) and implementing no-slip boundary condition on each of those particulates [14]. In another recent work, Van Nimwegen et al. [15] utilized the idea of discrete momentum forcing and mass source/sink (proposed by Kim et al. [16]) along with a Fast Fourier Transform (FFT) based solver. They simulated turbulent flows in straight pipes with large range of roughness topographies with second order accuracy. It can also be noted that turbulent flow predictions using IBM are reasonably accurate even without using any special wall treatment. Tyagi and Acharya [17], Roy and Acharya [18] coupled IBM with large-eddy simulation for prediction of complex turbulent flows. Nam and Lien [19] performed LES-IBM using a coarse mesh in which wall functions were used to calculate the tangential velocities at the near-wall fluid points and velocities at the immersed points were obtained by a bi-linear/tri-linear interpolation using those fluid point values.

Despite the fact that IBM offers simplicity in the study of flows with moving and complex bodies, its utility has been limited by a few serious issues. One of those is the high ratio of solid to fluid nodes/cells (especially in internal flows, like flow in arteries or in S-bend channels) [10,20]. This is undesirable because significant portion of the memory is associated with the solid nodes in which solution is not needed. de Zélicourt et al. [20] proposed an unstructured Cartesian grid method in order to address the issue of high ratio of solid to fluid nodes. Anupindi et al. [10] proposed multi-block approach to avoid the same. The later approach also facilitated the cases in which the flow-domain boundaries were not aligned with the global grid boundaries. However, there are scopes to propose simpler and efficient schemes for these cases. Also, in both of the above mentioned reports, these schemes were never demonstrated for the deforming/ moving boundaries. More detailed discussion on this issue is appended in Section 2.

Furthermore, there is another serious concern in IBM based moving boundary flow simulations. Spurious fluctuations are observed in the time history of pressure at the intercepted (immersed) cells near the moving boundary [21,22]. Liu and Hu [23] discussed the causality of spurious pressure fluctuations in a great detail. They showed that as the moving boundary entered into a fresh Eulerian cell, an unphysical calculation of normal velocity derivative was obtained, which perturbed the growth of pressure field at the vicinity of the moving boundary. This particular problem also restricts the use of IBM in Fluid–Structure interaction (FSI) problems. However, the pressure fluctuation is less severe for fixed bodies and does not need any special treatment [21]. One of the solutions to avoid the spurious pressure fluctuation is using the Cartesian grid (cut-cell) method [24,25]. Although cut-cell method eliminates the spurious pressure fluctuations near the moving boundaries, it introduces issues like matrix stiffness and very high numerical complexities in treatment of geometrical irregularities, especially for 3D bodies [22,24,25]. More difficulties arise due to the fact that the forcing functions and interpolation stencils change continuously with changes in the cut cell shapes as the boundary moves. It has also been found that spurious pressure fluctuation is associated with improper mass conservation at the immersed cells [22,23,26]. This is also reported that global mass conservation is not well-satisfied in most of the simplistic implementations of IBM. In the implementation of IBM through discrete forcing approaches [7,27] (this approach is also known as sharp-interface immersed boundary method), velocity and pressure are directly interpolated at the immersed cells but the continuity equation is never solved explicitly there. Muldoon and Acharya [28] suggested to constrain the velocities at the immersed cells to reduce mass loss and showed good result for flows over two-dimensional fixed obstacles. Immersed finite element method (Zhang et al. [29]) was used by Lee et al. [30] to solve 2D FSI problem and reported good agreement with reported numerical results. In this study, instead of using a reproducing kernel particle method (RKPM), a transformed finite element basis function was used. However, the complexity involved in this method would be more pronounced for 3D cases and also for large deforming bodies. Kang et al. [31] proposed an immersed boundary approximation method in which a divergence minimization equation was solved and a noise-free wall-pressure spectrum was obtained within the turbulent boundary layer. However, this methodology has not been tested for moving boundary problems. Liao et al. [21] proposed a solid-body forcing strategy, which suppressed the pressure oscillation in moving boundary problem. Seo and Mittal [22] proposed a scheme for sharp interface method utilizing the concepts of cut-cell method. They used a forcing function to accommodate boundary movement across the fluid cells. This formulation helped in achieving better results in terms of transient pressure fluctuations and avoided the issue of stiff equations. However, there were implementational complexities involved and hence, this method needed intricate coding logistics. Luo et al. [32] reconstructed the interpolated solution at the immersed cell using another set of equations (called hybrid equations) to counter the temporal oscillation. Their implementation helped in avoiding restrictions like lower CFL criteria (<0.02) and finer spatial resolution but it introduced one extra set of equations. Li and Hu [23] used local grid refinement, higher time steps and dynamic interpolation functions and suppressed the temporal pressure fluctuations.

However, it has been observed that most of the special treatments for smooth pressure behavior are usually computationally intensive. A few of these schemes also reduce the sharpness of the boundary [12], while a few others involve complexity in coding logistics [22,31,32]. Thus, the earlier demonstrated schemes lack the simplicity of IBM and inhibit its usage to large number of applications.

In order to address the above-discussed issues and to extend the capability/reach of IBM for a wider range of applications, the present study attempts to demonstrate a simple implementation of an efficient sharp interface immersed boundary method. In the present scheme, velocity divergence is iteratively reduced to zero at the immersed cell satisfying the mass conservation and hence a smooth pressure field is obtained. An efficient search algorithm is presented and implemented for moving boundary problem, which increases the overall computational speed. Here, the solution (field variables) at the immersed cell is reconstructed using a simple polynomial expression. Marker and Cell (MAC) [33,34] and SOLA [35] solvers are coupled to solve the momentum equations, where MAC is used for the fluid cells and SOLA for the cells intercepted by the solid boundary. Thus, the need of solving the full pressure Poisson equation at the intercepted cells is eliminated. Cases with high ratio of solid to fluid nodes (internal flow) are discussed and the reduction of the computational overhead is demonstrated. The scheme is also tested over complex geometries, in which the inlet and outlet sections are not aligned with the global boundaries. Flow in curved geometries like bend channels and tubes are investigated. Flow over cylinders and airfoils (fixed and moving) are also simulated and compared with the available data.

2. Computational methodology

2.1. Governing equations and basic solver

The non-dimensional governing equations for the conservation of mass and momentum for incompressible flow of a Newtonian