Computers and Fluids 142 (2017) 45–56

Contents lists available at ScienceDirect

Computers and Fluids

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

An immersed boundary-gas kinetic flux solver for simulation of incompressible flows

Y. Sun, C. Shu*, Y. Wang, C.J. Teo, Z. Chen

Department of Mechanical Engineering, National University of Singapore, 10 Kent Ridge Crescent, Singapore 119260

ARTICLE INFO

Article history:
Received 15 August 2015
Revised 30 November 2015
Accepted 11 January 2016
Available online 14 January 2016

Keywords:
Immersed boundary method
Gas-kinetic flux solver
Velocity correction
Moving boundary flows

ABSTRACT

In this paper, an immersed boundary-gas kinetic flux solver (IB-GKFS) is presented for simulation of incompressible viscous flows. In the present scheme, a simple Cartesian mesh is applied for the flow field and a set of Lagrangian points are used to represent the solid boundary. The solution process of the IB-GKFS can be separated into two steps, the predictor step and the velocity correction step. In the predictor step, the intermediate flow field is obtained by applying the gas kinetic flux solver. As the solid boundary is not considered in this step, there is no external force added in gas distribution function during the evaluation of numerical flux at each cell interface. In the velocity correction step, no-slip boundary condition is imposed at all boundary points to make velocity correction on the surrounding Eulerian points. The implicit boundary condition-enforced immersed boundary method is applied so that no-slip boundary condition can be accurately fulfilled and flow penetration is entirely avoided. The decoupled feature of the predictor step and velocity correction step makes the current scheme simple and efficient because the flux on each cell interface only needs to be calculated once in every time step. With a simple Cartesian mesh and flexible boundary condition treatment, the IB-GKFS can be conveniently applied to solve complex and moving boundary problems. Several numerical experiments are conducted to validate the present scheme, including the flow past a stationary circular cylinder and the NACA0012 airfoil, flow past an in-line oscillating cylinder with prescribed motions. After that, the typical fluid-structure interaction problem of one particle sedimentation in a rectangular domain is further considered. The numerical results of those test cases demonstrate the good capability of the present scheme.

1. Introduction

In the last couple of decades, the gas-kinetic scheme has attracted growing attention. The gas-kinetic BGK scheme [1] is one of the representative works. In this approach, the continuous Boltzmann equation is solved at local cell interface to evaluate the gas distribution function at each cell interface. The Bhatnagar–Gross–Krook collision model [2] is adopted in the solution process to control the dissipation in the transport. It has been proven that this scheme can provide stable and accurate solutions for both smooth flow region and discontinuous region with a delicate dissipative mechanism [1]. Furthermore, it is positivity-preserving and the entropy condition is always satisfied [3]. Based on the gas-kinetic BGK scheme, Sun et al. [4] proposed the gas-kinetic flux solver (GKFS) recently, in which the gas distribution function is evaluated in a simple way. Different from the conventional gas-kinetic BGK scheme [1], the non-equilibrium distribution function is evaluated by the difference of equilibrium distribution functions between the cell interface and its surrounding points. As a consequence, the gas-distribution function can be explicitly calculated and simple explicit formulations for the numerical flux are derived. Moreover, it has been shown that the GKFS has an advantage in the computational efficiency as compared with the conventional gas-kinetic BGK scheme [4].

Although the GKFS has been applied to solve both incompressible and compressible flow problems, it is still challenging to simulate flow problems with complex geometries or moving objects because of tedious grid generation and implementation of boundary conditions. In comparison with the body-fitted mesh, the Cartesian mesh is much easier and more efficient to be implemented on. The immersed boundary method (IBM) which uses the simple Cartesian mesh has been widely applied in complex as well as moving boundary problems due to its simplicity and flexibility. In the IBM approach, two types of independent meshes are used: the Cartesian mesh for the flow field and the Lagrangian grid to represent solid boundaries. The basic idea of IBM is that the effect of the immersed boundary is first represented by the restoring force on the Lagrangian grid and then distribute to the adjacent Cartesian mesh

* Corresponding author. Tel.: +65 6516 6476; fax: +65 6779 1459.
E-mail address: mpeshuc@nus.edu.sg (C. Shu).

http://dx.doi.org/10.1016/j.compfluid.2016.01.007

to consider the effect of the boundary condition. In this way, the Cartesian mesh does not have to fit the solid boundary and therefore the tedious grid regeneration or transformation is avoided. The above distinctive features of IBM make it a popular approach in dealing with moving boundary problems.

The IBM was first proposed by Peskin [5] to simulate blood flows in the heart in 1970s. After then, various improvements have been made and numerous variants were proposed, in which the major difference is on the computation of restoring force [6]. Goldstein et al. [7] developed the virtual boundary method in the framework of spectral method. Rather than the way in Peskin [5] where Hook's law is used, the restoring force in virtual boundary method is calculated by a feedback-forcing method based on the difference of virtual and predicted velocities on the boundary. Mohd-Yusof [8] and Fadlun et al. [9] proposed a direct forcing scheme to consider the interaction between the immersed boundary and fluid. In this scheme, the boundary condition is implemented by setting the desired velocity to the points which are close to the immersed boundary at every time step and then the restoring force can be calculated by the momentum equations. Compared to the feedback-forcing method, there is no unknown constant to be determined in the direct forcing scheme, which makes this scheme more applicable. Some modifications are also made to improve the direct forcing scheme [10–12]. In the above IBM versions, the incompressible Navier-Stokes equations are usually solved to obtain the flow field. There is another attempt to combine the IBM with the lattice Boltzmann method (LBM) and significant progress has been made. Feng and Michaelides [13,14] firstly applied the IB-LBM to simulate the rigid particle motion. Niu et al. [15] proposed the momentum exchange-based IB-LBM, in which the forcing term is simply calculated by the momentum exchange of the particles on the boundary. There are other versions of IBM in the literature, such as the Cartesian grid method [16].

In the above immersed boundary methods, one common defect is that the restoring force is pre-calculated, which is the main reason why the no-slip boundary condition is not accurately satisfied [6]. Shu et al. [17] and Wu and Shu [6] discussed this problem and proposed an implicit velocity correction-based method. In this method, the no-slip boundary condition is implicitly imposed by constructing and solving a linear system with the implementation of boundary condition. The restoring force takes the form of velocity correction and can be determined implicitly. Wang et al. [18] extended the implicit velocity correction-based IBM into the lattice Boltzmann flux solver [19] and applied it to study strongly-coupled fluid-structure interaction problems. Similar work related with the implicit velocity correction method can be found in [20,21].

In the present paper, following the idea of Wu and Shu [6], an immersed boundary- gas kinetic flux solver (IB-GFKS) will be proposed to simulate viscous flows with stationary and moving boundaries. Different from the work of Yuan et al. [22] where gas distribution function involves the effect of external force, in the present scheme, a fractional-step method is introduced to decouple the solution of the flow field and the implementation of boundary conditions. In the solution process, the predictor step is first conducted and the gas-kinetic flux solver can be directly applied to solve the flow field without external forcing term. After that, a corrector step is applied to modify the velocity field by applying the boundary condition-enforced IBM. As a result, the no-slip boundary condition can be accurately satisfied at each time step. The application of the current IB-GKFS is as simple as original GKFS. Other advantages of the present scheme are consistent with the IBM, where a regular Cartesian mesh can be used and there is no special treatment for the mesh. To validate the present scheme, it is first used to simulate stationary boundary problems, including the flow past a circular cylinder and flow over a NACA0012 airfoil. After that, it will be applied to solve moving boundary problems, such as the flow past an in-line oscillating cylinder and one particle sedimentation in a rectangular box. The obtained results basically agree well with available data in the literature, which shows the capability of the present method in dealing with complex as well as moving boundaries.

2. Boundary condition-enforced immersed boundary-gas kinetic flux solver

In the immersed boundary method, the immersed body is represented in the form of a closed curve Γ in the computational domain Ω. There are two types of points, the Eulerian points $\boldsymbol{x}$ to discretize the governing equations and the Lagrangian points $\boldsymbol{X}$ to represent the immersed boundary. In this way, the effects of the immersed body are transformed into forcing terms on Eulerian points.

To take the immersed boundary into consideration, the governing equations (Navier–Stokes equations) for the incompressible flow can be written as

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \boldsymbol{u}) = 0 \tag{1}$$

$$\frac{\partial}{\partial t}(\rho \boldsymbol{u}) + \nabla \cdot (\rho \boldsymbol{u}\boldsymbol{u}) = -\nabla p + \nabla \cdot \left[\mu\left(\nabla \boldsymbol{u} + \nabla \boldsymbol{u}^T\right)\right] + \boldsymbol{f} \tag{2}$$

where ρ, $\boldsymbol{u}$, p and μ are the density, velocities, pressure and dynamic viscosity, respectively. $\boldsymbol{f}$ is the restoring force acting on the fluid by the immersed boundary. As pointed out by Shu et al. [17], the effect of the restoring force $\boldsymbol{f}$ in the momentum equations is equivalent to making velocity correction in the flow filed. Therefore, Eqs. (1) and (2) can be solved by applying the following fractional step method:

Step 1 (predictor step): Solve the Navier–Stokes equations without forcing term

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \boldsymbol{u}) = 0 \tag{3}$$

$$\frac{\partial}{\partial t}(\rho \boldsymbol{u}) + \nabla \cdot (\rho \boldsymbol{u}\boldsymbol{u}) = -\nabla p + \nabla \cdot \left[\mu\left(\nabla \boldsymbol{u} + \nabla \boldsymbol{u}^T\right)\right] \tag{4}$$

and obtain the density ρ^{n+1} and intermediate velocity $\boldsymbol{u}*$ at next time step.

Step 2 (corrector step): Correct the velocity field through

$$\frac{\rho^{n+1}\left(\boldsymbol{u}^{n+1} - \boldsymbol{u}^*\right)}{\Delta t} = \boldsymbol{f} \tag{5}$$

It is clear that the flow field is firstly predicted by solving the governing equations in (3) and (4). In this paper, a newly developed gas-kinetic flux solver is used to solve the flow field via the finite volume discretization. After that, the boundary condition-enforced immersed boundary method [6,18] is applied to make corrections of the flow field to satisfy the no-slip boundary condition. Both of the two procedures will be introduced in detail in the following sections.

2.1. Gas-kinetic flux solver for prediction of flow field $\boldsymbol{u}*$

To evaluate the intermediate flow field $\boldsymbol{u}*$ and ρ^{n+1}, the newly developed gas-kinetic flux solver is applied to solve the Navier–Stokes equations without forcing term in Eqs. (3) and (4). It has been proven that the current gas-kinetic flux solver can solve flow problems ranging from incompressible flows to hypersonic flows. Three schemes were proposed in the gas-kinetic flux solver [4] and