Journal of Computational Science 12 (2016) 51–61

Contents lists available at ScienceDirect

Journal of Computational Science

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

Simulation of the motion of two elastic membranes in Poiseuille shear flow via a combined immersed boundary-lattice Boltzmann method

As'ad Alizadeh, Abdolrahman Dadvand*

Department of Mechanical Engineering, Urmia University of Technology, Urmia, Iran

ARTICLE INFO

Article history:
Received 29 June 2015
Received in revised form
17 November 2015
Accepted 22 November 2015
Available online 2 December 2015

Keywords:
Lattice Boltzmann method (LBM)
Immersed boundary method (IBM)
Elastic membrane
Interaction
Grooved channel

ABSTRACT

In this study, the motion, deformation and rotation of two elastic membranes in a viscous shear flow in a microchannel with and without a groove are simulated utilizing a combined LBM–IBM. The membranes are considered as immersed elastic boundaries in the fluid flow. The membranes are represented in Lagrangian coordinates, while the fluid flow field is discretized by a uniform and fixed Eulerian mesh. The interaction of the fluid and membranes is modeled using an appropriate form of the Dirac delta function. Two geometrically different channels, namely, a simple channel and a channel with a groove are considered. In the simple channel case, when the membranes are placed on the symmetry axis of the channel, they continue to move and deform without any lift force and rotation induced. However, when the membranes are located off the symmetry axis, the pressure difference produced in the flow around the membranes would apply lift forces on them and expel them toward the center of the channel. In the case of channel with a groove, it was found that when a membrane tensile modulus was reduced its flexibility as well as rotational speed would increase. This would, in turn, result in a pressure difference in the flow around the membrane. The pressure difference would apply a lift force on the membrane and move it out of the groove. It is worth mentioning that the results of the present simulation have a good agreement with the available numerical and experimental results.

1. Introduction

Most natural, engineering, biological and medical problems include the interaction of elastic solids and viscous fluids [1]. The movement of an immersed object in a fluid including the particle deposition process, the drug delivery through the blood, the red blood cells shape change in the capillaries and arteries, etc. have been studied by many researchers. The difficulty of dealing with such problems would be due to complexity of the grid generation and application of boundary conditions involved.

The numerical methods developed for solving such problems are divided into two main categories: the dynamic- (moving-) mesh approach and the fixed-mesh approach. In the dynamic mesh approach, the computational domain is attached to the moving object and is updated along with the object movement in the fluid [2–4]. In the context of fixed mesh approach, various numerical methods have been proposed [5–8], among which one may refer to the immersed boundary method (IBM). At first this method was introduced for modeling and simulation of blood flow in the heart and heart valves dynamic [9–13]. In this approach, the Eulerian fixed grid and the moving Lagrangian grid are used for the fluid domain and the solid object, respectively. In this approach, the immersed object in the fluid is considered as an object with flexible boundary. The fluid motion over the object would result in small deformations to its boundary. On the other hand, the object's tendency to return to the original state would create a force on the interface between the fluid and the object. Once this force is calculated on the Lagrangian grid points representing the object, the effect of this force on the Eulerian grid points is calculated and is added to the fluid domain equations to account for the effect of the presence of the object. Therefore, in whole computational domain the Navier–Stokes (N–S) equation must be solved with the forcing term resulting from the presence of the object in fluid [8].

There have been many studies carried out to simulate the elastic/rigid immersed body motion in a fluid using the IBM. In most of these studies, the N-S equation has been solved using conventional computational fluid dynamics (CFD) methods such as finite difference, finite element and finite volume discretization methods. Fadlun et al. [14] presented finite difference IBM for modeling fluid flow around a rigid object. The finite element-IBM has been developed to study the interaction of a submerged elastic solid and

* Corresponding author. Tel.: +98 441 3554180.
E-mail addresses: asad.alizadeh2010@gmail.com (A. Alizadeh), a.dadvand@mee.uut.ac.ir (A. Dadvand).

http://dx.doi.org/10.1016/j.jocs.2015.11.008

a fluid flow [15,16]. During the past two decades, the lattice Boltzmann method (LBM) has matured to an alternative and efficient numerical scheme for the simulation of fluid flows and transport problems [17–19]. Unlike conventional numerical schemes based on discretization of macroscopic continuum equations, the LBM is based on microscopic models and mesoscopic kinetic equations.

The LBM has been successfully used to simulate the single viscous fluid flow, the multi-component and multi-phase flows [20,21], flows under the complex geometric boundary conditions [22], suspension of fine particles and flow in micro-channels [23,24], a variety of complex flows of Newtonian and non-Newtonian fluids [25,26], diffusion-convection [27], diffusion-reaction [28], wave equation [29] and Poisson's equation [30]. The LBM is not limited to the hydrodynamic equations. This method can be used for simulation of processes that are difficult to be modeled using continuous methods. These include the blood flow in complex situations, the dynamics of blood components such as red blood cells [31], platelets and materials such as artificial blood. The LBM uses a fixed, regular grid rather than a time consuming adaptive lattice. It calculates fluid properties and velocity using the local values in a way that makes it suitable for massive parallel computing [19]. Unlike the conventional numerical methods such as finite difference method, finite element method and boundary element method that solve partial differential equations (PDEs) to calculate the physical quantities directly, in LBM these quantities are calculated using a distribution function. In addition, the corresponding PDE can be recovered directly by Chapman–Enskog expansion [32].

Lately, a combination of LBM and IBM is used for simulating the fluid–solid interaction involving rigid and/or flexible boundaries. Although many studies have been conducted on the motion of rigid and elastic objects within a fluid, most of these studies have focused on unbounded flows (infinite flow domain). In the limited work carried out to simulate the motion and deformation of elastic structures in shear flow, the numerical methods other than LBM have been used. Feng and Michaelides [33] proposed the first model of IBM–LBM. Their combined method involved most of the desirable properties of LBM and IBM. It uses an Eulerian grid (lattice) for flow field and a Lagrangian grid to trace the dynamics of solid particles (immersed object). This method not only represents the boundary smoother than the LBM, but also prevents fluctuations of velocities and forces acting on the object that can be seen in LBM. In general, it is so powerful in solving the problems with structure deformation [33]. This method was improved by Wu and Shu [8] and they implicitly calculated and applied the forces caused by the fluid acting on the immersed boundary, which in fact had a significant impact on the cost and accuracy. They simulated the motion of a particle in a typical shear flow and the motion of two particles in a channel using their proposed method.

Dupuis et al. [34] studied the flow over an impulsively started cylinder at moderate Reynolds (*Re*) number. They investigated how the coupling method of the forcing term between the Eulerian and Lagrangian grids could affect the results.

Zhang et al. [35,36] studied the dynamic behavior of red blood cell in shear flow and channel flow and investigated several hemodynamic and rheological properties. Cheng et al. [37] have proposed a proper model to simulate the fast boundary movements and high pressure gradient occurred in the fluid–solid interaction. In their research, mitral valve jet flow considering the interaction of leaflets and fluid has been simulated. Navidbakhsh and Rezazadeh [38] carried out a numerical study on the behavior of malaria-infected red blood cell. Vahidkhah and Abdollahi [39] simulated the motion of a massless elastic object in a two-dimensional viscous channel flow numerically using IBM–LBM. Dadvand et al. [40] investigated numerically the motion and deformation of a red blood cell in a viscous shear flow utilizing a combined LBM–IBM. Le et al. [41] used immersed interface method (IIM) to simulate the membrane motion in a two-dimensional channel flow.

In the present work, the motion, deformation and rotation of two elastic membranes in a viscous shear flow in a microchannel with rectangular cross section are simulated utilizing a combined LBM–IBM. Two geometrically different channels, namely, a simple channel and a channel with a groove are considered. In the simple channel case, the membranes are placed both on the symmetry axis of the channel and off the symmetry axis.

2. Governing equations

It was mentioned in introduction that in the IBM the fluid is represented on an Eulerian coordinate and the structure is represented on a Lagrangian coordinate. A typical two-dimensional example of an elastic solid membrane with curved boundary has been shown in Fig. 1. Consider a flexible solid membrane with the curved boundary Γ immersed in the two-dimensional incompressible viscous fluid domain Ω. The membrane boundary Γ is characterized by the Lagrangian parameter s, and the fluid domain Ω is represented by Eulerian coordinates $\vec{x}$. Hence any point on the membrane can be written as $\vec{X}(s,t)$ where s is arc length, and t is time.

Hence, the equations governing the combination of fluid and solid motions are as following:

$$\nabla \times \vec{u} = 0 \tag{1}$$

$$\rho\left(\frac{\partial \vec{u}}{\partial t} + \vec{u} \times \nabla \vec{u}\right) = -\nabla p + \eta \nabla^2 \vec{u} + \vec{f}\left(\vec{x}, t\right) \tag{2}$$

$$\vec{f}\left(\vec{x}, t\right) = \int_{\Gamma} \vec{F}(s,t)\,\delta\left(\vec{x} - \vec{X}(s,t)\right) \mathrm{d}s \tag{3}$$

To satisfy the no-slip boundary condition on the fluid–solid interface, the velocity of any point on the solid surface must be equal to that of the adjacent fluid particle, i.e.,

$$\vec{U}(s,t) = \vec{u}\left(\vec{X}(s,t), t\right) = \frac{\partial \vec{X}(s,t)}{\partial t} = \int_{\Gamma} \vec{u}\left(\vec{x}, t\right)\delta\left(\vec{x} - \vec{X}(s,t)\right) \mathrm{d}\vec{x} \tag{4}$$

In the above equations, ρ and η are the mass density and dynamic viscosity of the fluid, respectively. In addition, $\vec{u}$ and p indicate the velocity and pressure fields, respectively. The term $\vec{f}$ on the right-side of Eq. (2) denotes the membrane forces (tensile and bending) due to the elastic boundary immersed in the fluid.

Eq. (3) indicates that the force density of the fluid $\vec{f}\left(\vec{x}, t\right)$ is obtained from the force density of the membrane $\vec{F}(s,t)$. Eq. (4) represents the no-slip condition at the fluid–solid interface, as the solid

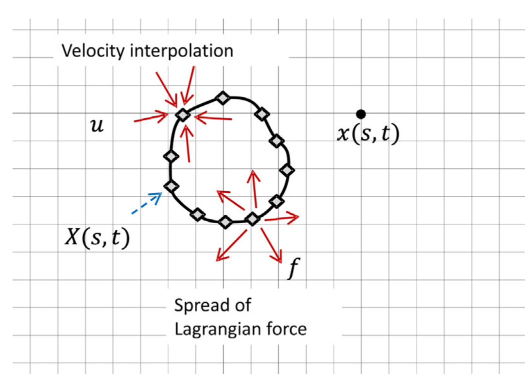

Fig. 1. Schematic representation of immersed boundary (Lagrangian coordinates) and Eulerian mesh for fluid and spreading the Lagrangian force from the membrane boundary points to the Eulerian nodes.