Journal of Computational Physics 294 (2015) 524–546

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

Numerical simulation of particle motion using a combined MacCormack and immersed boundary method

P. Zhang *, A. Benard

Department of Mechanical Engineering, Michigan State University, East Lansing, MI 48824, United States

ARTICLE INFO

Article history:
Received 12 July 2014
Received in revised form 2 January 2015
Accepted 10 March 2015
Available online 18 March 2015

Keywords:
Fluid–structure interaction
Particulate flows
Particle migration
DNS
Immersed boundary method
MacCormack

ABSTRACT

A numerical approach is presented for the direct numerical simulation of particle motion that combines the MacCormack scheme and the immersed boundary method. It exhibits the advantageous features of the explicit MacCormack scheme which is second-order accurate in time and space with simplicity in programing. The approach solves the compressible Navier–Stokes equations and uses the immersed boundary method to tackle the interactions between the fluid and the suspended particles. The force due to the interaction of two phases is computed via an elastic forcing method. The numerical approach is validated using uniform flow past a stationary circular cylinder, sedimentation of circular discs, and particle motion (orientation and translation) in unidirectional flows. Results are also compared to simulation obtained from a mixture model for solid particles for the same flow conditions.

1. Introduction

Predicting the motion of suspended solids is relevant to composite materials processing, biological systems (red blood motion, aquatic animal locomotion), and related to the motion of vehicles or aircrafts. Amongst the methods presented for modeling a moving solid with an underlying fixed Cartesian mesh are immersed boundary methods [29,8,24,6,14,39] and fictitious domain methods [28]. Mittal and Iaccarino [22] conducted a detailed review on both approaches which involve a structure "immersed" in a fluid such as shown in Fig. 1 with an overlaid Cartesian mesh.

In the immersed boundary method, a set of discrete points are used to delineate or mark the structure surface and evaluate the force at those points. The points are treated as virtual entities existing in the flow domain and correspond to the structure surface of interest. The immersed boundary method was first proposed by Peskin [29,30] and in this approach Lagrangian points, shown in Fig. 1, are linked with springs representing the structure. The springs are dampened or stretched by the surrounding flow. Goldstein et al. [8] used a similar idea for a stationary rigid structure by applying large stiffness coefficients. Lee [42] remarked that a large stiffness coefficient may cause numerical instabilities if the time step used is larger than the characteristic time scales of the spring oscillation. Instead of evaluating the force on the Lagrangian points with an elastic boundary assumption, Mohd-Yosof [24] obtained a force density term from the momentum equations by imposing a desired velocity at the points wherever the surface is intersected with the Eulerian mesh via a B-Spine function. Fadlun et al. [6] applied the latter method to flow interactions with complex geometries. The desired velocity on

* Corresponding author. Tel.: +1 517 977 6478. Present address: 14711 Norwood Peak Ln, Cypress, TX 77429.
E-mail address: pusheng.zhang@yahoo.com (P. Zhang).

http://dx.doi.org/10.1016/j.jcp.2015.03.021
0021-9991/

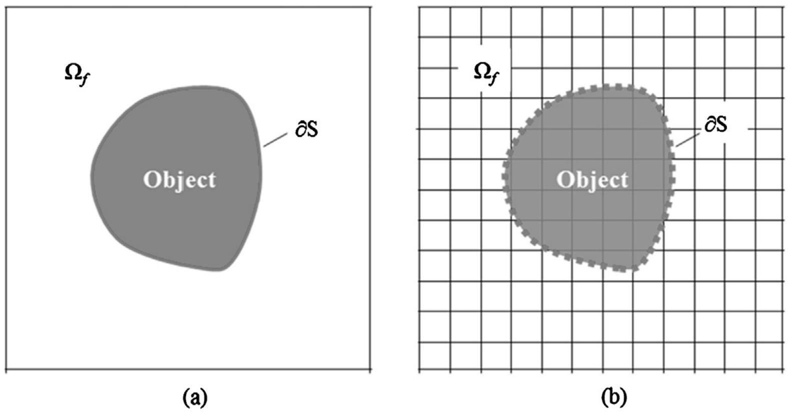

Fig. 1. Schematic of an Eulerian mesh for the fluid domain Ω_f and a Lagrangian mesh around the fluid–object interface ∂S.

the boundary is obtained via volume fraction weighing or a linear interpolation. Kim et al. [14] improved the approximation scheme for stability and accuracy by using a second-order-accurate interpolation scheme. However oscillations occur at the boundary when the above methods were used to move the structure boundaries [38]. Uhlmann [39] improved the stability of the direct forcing method by combining the Dirac delta function of Peskin [29] to smooth the force on the boundary of a rigid body. The IB method has significantly evolved and is now a method of choice when dealing with fluid–structure interaction problems [7,18,10,6,39,25].

In this work, the IB method is employed to model the motion of particles and the compressible N–S equation are solved using an explicit MacCormack scheme [20]. The MacCormack scheme has second-order accuracy in space and time and has been widely applied to compressible and incompressible flows due to its ease of implementation and parallelization. Literature on using the IB method combined with the MacCormack scheme for fluid–structure interaction simulations cannot be found. In addition, to compute the Lagrangian forcing representing the solid immersed boundary, two methods including an elastic forcing method and an interpolating scheme are employed and compared below. A collision scheme presented by Glowinski et al. [9] is used to account for rigid body collisions. Simulations of flow passing a stationary cylinder, particle sedimentation, and inertia-induced particle sedimentation are used to tests the method. The method is also contrasted with results of the mixture model obtain from the commercial software ANSYS FLUENT [2]. The behavior of neutrally buoyant particles is studied for Couette–Poiseuille flow problems.

2. The MacCormack scheme and the immersed boundary method

In this work, an explicit finite different algorithm called MacCormack scheme is employed to solve the governing equations for particulate flows. Particulate flows are inherently unsteady flow which require an accurate timewise solution to the complete transient Navier–Stokes equation. The time-dependent Navier–Stokes equation has a mathematical nature of mixed parabolic and elliptic behaviors that can be well handled by a time-matching method. On the other hand, in order to obtain accurate trajectory of the particle motion, a very small time step is inevitable to reduce the truncation error while solving the partial differential equations. Therefore, the numerical scheme selected has to be highly efficient for the reduction of the computational effort especially for dense flows.

The MacCormack scheme being used is a time-matching explicit approach and is well posed to particulate flows due to its efficiency. Many numerical schemes used for solving the incompressible Navier–Stokes equations have to involve solving a Poisson equation for pressure. Finding a solution to the Poisson equation is the most costly step. Due to the elliptic mathematic behavior of the equation, the unknown pressure at all the grid points has to be solved simultaneously by solving a large system of algebraic equations depending on the mesh size. In contrast, the MacCormack scheme surmounts the difficulty of solving the Poisson equation by relating the pressure to the flow density through an equation of state. The technique introduces an artificial compressibility proposed by Chorin [5] which reduces the computational effort dramatically by solving the governing in a fully explicit manner.

To implement the IB method in combination with the MacCormack scheme, the effect of the immersed structure on its surrounding flow is modeled through a force density vector **f** included in the compressible Navier–Stokes equation. For a two-dimensional problem, the equations for the MacCormack/IB method are presented in a generic form for the transport equation as

$$\frac{\partial \mathbf{G}}{\partial t} + \frac{\partial \mathbf{E}}{\partial x} + \frac{\partial \mathbf{F}}{\partial y} = \mathbf{f} \tag{1}$$

where **G**, **E**, and **F** are arrays with components given by continuity the components of the momentum equation, i.e.