Available online at www.sciencedirect.com
ScienceDirect
Journal of Computational Physics 227 (2008) 6660–6680
JOURNAL OF COMPUTATIONAL PHYSICS
www.elsevier.com/locate/jcp

Derivation and validation of a novel implicit second-order accurate immersed boundary method

Andreas Mark *, Berend G.M. van Wachem

Chalmers University of Technology, Department of Applied Mechanics, 412 96 Gothenburg, Sweden

Received 16 May 2007; received in revised form 17 March 2008; accepted 20 March 2008
Available online 1 April 2008

Abstract

A novel implicit second-order accurate immersed boundary method (IBM) for simulating the flow around arbitrary stationary bodies is developed, implemented and validated in this paper.

The IBM is used to efficiently take into account the existence of bodies within the fluid domain. The flow domain consist of simple Cartesian cells whereas the body can be arbitrary. At the triangulated interface of the body and the fluid, the immersed boundary, the coefficients obtained from discretizing the Navier–Stokes equations are closed with a second-order accurate interpolation arising from the immersed boundary condition employed at the interface. Two different conditions are developed in this paper and it is shown that for the mirroring method the resulting coefficients lead to a well-posed and diagonally dominant system which can be efficiently solved with a preconditioned Krylov sub-space solver. The immersed boundary condition generates a fictitious reversed velocity field inside the immersed boundary, which is excluded from the continuity equation to account for the presence of the IB in the pressure correction equation, resulting in no mass flux over the IB. The force acting on the object from the fluid is determined by integrating the pressure and the viscous forces over the object.

The method is validated by simulating the flow around a sphere for a range of *Re* numbers. It is shown that the drag is very well in agreement with experimental data. Accuracy and convergence studies are employed, proving the second-order accuracy of the method and showing the superiority in convergence rate compared to other IBM. Finally the drag force of a cluster of non-spherical particles is employed to show the generality and potential of the method.

Keywords: Immersed boundary method; Finite-volume method; Implicit method; Immersed boundary condition; Fluid–structure interaction

1. Introduction

Although dispersed multiphase flows are common in both nature and industry, their fundamental behavior is still poorly understood. In recent years, multiphase computational fluid dynamics (CFD) has become an

* Corresponding author. Tel.: +46 31 772 50 34; fax: +46 31 18 09 76.
E-mail addresses: andreas.mark@chalmers.se (A. Mark), berend@chalmers.se (B.G.M. van Wachem).

0021-9991/$ - see front matter © 2008 Elsevier Inc. All rights reserved.
doi:10.1016/j.jcp.2008.03.031

important tool to model and gain more insight into such flows. However, to successfully predict and simulate multiphase flows at a large scale, the fundamental behavior at small scale needs to be understood. The majority of multiphase flows are fluid–solid flows, in which a dispersed solid phase is present in a continuous fluid phase.

Almost all multiphase CFD today is directly aimed at resolving large scales. One of the key issues in successfully describing fluid–solid flows is the physics of the interaction of the two phases. This interaction is a complex process and, to successfully capture it, all the important length and time scales must be resolved; hence true direct numerical simulations (DNS) are required. In true DNS, the full Navier–Stokes equations are solved, taking into account the particle surface, thereby resolving the boundary layer and wake of each particle. In this article, we present a second-order accurate and relatively inexpensive true DNS method to accurately resolve the flow around individual particles.

One of the first methods developed to accurately simulate fluid–solid flows is the arbitrary Lagrangian–Eulerian (ALE) method, proposed by Hu [11,12]. In this method, an unstructured grid is created around the particles and adapted as the particles move. This requires remeshing each time step. Although the method shows good results, the remeshing is computationally very expensive and makes it difficult to provide accurate results. To decrease the computational cost involved by remeshing and to potentially increase the accuracy, three methods without the necessity of remeshing have been employed in the literature: Cartesian, Lagrange Multiplier and Immersed Boundary methods.

In the Cartesian method, or the cell-splitting method [26,33,13,17], the domain mainly consists of Cartesian grid cells. Where a Cartesian cell is cut by the surface segment of a particle, the fluid cells are split and locally unstructured cells are formed. The flow is resolved for the new grid with irregular boundary conditions. The drawback is that the irregular boundary conditions are hard to generalize and small cells can lead to problems with the conservation and stability in the solver.

The Lagrange multiplier method [7,8,6] is a finite-element method where the coupling between the boundary of the particle and the fluid is introduced by a Lagrange multiplier in the integral formulation of the Navier–Stokes equations. The weak coupling is implicitly implemented and, by minimizing the combined integral formulation, the solution of the flow field is found. The method is second-order accurate in space. The drawback with the method is that it is relatively expensive and only preserves the mass and momentum globally, not locally.

The immersed boundary (IB) method resolves these difficulties by using a Cartesian grid in the whole domain. Where the particle surface segments (or immersed boundaries) cross the Cartesian grid cells, the flow variables are directly modified, without the necessity of rearranging the grid. This modification of the flow variables, representing the coupling between the two phases, can be implemented in several different ways. One common way is to exert a Lagrangian force at the immersed boundary onto the fluid phase. The Navier–Stokes equations are then solved on the Cartesian grid, including a number of point forces, representing the influence of the immersed boundaries on the fluid flow.

There exist in general two methods for calculating the forces to represent the immersed boundaries. In the first approach, the forces are calculated from the momentum equations and distributed on the Cartesian mesh by employing a distribution function, first proposed by Peskin [29]; this is called the distributive IB method. The second approach is a non-distributive method, where the Eulerian force is calculated from the momentum equations directly or by employing a boundary condition exactly at the immersed boundary.

The distributive IB method [29,18] employs a function, based on the discrete Dirac delta function, to distribute the Lagrangian forces on the Cartesian mesh. As a result, the Lagrangian forces are smeared out over several grid cells, leading to a smeared representation of the immersed boundary and to only first-order accuracy. In this method, the Lagrangian force is determined explicitly from the generalized Hooke's law.

Goldstein et al. [9] also employs the distributive IB method, but calculates the Lagrangian force by constraining the fluid to the no-slip boundary condition on the immersed boundary. The force proposed by Goldstein et al. [9] contains two unphysical parameters which are dependent on the time scales of the flow. These constants tend to become relatively large, which gives stiff equations to solve. Due to this the method only works well for small time steps. Silva and co-workers [21,22,27] improved the calculation of the Lagrangian forces by making the forces originate from a physical point of view, generally increasing the robustness. The method is explicit and first-order accurate.