Available online at www.sciencedirect.com
ScienceDirect
Journal of Computational Physics 227 (2008) 9121–9137
JOURNAL OF COMPUTATIONAL PHYSICS
www.elsevier.com/locate/jcp

A vortex level set method for the two-way coupling of an incompressible fluid with colliding rigid bodies

M. Coquerelle [a,b], G.-H. Cottet [a,*]

[a] *Université de Grenoble and CNRS, Laboratoire Jean Kuntzmann, BP 53, 38041 Grenoble Cedex 9, France*
[b] *INRIA Grenoble – Rhône-Alpes, 38334 Saint Ismier Cedex, France*

Received 23 June 2007; received in revised form 12 February 2008; accepted 26 March 2008
Available online 12 April 2008

Abstract

We present a vortex method for the simulation of the interaction of an incompressible flow with rigid bodies. The method is based on a penalization technique where the system is considered as a single flow, subject to the Navier–Stokes equation with a penalization term that enforces continuity at the solid–fluid interface and rigid motion inside the solid. Level set functions are used to capture interfaces, compute rigid motions inside the solid bodies and model collisions between bodies. A vortex in cell algorithm is built on this method. Numerical comparisons with existing 3D methods on problems of sedimentation and collision of spheres are provided to illustrate the capabilities of the method.

Keywords: Incompressible flow; Fluid–solid interaction; Vortex method; Level set method; Penalization method

1. Introduction

Vortex methods have long been used for the simulation of bluff-body flows. The success of these methods comes from the combination of several features, well illustrated in the pioneering work [18] for 2D cylinder wakes, as well in more recent 3D calculations [21,11]: stability, localization of the computations in the regions of interest and natural treatment of far-field boundary conditions. Moreover when combined with fast grid solvers to compute velocity fields, and more generally all quantities that are not directly related to the transport of vorticity, these methods offer for external flows economical alternatives to more traditional grid-based Eulerian solvers.

In this paper we consider the extension of the Vortex In Cell method [7] to the case where several rigid bodies interact under the action of gravity with an incompressible flow. The difference with previously considered cases is that the rigid bodies move freely, and eventually collide, under the combined action of gravity and

* Corresponding author. Fax: +33 4 76 63 12 63.
E-mail address: lmc@imag.fr (G.-H. Cottet).

0021-9991/$ - see front matter
doi:10.1016/j.jcp.2008.03.041

forces imparted by the flow. The numerical challenge in this problem lies in the coupling of stresses and velocities at the moving fluid/solid interfaces and in the modeling of collisions.

Let us first consider the fluid/solid interaction problem. This type of problem is often dealt with by ALE (for arbitrary Lagrangian–Eulerian) methods where the interface is tracked, flow and solid equations are solved separately, and continuity conditions for the velocity and stress are explicitly enforced at the interface (see [16] and the references therein). These methods are accurate but hard to implement in 3D and expensive, in particular if several objects interact in a non laminar flow. Here we consider a different approach. The fluid–solid system is considered as a single flow. The interface is captured by a level set method and the rigid constraint together with the continuity conditions are implicitly recovered through a penalization method. When formulated in the primitive variables, this method extends to the case of the two-way fluid–solid coupling the method of [1] which was introduced for the computation of bluff-body flows. The benefit that can be expected from this approach comes from the fact that fast grid solvers, based for instance on particle or finite-difference methods, can be used, leading to substantial computational savings, in particular for 3D cases, compared to body fitted methods.

Our method is related in particular to the fictitious domain approach of [14] and to the projection method of [20]. The fictitious domain approach of [14] was already derived as an alternative to ALE methods. The rigid motion inside the solid bodies is enforced through a Lagrange multiplier, which somehow plays the role of the pressure for the incompressibility constraint. The method is defined in a variational framework which is well suited for finite element discretization. A number of validations of this method have been performed and we will extensively use them to test our approach. A more recent work in the same spirit is the penalty method of [17]. In this work the rigid motion is also modeled in a variational framework leading to a minimization problem for the velocity. The functional to minimize in the space of rigid motions inside the body is approximated by a functional over all velocities with a penalization of the deformation inside the rigid bodies. In our method the interfaces are captured by advection equation which are connected to the flow equations only through the advection velocity. The added flexibility in the choice of the solvers makes the method fast and rather simple to implement, in particular when several bodies are considered.

The projection method of [20] consists of alternating Navier–Stokes solvers and projection steps where the rigid motion is recovered inside the body. This method is in the same spirit as the classical splitting method for the Navier–Stokes equations to enforce divergence free flows. As we will see it appears as a particular case of our penalization method, when a first order explicit time-discretization is chosen. Our approach is more general. In particular an implicit treatment of the penalization term allows larger penalization coefficients and thus more accurate results.

Besides providing a simple way to capture body boundaries and enforce rigid motions, an additional benefit of level set methods is to enable simple collision models to deal with contacts. Our starting point is a simple dynamical system with short range Hamiltonian forces to model single point collisions. The level set approach allows to generalize in a very straightforward way this system to more general collisions. Level set functions are used both to measure distance between interfaces and to localize the forces on the interfaces. The resulting collision forces appear just as an additional force in the single flow equation representing the fluid/solid system.

In this paper we focus on a vorticity formulation based on this approach which is appropriate for a particle discretization. We derive and illustrate particle-grid algorithms suitable for the simulation of the level set fluid–structure model.

This work is part of a series of papers where we systematically investigate multi-phase flow modeling and the associated algorithms for generic fluid–structure interaction problems. [2] is concerned with the numerical analysis of the underlying penalization method in primitive variables, and [6] deals with application of the present method in computer graphics. Refs. [8,9] deal with modeling and numerical issues of related methods for the case of elastic solids.

An outline of the paper is as follows. In Section 2 we present our flow model. In Section 3 we describe its particle discretization. In Section 4, to validate our method and discuss its efficiency, we focus on test cases where quantitative comparisons with [14,20] are possible, namely the sedimentation and collision of spheres. Finally Section 5 is devoted to concluding remarks and perspectives.