Journal of Computational Physics 231 (2012) 3246–3263

Contents lists available at SciVerse ScienceDirect

Journal of Computational Physics

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

Efficient symmetric positive definite second-order accurate monolithic solver for fluid/solid interactions

Frédéric Gibou [a,b], Chohong Min [c,*]

[a] Mechanical Engineering Department, University of California, Santa Barbara, CA 93106, United States
[b] Computer Science Department, University of California, Santa Barbara, CA 93106, United States
[c] Mathematics Department, Ewha Womans University, Seoul 120-750, South Korea

ARTICLE INFO

Article history:
Received 9 September 2010
Received in revised form 22 June 2011
Accepted 3 January 2012
Available online 20 January 2012

Keywords:
Fluid
Solid
Rigid body
Interaction between fluid and solid
Level-set method
Navier–Stokes equations

ABSTRACT

We introduce a robust and efficient method to simulate strongly coupled (monolithic) fluid/rigid-body interactions. We take a fractional step approach, where the intermediate state variables of the fluid and of the solid are solved independently, before their interactions are enforced via a projection step. The projection step produces a symmetric positive definite linear system that can be efficiently solved using the preconditioned conjugate gradient method. In particular, we show how one can use the standard preconditioner used in standard fluid simulations to precondition the linear system associated with the projection step of our fluid/solid algorithm. Overall, the computational time to solve the projection step of our fluid/solid algorithm is similar to the time needed to solve the standard fluid-only projection step. The monolithic treatment results in a stable projection step, i.e. the kinetic energy does not increase in the projection step. Numerical results indicate that the method is second-order accurate in the L^{∞}-norm and demonstrate that its solutions agree quantitatively with experimental results.

1. Introduction

Understanding and predicting fluid–structure interaction is crucial in many areas of science and engineering. Examples include the study of particle motion in liquids, with application to industrial solidification or pertaining to the understanding of sedimentation in geology, the study of heart valves or aneurisms, or the design of engineering systems such as swimming structures. Considerable work has been done on the design of numerical methods to provide an accurate predictive tool for fluid/rigid-body coupling. Arbitrary Lagrangian–Eulerian (ALE) schemes have been successfully employed in the case where the structure deformation is low [10]. Schemes based on the Lattice–Boltzmann method have also been used (see e.g. the recent work of [12]). Immersed boundary methods have provided a framework for coupling fluids and rigid or elastic bodies. Several applications of such methods exist in the literature, with maybe the most famous one being the application of blood flow in the heart [25,26]. Within this framework, the coupling is expressed through the use of a delta formulation, which smears some of the variables near the interface but provides a straightforward approach. Coquerelle and Cottet [8] introduced a vortex method [9] for the simulation of the interaction of an incompressible flow with rigid bodies. In particular, they consider a single flow and use a penalization technique to enforce continuity at the solid–fluid interface and rigid motion inside the solid. In this case, the interface is represented by a level-set function and the quantities are smeared across the interface.

* Corresponding author.
E-mail addresses: fgibou@enginnering.ucsb.edu (F. Gibou), chohong@ewha.ac.kr (C. Min).

0021-9991/$ - see front matter © 2012 Elsevier Inc. All rights reserved.
doi:10.1016/j.jcp.2012.01.009

In this paper, we are focusing on the two-way coupling between a fluid and a rigid body and present a sharp treatment for their coupling. In this context, existing numerical methods fall into two categories: partitioned approaches, where equations for the fluid are first solved before equations for the solid are solved, and monolithic approaches, where equations for the fluid and the rigid body are solved simultaneously [1,11]. An advantage of the partitioned approach is that existing fluid and structure solvers can be used in an iterative process. A disadvantage is that it is difficult to develop methods that guarantee numerical stability, which in practice can translate into spurious numerical oscillations. Also, the accuracy may suffer from the lack of strong coupling, even if this can somewhat be remedied with sub-iteration procedures. Monolithic approaches have the potential of being more stable and more accurate, but their design and analysis is not as straightforward.

In [3], Batty et al. have formulated the interactions between fluids and non-deformable solids as a kinetic energy minimization problem, and discretized the interactions through the corresponding Euler–Lagrange equation. In [28], Robinson-Mosher et al. have presented a monolithic approach based on a projection framework. A particularly important feature of this approach is the design of a symmetric positive definite (SPD) system for the projection step, an improvement on their previous work that produced an indefinite system [29]. Designing a SPD system guarantees that its solution can be computed with fast solvers. Their method also preserves momentum, which impacts positively on the stability of the method. However the method is only first-order accurate in the L^∞-norm and it is not clear how to best precondition the linear system.

In the present paper, we present an SPD approach that is second-order accurate in the L^∞-norm and for which the linear system can be preconditioned easily. Our method is unconditionally stable and conserves the momentum transfer between the fluid and the solid in the projection step. We take a fractional step approach, where the intermediate state variables of the fluid and of the solid are solved independently, before their interactions are enforced via a projection step. Using the Heaviside function of the fluid region, the projection is formulated as a simple Poisson-type equation and can be easily implemented as a small addition to the standard projection method for fluids on a MAC grid. We show how one can use the standard preconditioner used in fluid simulations to efficiently precondition the linear system associated with our fluid/solid coupling projection step, providing a simulation framework as efficient as standard fluid solvers. We present numerical results in two and three spatial dimensions that indicate that our method is second-order accurate. We also provide simulation results that are in agreement with the experimental results of Cate et al. [6].

2. Governing equations

The interactions between a fluid and a rigid body are modeled by equations of motion for the fluid and the solid, as well as their respective boundary conditions. We consider a computational domain $\Omega = \Omega^f \cup \Omega^s$ with boundary $\partial\Omega$, where a rigid body defined by Ω^s with boundary Γ is immersed in a fluid defined in a region $\Omega^f \subset \Omega$; see the schematic in Fig. 2.1.

2.1. Fluid equations

In the case where the viscosity is constant the motion of incompressible flows is described by the incompressible Navier–Stokes equations of the following forms:

$$\boldsymbol{U}_t + (\boldsymbol{U}\cdot\nabla)\boldsymbol{U} + \frac{\nabla p}{\rho} = \frac{\mu}{\rho}\Delta\boldsymbol{U} + \boldsymbol{g} \text{ in } \Omega^f,$$
$$\nabla\cdot\boldsymbol{U} = 0, \text{ in } \Omega^f,$$

where ρ is the fluid's density, μ is the fluid's viscosity, $\boldsymbol{U} = (u, v, w)$ is the velocity field and $\boldsymbol{g}$ the gravity field.

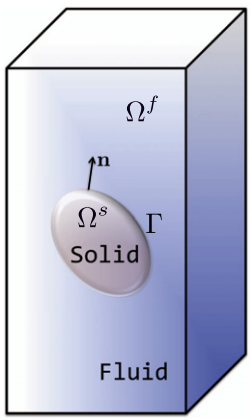

Fig. 2.1. Schematic of a typical fluid/solid computational set up.