Journal of Computational Physics 277 (2014) 201–227

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

Level set immersed boundary method for coupled simulation of air/water interaction with complex floating structures

Antoni Calderer [a], Seokkoo Kang [b], Fotis Sotiropoulos [a,*]

[a] St. Anthony Falls Laboratory and Department of Civil Engineering, University of Minnesota, 2 Third Avenue SE, Minneapolis, MN 55414, United States

[b] Civil and Environmental Engineering, Hanyang University, Seoul, South Korea

ARTICLE INFO

Article history:
Received 13 January 2014
Received in revised form 25 July 2014
Accepted 4 August 2014
Available online 11 August 2014

Keywords:
Fluid–structure interaction
Two-phase free surface flow
Large eddy simulation
Level set method
Immersed boundary method
Floating structures
Falling wedge

ABSTRACT

We propose a new computational approach for simulating the coupled interaction between air–water flows and arbitrarily complex floating rigid bodies. The numerical method integrates the fluid–structure interaction (FSI) curvilinear immersed boundary (CURVIB) method of Borazjani et al. (2008) [21] with a level set approach for simulating free surface flows in arbitrarily complex domains. We show that when applying the CURVIB method to simulate two-phase flow FSI problems the approach used to calculate the force imparted on the body is critical for determining the overall accuracy of the method. We develop and demonstrate the accuracy of a new approach for calculating the force, namely the pressure projection boundary condition (PPBC), which is based on projecting the pressure on the surface of the body using the momentum equation along the local normal to the body direction. Extensive numerical tests show that the new approach greatly improves the ability of the method to correctly predict the dynamics of the floating structure motion. To demonstrate the predictive capabilities of the method and its ability to simulate non-linear free surface phenomena, such as breaking waves, we apply it to various two- and three-dimensional problems involving complex rigid bodies interacting with a free surface both with prescribed body motion and coupled FSI. We show that for all cases the proposed method yields results in very good accuracy with benchmark numerical data and available experiments. The simulations also reveal the onset of dynamically rich, energetic coherent structures in the air phase induced by the waves generated as the rigid body interacts with the free surface.

1. Introduction

Fluid–structure interaction (FSI) problems involving complex floating structures are encountered in a wide range of engineering applications. Examples include floating oil platforms, wave energy conversion devices, and offshore wind turbines. Numerical simulation of such problems is often the only viable tool for elucidating the underlying physics and optimizing structural designs. In spite of significant computational advances over the last decades, however, the numerical simulation of floating structures of engineering interest continues to pose major challenge. Modeling the coupled interaction of geometrically complex moving structures with a single phase flow field is already a difficult task that requires integrating numerical

* Corresponding author.
E-mail address: fotis@umn.edu (F. Sotiropoulos).

http://dx.doi.org/10.1016/j.jcp.2014.08.010
0021-9991/

techniques capable of coupling the fluid and structural domains in a robust and efficient manner. In the case of floating structures, the difficulty of the problem is exacerbated by the inherent two-phase nature of such flows and the non-linear dynamics of the free surface interface dominated by the role of complex phenomena such as turbulence, free surface effects, wave breaking and structure overtopping.

For the most part, previous studies have not considered all the physical aspects required for studying such a complicated problem. Some of these studies treat the problem as a single-phase flow, i.e. instead of solving for the two phases, only the water side is computed while the effect of the air is introduced by satisfying the kinematic and dynamic boundary conditions on the free surface (see for example [1–3]). Reducing the problem to single-phase flow, and thus neglecting the effect that the air has on the motion of the floating structure, may be a reasonable simplification in specific applications. It is important to consider, however, that important underlying mechanisms of the coupled air/water flow cannot be accounted for by such numerical methods. For example, Iafrati et al. [4] demonstrated that the dissipation of most of the energy induced by a breaking wave occurs via complex vortex structures that develop in the air phase. In addition, in applications such as the study of floating wind turbines being able to resolve the two phases of the flow in a coupled manner is a critical requisite of the modeling approach.

The classic approach for simulating flows around moving bodies is the so-called Arbitrary Lagrangian–Eulerian (ALE), in which the mesh conforms to the moving boundary/interface at all time, and consequently boundary conditions on the interface can be satisfied precisely. Examples of applications are given in [5], for dealing with the solid–fluid interface of a floating structure, and in [6,7] for tracking the free surface interface. The ALE method, however, is not practical for solving situations where the boundary and/or interface is arbitrarily complex, and/or undergoes large deformations. In the particular case that the interface is the free surface the applicability of the ALE approach is limited to problems for which the free surface remains smooth and continuous.

The approach that circumvents the limitations of the ALE methods in such complex situations is by using immersed boundary (IB) methods. IB methods were originally introduced by Peskin [8] to study the blood flow in the heart. In the IB methods the flow equations are solved on a fixed, non-boundary-conforming grid and the effect of the boundary/interface is accounted for by introducing a fictitious force field (see Sotiropoulos and Yang [9] and Mittal and Iaccarino [10] for a review of the different IB methods). A major difficulty of IB methods that is particularly critical in FSI applications arise from the approach used to impose the boundary conditions between the fluid domain, represented by the Eulerian mesh, and the solid body which is typically tracked with a Lagrangian mesh. In the classical IB approach of Peskin the force field is calculated from constitutive laws and imposed on the background grid by employing a discrete delta function. The displacements of the structure follow the motion of the surrounding fluid. In other variations of the IB method such as the hybrid Cartesian/immersed boundary method (HCIB) proposed by Mohd-Yusof [11], the fictitious force added to account for the effect of the interface is not computed explicitly but is introduced implicitly by imposing velocity boundary conditions at the grid nodes located in the vicinity of the interface (referred to as IB nodes). To then solve the equations of motion of the moving body, the forces and moments that the fluid exerts to the structure need to be computed by integrating the pressure and shear stresses either directly at the background mesh or at the body surface via extrapolation. In the cut-cell method proposed by Clarke [12] the cells near the boundary are modified so that they conform to its geometry and the velocity boundary condition can be imposed with high accuracy. However, similar to the HCIB, the forces of the fluid acting on the body need to be calculated with special treatments.

The most straightforward technique for computing the forces and moments acting on the structure is by projecting the pressure and shear stresses to all the elements of the Lagrangian mesh of the body and perform a subsequent integration along the surface. Such an approach has been widely applied in single-phase flow problems [13–15] as well as in FSI applications with floating structures [1]. In [15] the force at each material point of the structural mesh was calculated by applying inversed distance weight coefficients to the stresses of the nearest IB nodes. Alternatively, Haeri and Shrimpton [16] proposed a method in which two equally spaced auxiliary points located in the wall normal line centered on each material element are defined. In these auxiliary points the stress tensor is reconstructed by interpolating from eight surrounding fluid cells. Finally, the auxiliary points are used to extrapolate with second order accuracy the values of the stress tensor at the center of each Lagrangian element.

The computational cost of the aforementioned projection techniques is considerable and implementation of such algorithms in parallel computing is not straightforward. Several authors have proposed alternative methods to simplify the calculation of the forces and torques. For example, Balaras [17] proposed a method applicable to moving bodies based on the idea of Lai and Peskin [18] in which the stresses are integrated directly at the underlying fluid mesh along a rectangular bounding box. With such an approach there is no need for performing any projection step and the parallelization of the algorithm is straightforward. However, the force calculation expression contains a term that involves integration within the domain inside the bounding box and exterior to the body. This term becomes difficult to compute for geometrically complex bodies and/or moving bodies. Shen et al. [19] extended the method of Balaras to ease the implementation difficulties in complex and moving bodies by splitting the force in two parts, one representing the flow external to the bounding box, and a second representing the virtual flow inside the domain of the body. In the work of Sanders et al. [20] the solid domain is filled with a fictitious fluid that is forced to move at the same velocity as the solid body. The forcing terms to be applied in the equation of motion are obtained by integration within the interior of the solid domain. Borazjani et al. [21] proposed an approach that is also based on integrating the stresses directly at the Eulerian fluid mesh (details of the algorithm are