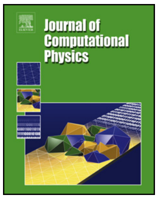
Journal of Computational Physics 355 (2018) 144–175

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp



Fluid–structure interaction simulation of floating structures interacting with complex, large-scale ocean waves and atmospheric turbulence with application to floating offshore wind turbines

Antoni Calderer [a], Xin Guo [a], Lian Shen [a,b], Fotis Sotiropoulos [c,*]

[a] Saint Anthony Falls Laboratory, University of Minnesota, 2 Third Avenue SE, Minneapolis, MN 55414, United States
[b] Department of Mechanical Engineering, University of Minnesota, Minneapolis, MN 55455, United States
[c] Department of Civil Engineering, College of Engineering and Applied Sciences, Stony Brook University, Stony Brook, NY, United States

ARTICLE INFO

Article history:
Received 1 April 2016
Received in revised form 4 August 2017
Accepted 5 November 2017
Available online 10 November 2017

Keywords:
Fluid–structure interaction
Two-phase free surface flow
Large-eddy simulation
Level set method
Wind
Wave

ABSTRACT

We develop a numerical method for simulating coupled interactions of complex floating structures with large-scale ocean waves and atmospheric turbulence. We employ an efficient large-scale model to develop offshore wind and wave environmental conditions, which are then incorporated into a high resolution two-phase flow solver with fluid–structure interaction (FSI). The large-scale wind–wave interaction model is based on a two-fluid dynamically-coupled approach that employs a high-order spectral method for simulating the water motion and a viscous solver with undulatory boundaries for the air motion. The two-phase flow FSI solver is based on the level set method and is capable of simulating the coupled dynamic interaction of arbitrarily complex bodies with airflow and waves. The large-scale wave field solver is coupled with the near-field FSI solver with a one-way coupling approach by feeding into the latter waves via a pressure-forcing method combined with the level set method. We validate the model for both simple wave trains and three-dimensional directional waves and compare the results with experimental and theoretical solutions. Finally, we demonstrate the capabilities of the new computational framework by carrying out large-eddy simulation of a floating offshore wind turbine interacting with realistic ocean wind and waves.

1. Introduction

The potential seen in the ocean as an enormous supply of clean energy resource has motivated an increased attention in the scientific community towards fluid–structure interaction (FSI) problems involving waves and complex floating structures, such as wave energy converter (WEC) devices and offshore wind turbines, as further extension of previous studies on ship hydrodynamics and floating platforms in the petroleum industry. Such types of problems have generally been studied using simplified models assuming inviscid and irrotational flows as in [1–5] dealing with floating wind turbines, or in [6–8] applied to the study of WECs. Potential flow models are accurate for simulating problems with low amplitude motions

---

[*] Corresponding author.
E-mail address: fotis.sotiropoulos@stonybrook.edu (F. Sotiropoulos).

https://doi.org/10.1016/j.jcp.2017.11.006
0021-9991/

and simplified non-breaking waves. However, offshore environments are often subject to more complex non-linear wave phenomena such as turbulence, wave–turbulence interaction, and wave overturning and breaking. Such cases in which viscosity plays an important role can be accurately represented by a Navier–Stokes solver in combination with turbulence models.

One of the main challenges in applying Navier–Stokes solvers to ocean wave problems is to deal with processes occurring at a disparate range of scales. The study of the interaction of a floating structure with swells is an example. While swells interact with the wind flow and evolve for long distances requiring a large computational domain, the floating structure highly depends on flow motions at much smaller scales in the vicinity of the structure and requires the use of very fine meshes. One can reduce the high computational cost of viscous solvers and deal with multi-scale problems more efficiently by using a domain decomposition approach, limiting the application of the viscous solver to regions where the complexity of the flow requires it, and take advantage of other methods, such as the potential flow theory for treating the wave motion in the regions far away. Iafrati and Campana [9] used a domain decomposition approach to simulate two-dimensional (2D) breaking water waves. In their work, a two phase Navier–Stokes solver based on the level set method was applied in the upper part of the domain, containing the free surface, where wave breaking occurs and viscous effects are most important, and an inviscid flow model was used in the lower part of the computational domain far from the free surface. In a later work, Colicchio et al. [10] developed another domain decomposition approach coupling also a level set based, two-phase, viscous flow solver, applied in a region where complex processes occur, with a potential flow solver applied in the regions with mild flow conditions. Both the methods of Iagrati et al. [9] and Colicchio et al. [10] are formulated in a three-dimensional (3D) context but have only been applied to 2D problems.

A key aspect for developing such multi-scale methods is to choose an appropriate technique for transferring the far-field flow solution as input to the near-field solver. In particular, a major challenge in this regard is the approach for prescribing a specific large-scale wave environment as input into the 3D Navier–Stokes flow solver. The simplest and most obvious way is by directly specifying at the inlet boundary the velocity profile and surface elevation. For example, in the work of Colicchio et al. [10], an algorithm for coupling a potential flow based boundary element method (BEM) solver with a Navier–Stokes level set solver is presented. In Repalle et al. [11], theoretical wave velocities are fed into a Reynolds-averaged Navier–Stokes (RANS) model to simulate the wave run-up on a spar cylinder. In Xie et al. [12] waves from a potential flow solver are directly prescribed as boundary conditions for a Navier–Stokes solver. In Christensen [13], waves from a Boussinesq based model are incorporated to a Navier–Stokes solver. Generation of waves by specifying inlet boundary conditions can be problematic when strong reflected waves reach the inlet boundary. For example, Wei and Kirby [14] demonstrated that, even if a generating–absorbing boundary condition is employed, large errors can accumulate and lead to inaccurate solutions after computation for long duration.

Another challenge in simulating ocean wave flows is that when the far-field potential flow solution is fed to the near-field viscous solver directly, both the governing equations and free-surface boundary conditions are unsatisfied. Waves at high frequency would be generated. Due to the finite simulation domain, such high frequency waves would reflect or reenter at the computational domain boundary and become residual standing waves. Dommermuth [15] and Mui and Dommermuth [16] proposed a velocity decomposition method, which decomposes the water velocity field into potential motions and rotational motions. The potential and rotational motions are advanced in time according to their corresponding governing equations. By gradually applying a pressure corresponding to the rotational motions at the free surface, they reduced the high frequency waves. Note that the velocity decomposition method is computationally expensive because two sets of velocity (i.e., potential and rotational motions) need to be simulated.

An approach that can avoid the aforementioned difficulties when dealing with wave reflections is to employ an internal wave maker in combination with the use of sponge layers at the boundaries. The basic idea of internal wave generators is to apply an oscillatory force within an internal region of the domain, known as the source region. The force is introduced by adding a source/sink term either in the continuity or momentum equations. The internal wave generation method based on a mass source/sink was proposed by Lin and Liu [17]. They derived source function expressions based on the fact that the increase/decrease of mass in the source region contributes to the target wave generation. Given a submerged rectangular source region, their method was used to obtain expressions for the following wave cases: linear monochromatic waves, irregular waves, Stokes waves, cnoidal waves, and solitary waves. They demonstrated the accuracy of the method by comparing the results to analytical solutions. They also showed that the internal wave generator was not affected by the presence of reflected waves. Although this method has been widely used by many authors [18–20] for generating 2D waves, it has not been extended to the generation of 3D directional waves. A step further in the development of internal wave makers is to implement the source terms, not in the continuity equation but in the momentum equations. It is not obvious, however, how to derive a forcing term expression that can generate a free surface wave pattern with the specific target amplitude, because the free surface elevation is not a variable in the momentum equation. Such relation, however, can be directly established in exact form in the depth-integrated Boussinesq equations as shown by Wei et al. [21]. In their work [21], source terms for the momentum equations were proposed for generating regular and irregular waves. The idea of Wei et al. [21] was later implemented by Choi and Yoon [22] in a RANS turbulence model and by Ha et al. [23] in a large-eddy simulation (LES) model. In particular, the capabilities of the method to generate directional waves in a 3D basin were successfully demonstrated. All the above internal wave makers employ a fixed rectangular domain as source region that is located under the free surface. An alternative momentum source method is that of Guo and Shen [24], in which the source region is not fixed but follows the motion of the free surface. This is equivalent of applying a surface pressure on the