Ocean Engineering 152 (2018) 257–272

Contents lists available at ScienceDirect

Ocean Engineering

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

Improved numerical wave generation for modelling ocean and coastal engineering problems

Pedro J. Martínez-Ferrer [a,*], Ling Qian [a], Zhihua Ma [a], Derek M. Causon [a], Clive G. Mingham [a]

[a] Centre for Mathematical Modelling and Flow Analysis, Manchester Metropolitan University, Chester Street, Manchester, M1 5GD, United Kingdom

ARTICLE INFO

Keywords:
Wave generation
Wave structure interaction
Floating bodies
Coupling

ABSTRACT

We introduce a dynamic-boundary numerical wave generation procedure developed for wave structure interaction (WSI) simulations typical of ocean and coastal engineering problems. This implementation relies on a dynamic mesh which deforms in order to replicate the motion of the wave-maker, and it is integrated in wsiFoam: a multi-region coupling strategy applied to two-phase Navier-Stokes solvers developed in our previous work [Martínez Ferrer et al. A multi-region coupling scheme for compressible and incompressible flow solvers for two-phase flow in a numerical wave tank. Computer & Fluids 125 (2016) 116–129]. The combination of the dynamic-boundary method with a multi-region mesh counteracts the increase in computational cost, which is intrinsic to simulations featuring dynamic domains. This approach results in a high performance computing wave generation strategy that can be utilised in a numerical wave tank to carry out accurate and efficient simulations of wave generation, propagation and interaction with fixed structures and floating bodies.

We conduct a series of benchmarks to verify the implementation of this wave generation method and the capabilities of the solver wsiFoam to deal with wave structure interaction problems. These benchmarks include regular and focused waves, wave interaction with a floating body and the modelling of a wave energy converter, using different wave-maker geometries: piston, flap and plunger. The results gathered in this work agree well with experimental data measured in the laboratory and other numerical simulations.

1. Introduction

Numerical wave tanks (NWTs) constitute an essential tool for design and analysis in ocean and coastal engineering problems. NWTs must be validated against experiments conducted in the laboratory, from wave generation to the evaluation of wave impacts on fixed or floating objects, offshore structures, performance and survivability of wave energy converters, etc. The main objectives of a NWT are to complement experiments, e.g. retrieving useful data which otherwise would be difficult to measure experimentally, to simulate full scale geometries in open and real sea state conditions, or even to explore and assess new designs of coastal defence systems, offshore platforms and marine vessels. Computational Fluid Dynamics (CFD) has been extensively used in NWTs with a large variety of simplified and detailed models depending on the degree of physics required or the computational resources available. In general, fully non-linear potential flow models (Grilli and Horrillo, 1997; Ma and Yan, 2006) have been widely adopted due to the simplicity of their equations and the good accuracy achieved on wave propagation. With the increasing power and efficiency of computational resources and the development of high performance computing (HPC), Navier-Stokes models in NWTs are experiencing a growing demand (Lubin et al., 2006; Higuera et al., 2013a) because they allow for a detailed analysis of the flow physics accounting for vorticity, viscosity and air entrainment/entrapment effects, at the expense of higher computational costs. Moreover, special attention has been given recently to compressibility effects in water-air mixtures characteristic of violent wave impacts (Peregrine et al., 2006; Bredmose et al., 2009; Lugni et al., 2010), which are simulated with expensive numerical methods such as compressible smooth particle hydrodynamics (SPH) (Colagrossi and Landrini, 2003; Guilcher et al., 2013) and the volume of fluid (VOF) method (Ma et al., 2014, 2015).

Carrying out very detailed simulations, e.g. based on the resolution of the Navier-Stokes equations, in the entire computational domain, which can feature an extension of several hundreds of metres in typical ocean engineering problems, remains impractical today even with the current state of the art in HPC, e.g. parallel heterogeneous computing (CPU + GPU). Therefore, coupled simulations in which specialised numerical solvers work in different regions of the NWT become a good strategy to

* Corresponding author. Now at Barcelona Supercomputing Center, C/ Jordi Girona 31, 08034, Barcelona, Spain.
E-mail address: pedro.martinez-ferrer@bsc.es (P.J. Martínez-Ferrer).

https://doi.org/10.1016/j.oceaneng.2018.01.052
Received 3 October 2016; Received in revised form 20 September 2017; Accepted 10 January 2018

overcome the current challenges in the numerical modelling of ocean and coastal engineering problems (Martínez Ferrer et al., 2016a). Such a *multi-region* wave tank may be principally composed of: (i) a relatively quick fully non-linear potential (FNLP) solver for wave propagation in large extensions of the mesh, (ii) incompressible and compressible Navier-Stokes solvers to study rotational, viscous and complex flows, with associated effects of compressibility in some cases, and (iii) a computational structural dynamics (CSD) solver for the WSI on rigid and deforming bodies. There have been some efforts in this direction and, more specifically, in the coupling between irrotational and viscous flows, see for instance references (Iafrati and Campana, 2003; Sriram et al., 2012; Zhang et al., 2013). However, the diversity in the coupling solutions and the increasing complexity in numerical modelling have prevented these coupled strategies from becoming popular, specially in HPC where the domain decomposition needs to be properly handled between different regions (Schluter et al., 2005). A recent work on multi-region coupling for incompressible and compressible two-phase flow solvers in a numerical wave tank has been proposed in (Martínez Ferrer et al., 2016a), where the coupling between regions is treated as another boundary condition to simplify the numerical modelling and facilitate programming on HPC architectures.

One of the essential components of a wave tank is the realistic generation of waves. In the laboratory, the *physical modelling* aims to replicate waves found in nature by using different wave generating mechanisms (Svendsen, 1985). Thus, wave generators are generally classified in three main categories: (i) pistons, for the physical modelling of shallow water waves, (ii) flaps and (iii) plungers utilised for deeper water waves. The relation between the wave generator motions and the dynamics of the generated waves have been studied, both theoretically and experimentally, using first-order linearized hydrodynamic equations for pistons and flaps (Biesel and Suquet, 1954; Ursell et al., 1960) as well as plungers (Hyun, 1976), see also (Dean and Dalrymple, 1991). More recently, linear theory was revisited with a fully second-order wave--maker theory to correctly reproduce in the laboratory the lower and higher harmonic wave components found in irregular sea states (Schaffer, 1996). On the other hand, the *numerical modelling* of waves in NWTs has been traditionally limited to static-boundary wave generation, where the velocity and free surface wave profile are specified at the boundaries, e.g. Dirichlet or Neumann boundary conditions, based on different wave theories for regular and irregular types of wave (Le Mehaute, 1976). Static-boundary wave generation methods have been applied to both potential (Wei and Kirby, 1995; Mehmood et al., 2016) and Navier-Stokes flows, see for instance (Jacobsen et al., 2012; Higuera et al., 2013b). These methods are relatively easy to implement and offer an attractive computational cost as they do not require moving mesh components for numerical wave generation. On the other hand, wave generation methods based on dynamic-boundaries can replicate the exact motion of the paddles used in the experiments and, consequently, close the gap between the physical modelling in the laboratory and the numerical modelling in NWTs. Dynamic-boundary methods are more popular among potential flows solved with the boundary element method (BEM) (Ma and Yan, 2006; Grilli et al., 2002) but they are not commonly found in Navier-Stokes Eulerian flow solvers (Henry et al., 2014), as the introduction of a deforming mesh to accommodate the motion of the paddles contributes to a significant increase in the computational cost, which is already elevated in Navier-Stokes solvers. Another recent example can be found in the literature (Higuera et al., 2015), where a piston-type wave-maker was implemented within the VOF method. However, it was found that the compelling increase in terms of computational cost compared to static-boundary methods questioned the suitability of such methods for large scale applications.

The aforementioned low efficiency problem related to dynamic-boundary methods needs to be carefully addressed through further development, as this approach proves to be necessary in cases dealing with confined geometries susceptible of wave reflection (Henry et al., 2014), or in cases where static-boundary methods cannot provide accurate predictions (Yan et al., 2015). The aim of this paper is therefore to develop an efficient and versatile dynamic-boundary numerical wave generation method in a multi-region NWT in order to replicate the most common wave-makers found in laboratories, including pistons, flaps as well as plungers. Special emphasis is given in the comparison of this approach against other well established static-boundary numerical wave generation methods not only in terms of accuracy, but also and more specifically in terms of computational efficiency. The rest of the paper is organised as follows: Section 2 describes the two-phase incompressible Navier-Stokes solver used to carry out our numerical investigations, the dynamic-boundary wave-maker implementation and its integration in a multi-region computational domain. Results and discussions are presented in Section 3 and Section 4 is dedicated to conclusions and future work.

2. Numerical procedures

We utilise the open-source CFD library OpenFOAM (Jasak, 1996) to carry out the simulations presented in this work. OpenFOAM numerical solvers rely on a cell-centered, co-located finite-volume method. This library is widely employed in research and industry and it offers the possibility to read, improve and modify the available code for free.

In this paper we present a new dynamic mesh algorithm which mimics a physical wave-maker such as those employed in experimental wave tanks. We apply this algorithm in conjunction with a novel multi-region coupling strategy presented in (Martínez Ferrer et al., 2016a), namely “wsiFoam”, in order to increase the efficiency of the method. A description of the solvers, the dynamic mesh and the coupling strategy is detailed below.

2.1. *The numerical solver*

The simulations performed in this work are conducted with a slightly modified version of “interFoam”, which is an incompressible two-phase pressure-based solver (Rusche, 2002) that has already been successfully applied in a wide variety of naval and ocean engineering applications, see for instance (Higuera et al., 2013a; Ma et al., 2016). It is based on the VOF method to describe an incompressible two-phase flow mixture, i.e. air and water, wherein each phase is assumed to be homogeneous and in mechanical equilibrium: identical velocity and pressure. Furthermore, this solver makes special emphasis on maintaining a sharp water free surface (interface-capturing) by using artificial compression terms.

The mass balance equation for the incompressible ($\nabla\cdot\mathbf{U} = 0$) two-phase flow mixture can be reduced to the mass balance equation for the water volume fraction $\alpha \in [0, 1]$:

$$\frac{\partial \alpha}{\partial t} + \nabla\cdot\mathbf{U}\alpha + \nabla\cdot\mathbf{U}_c\alpha(1-\alpha) = 0, \tag{1}$$

where $\mathbf{U}$ is the mixture velocity vector and $\mathbf{U}_c = \min[\mathbf{U}, \max(\mathbf{U})]$. Herein the density of the mixture is $\rho = \alpha\rho_w + (1-\alpha)\rho_a$ with constant partial densities $\rho_w = 1000\ \mathrm{kg/m^3}$ and $\rho_a = 1.1586\ \mathrm{kg/m^3}$. The third term in eq. (1) is an artificial compression quantity that sharpens the interface and guarantees bounded values of α by using the MULES procedure (Rusche, 2002; Weller, 2002).

The single momentum equation for the homogeneous mixture is given by

$$\frac{\partial \rho\mathbf{U}}{\partial t} + \nabla\cdot(\rho\mathbf{U}\mathbf{U}) - \nabla\cdot(\mu\nabla\mathbf{U}) = \sigma\kappa\nabla\alpha - \mathbf{g}\cdot\mathbf{x}\nabla\rho - \nabla p_d, \tag{2}$$

where σ denotes the surface tension coefficient and $\kappa = \nabla\cdot(\nabla\alpha/|\nabla\alpha|)$ represents the curvature of the interface. The mixture viscosity is given by $\mu = \alpha\mu_w + (1-\alpha)\mu_a$. The dynamic pressure is calculated as $p_d = p - \rho\mathbf{g}\cdot\mathbf{x}$ with $\mathbf{g}$ and $\mathbf{x}$ the gravity and position vectors, respectively.

The governing equations (1) and (2) are linearized and integrated