Journal of Fluids and Structures 65 (2016) 155–179

Contents lists available at ScienceDirect

Journal of Fluids and Structures

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

Two-phase SPH simulation of fluid–structure interactions

Kai Gong [a], Songdong Shao [b,e,f], Hua Liu [c,*], Benlong Wang [c], Soon-Keat Tan [d]

[a] Tropical Marine Science Institute, National University of Singapore, 18 Kent Ridge Road, 119227, Singapore
[b] Department of Civil and Structural Engineering, University of Sheffield, Sheffield S1 3JD, United Kingdom
[c] Department of Engineering Mechanics, MOE Key Laboratory of Hydrodynamics, Shanghai Jiao Tong University, Shanghai 200240, China
[d] NEWRI and Maritime Research Centre, School of Civil and Environmental Engineering, Nanyang Technological University, Nanyang Avenue, 639798, Singapore
[e] State Key Laboratory of Hydro-Science and Engineering, Tsinghua University, Beijing 100084, China
[f] School of Water Resources and Electric Power, Qinghai University, Xining 810016, China

ARTICLE INFO

Article history:
Received 8 May 2015
Received in revised form
15 May 2016
Accepted 22 May 2016
Available online 8 June 2016

Keywords:
SPH
Two-phase
Solid boundary
Fluid–structure interaction
Air cavity
Wedge entry

ABSTRACT

In this paper, a two-phase Smoothed Particle Hydrodynamics (SPH) method is used to simulate the fluid–structure interactions with violent deformation of the free surface. An improved solid boundary treatment is proposed based on the accurate pressure interpolations of the inner fluid particles. The model performance is validated by sloshing in a water tank and dam break flow impact on a vertical wall. In the practical model applications, a two-dimensional wedge entry into the static water is studied, for which the flow fields of water and air phases are computed simultaneously. It has been found that both the water flow around the wedge cavity and the air flow inside are reasonably predicted. Also the two-phase model has been found to accurately provide the flow features throughout the entire entry process, while the single-phase model can only predict the flows before the closure of the cavity due to the lack of air modeling. Besides, a laboratory experiment on the wedge entry has also been carried out for the validation purposes.

1. Introduction

Fluid–structure interactions are an important topic in the design and operation of coastal and offshore facilities. The accurate prediction of the flow fields as well as the structure responses could provide useful information to the industry. For example, the water entry is a typical fluid–structure impact problem which occurs in the field of marine hydrodynamics. Following the pioneering work of von Karman, quite a few research studies have been carried out to examine the hydrodynamic features during the entry process, such as the deformation of free surface, evolution of the cavity region, fluid impact force and motion of the falling object (Wang et al., 2015; Wang and Faltinsen, 2013). Early research works were mostly based on the analytical and experimental methods, such as reported by Zhao and Faltinsen (1993).

Since the interactions between the fluid and structure usually lead to the large free surface deformation and wave breaking, the numerical models based on the partial differential hydrodynamic equations, i.e. Navier–Stokes (N–S) equations, demonstrate their unique advantages of being able to provide the detailed flow information without the limitation imposed in the laboratory experiment and field observation. For example, Lin (2007) solved the Reynolds-averaged N–S equations for the study of water entry of a circular cylinder. In recent years, the mesh-free Smoothed Particle Hydrodynamics (referred as SPH)

---

* Corresponding author.
E-mail addresses: gkfly@hotmail.com (K. Gong), s.shao@sheffield.ac.uk (S. Shao), hliu@sjtu.edu.cn (H. Liu), ctansk@ntu.edu.sg (S.-K. Tan).

http://dx.doi.org/10.1016/j.jfluidstructs.2016.05.012
0889-9746/

method becomes quite attractive in simulating the violent deformation of free surface and the wave breaking. As a result, a variety of fluid–structure interaction problems have been investigated by using either SPH (Gao et al., 2012; Liu et al., 2014; Ren et al., 2014) or other similar mesh-free particle models (Koshizuka et al., 1998; Hwang et al., 2014). SPH was originally developed for the astrophysical computations by Gingold and Monaghan (1977) and later modified to model the fluid flows by Monaghan (1994). It is a pure Lagrangian particle method which does not need the grid to calculate the spatial derivatives. Instead, this is achieved by the analytical differentiation of the interpolation functions. Therefore, the continuity and momentum conservation equations are formulated as a set of ordinary differential equations, and the SPH particle positions and attributes are computed by using the standard numerical integration methods in the time domain. With regard to the water entry problem, there exist quite a few good SPH works including Vandamme et al. (2011), Skillen et al. (2013) and Liu et al. (2014). It has been found that the SPH method is quite promising in simulating these kinds of transient impact problems and very refined flow features can be captured by the mesh-free particle motion. For instance, Oger et al. (2005, 2006) pioneered the SPH application of water entry problems and showed that accurate solid boundary treatment is a key to the realistic simulation of wedge kinematics and dynamics. Besides, Maruzewski et al. (2010) extended the work to a much larger engineering scale using the HPC simulation.

Solid boundary treatment and air-phase modeling should be carefully examined to accurately predict the fluid motion and structural response during the fluid–structure interaction process. Both of them strongly influence the impact pressure and force on the structure and thus on its motion. For the water entry problems, for instance, the air cavity enclosed by the water could significantly affect the flow field and the hydrodynamic loading. The early multi-phase SPH modeling concept was proposed by Monaghan and Kocharyan (1995) for the compressible flows. Later substantial works have been implemented by Hu and Adams (2006, 2007) and Grenier et al. (2009) using more advanced solutions to treat the mass conservation and the interface stability. One promising improvement in the free surface flow simulations we follow here was made by Colagrossi and Landrini (2003), who proposed a straightforward and efficient two-phase model for the large density ratios and tested the model on a variety of benchmark cases. Besides, the robust treatment of solid boundary conditions is also quite important, especially for the boundaries surrounding the moving structure. The repulsive boundary (Monaghan, 1994) provided a simple treatment of the solid wall to prevent particle penetrations, but it could cause the pressure oscillations in the fluid region. The mirroring boundary (Cummins and Rudman, 1999) could be the most accurate one but it was achieved at the expense of the CPU time, and also the numerical program could become heavy for the complex boundary configurations. The so-called dynamic boundary (Gomez-Gesteira and Dalrymple, 2004) provided an alternative way to treat the solid boundary with economic computing time.

In this paper, we will further explore the two-phase water–air modeling concept of Colagrossi and Landrini (2003), and combine it with an improved solid boundary treatment from Gómez-Gesteira and Dalrymple (2004) to study the fluid–structure interactions including the challenging water entry of a wedge based on the study of Oger et al. (2006). In the previous works by the same authors (Gong et al., 2009, 2010), we investigated the early stage of the water entry by using a single-phase SPH model and found that there was almost no difference in the simulation results whether a single-phase or a two-phase model was used. One main objective of the present work is to investigate more in-depth hydrodynamics during the entire wedge entry process with the support of our own laboratory experiment. Prior to this, we will use two benchmark tests to validate the model accuracy, including the sloshing in a water tank and the dam break flow impact on a solid wall with air-cushion effect.

Here it should be noted that our present two-phase SPH model is only applicable to the low-speed entry problems. This is due to that in the more challenging high-speed water entry of 10–20 m/s, the compressibility of water could become significant and the severe cavitations could lead to the complex phase changes, so more advanced SPH models should be developed to account for these effects. Besides, the flow turbulence should be taken into consideration by studying various turbulence closure techniques (Violeau and Issa, 2007). However, most established turbulence models were based on the steady flow state and more investigations should be made to examine their validity in such a transit flow condition like the wedge entry. Also, the Reynolds scaling would provide an elegant approach to address these similar problems and make the relevant connections on different flow scenarios. Finally, we should be aware of a recent group of accurate techniques to treat the solid boundaries based on the boundary integrals, e.g. Kulasegaram et al. (2004) and De Leffe et al. (2009). This kind of approach has proved to be very accurate, although computationally expensive, and has been extended to the frictional forces and turbulence by Ferrand et al. (2010, 2012), to the incompressible SPH (ISPH) by Leroy et al. (2014) and to the thermal fluxes by Leroy et al. (2015). Our present model is less expensive than these boundary integral techniques, but it has the limitations when applied to other kinds of wall boundary condition such as the Neumann.

2. Principles of two-phase SPH model

The governing equations of the fluid dynamics follow the Navier–Stokes (N–S) equations. Assuming the fluid viscosity being not dominant, the shear stresses can be ignored and the following continuity and momentum equations are often used in the SPH representations:

$$\frac{d\rho}{dt} = -\rho\nabla\cdot\mathbf{u} \tag{1}$$

$$\frac{d\mathbf{u}}{dt} = -\frac{1}{\rho}\nabla P + \mathbf{g} \tag{2}$$