Ocean Engineering 123 (2016) 15–27

Contents lists available at ScienceDirect

Ocean Engineering

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

Numerical investigation of air enclosed wave impacts in a depressurised tank

Z.H. Ma *, D.M. Causon, L. Qian, C.G. Mingham, P. Martínez Ferrer

Centre for Mathematical Modelling and Flow Analysis, School of Computing, Mathematics and Digital Technology, Manchester Metropolitan University, Manchester M1 5GD, United Kingdom

ARTICLE INFO

Article history:
Received 15 October 2015
Received in revised form
12 April 2016
Accepted 28 June 2016

Keywords:
Naval engineering
Computational fluid dynamics
Breakwater

ABSTRACT

This paper presents a numerical investigation of a plunging wave impact event in a low-filling depressurised sloshing tank using a compressible multiphase flow model implemented in open-source CFD software. The main focus of this study is on the hydrodynamic loadings that impinge on the vertical wall of the tank. The detailed numerical solutions compare well with experimental results and confirm that an air trapped plunging wave impact causes the vertical wall to experience pulsating pressure loadings in which alternate positive and negative gauge pressures occur in sequence following the first applied pressure peak. The strongest pulsations of the pressure are found to be near the air pocket trapped by the water mass. The instantaneous pressure distribution along the vertical wall is nearly uniform in the area contained by the air pocket. The phases of pulsating pressures on the wall are in synchronisation with the expansion and contraction of the trapped air pocket. The pocket undergoes changes in shape, moves upwards with the water mass and eventually breaks up into small parts. A careful integration of the wall pressure reveals that the vertical structure as a whole experiences pulsating horizontal impact forces. It is found that the average period of pulsation cycles predicted in the present study is around 5–6 ms, and the loading pulsations are quickly damped out in 0.1–0.2 s. Further exploratory investigation of the fluid thermodynamics reveals that the temperature inside the trapped air pocket rises quickly for about 2 ms synchronised with the pocket's first contraction, then the generated heat is rapidly transferred away in around 3 ms.

1. Introduction

Water waves impacting on fixed and floating structures are a major concern in coastal and ocean engineering. Violent wave induced impulsive loadings may damage coastal defence walls, break up oil platforms and dismantle large ships. In oceans, large waves under harsh weather conditions may impinge at the bow of a liquefied natural gas carrier or other marine vessels and cause large forces and high frequency vibration. Green water slamming onto the deck may also cause severe damage. Meanwhile the liquid stored in a tank carried as cargo may undergo correspondingly violent motion resulting in strong wave impacts on the internal structure. The induced forces and moments may in turn cause significant changes in ship motions. These harsh hydrodynamic loadings represent a threat to the safety and stability of the vessel (Bai et al., 2015; Xu and Duan, 2009).

With the fast development of computer technology and numerical analysis, computational fluid dynamics (CFD) has been more frequently used in coastal and ocean engineering to predict wave loadings on structures. Traditionally, potential flow theory is chosen to mathematically describe the dynamics of water waves assuming the flow to be irrotational (Scolan et al., 2014). However, the underlying assumption is not valid when the wave overturns, which introduces strong vorticity into the flow. Therefore new methods based on the one- or two-phase incompressible Navier–Stokes equations have been proposed to deal with viscosity, vorticity and air entrainment/entrapment for wave breaking problems (Lubin et al., 2006; Qian et al., 2006). However fluid compressibility effects, which have a significant influence on the evolution of air trapping plunging waves observed in experiments (Bullock and Obhrai, 2007; Chan et al., 1988; Lugni et al., 2010a,b; Peregrine, 2003), still cannot be properly handled by these incompressible models. This requires a more detailed compressible multiphase flow model that can include all necessary physics to provide an accurate solution (Kapsenberg, 2011).

Recently researchers have started to explore the importance of fluid compressibility in numerical simulations of wave impacts by

* Corresponding author.
E-mail address: z.ma@mmu.ac.uk (Z.H. Ma).

http://dx.doi.org/10.1016/j.oceaneng.2016.06.044
0029-8018/

using the volume of fluid (VOF) method to capture the free surface. Bredmose et al. (2009) and Bredmose et al. (2015) proposed a weakly compressible flow model to compute wave impact events. A density-based numerical method employing an iterative Riemann solver was used to solve the conservative flow model. However benchmark tests of a one-dimensional shock wave passing through a water–air interface exhibited strong non-physical pressure oscillations at the material interface. Plumerault et al. (2012b) proposed a density-based compressible three-fluid model for describing aerated-water wave problems. Nevertheless it is noted that strong non-physical oscillations also arose at the material interface in their water–air shock tube results (Plumerault et al., 2012b). When this model was used to simulate a plunging wave impact problem, non-physical numerical oscillations occurred in the computation (Plumerault et al., 2012a). This caused the impact pressure to keep oscillating without decaying, which is contrary to the phenomenon observed in experiments that the impact loads were quickly damped out (Lugni et al., 2010a).

Bredmose et al. (2009), Bredmose et al. (2015), and Plumerault et al. (2012a,b) did not clarify the underlying reason causing the non-physical numerical oscillations in their computations. However, Ma et al. (2014) have pointed out that the non-physical oscillations are due to the use of fully-conservative flow models in their numerical calculations of multiphase flows. Instead of using a fully-conservative flow model, they proposed a quasi-conservative compressible two-phase flow model building on the conservation laws and a non-conservative advection equation for the volume fraction (Ma et al., 2014). A density-based method adopting a HLLC-type approximate Riemann solver was applied to solve the multiphase flow model. The method exhibited good numerical conservation behaviour and the non-physical pressure oscillations were effectively removed in water-air shock tube tests. The method has also been successfully extended to solve incipient cavitations and underwater explosions (Ma et al., 2015) as well as violent water entry of rigid flat plates under various aeration conditions (Ma et al., 2016). However, difficulties still arose in computing plunging wave impact problems as the pressure did not decay but kept ringing with a relatively large amplitude.

Although an initial study of the reported works of Bredmose et al. (2009), Bredmose et al. (2015), Plumerault et al. (2012a, b), and Ma et al. (2014) gives an impression that these flow models were properly constructed and suitable for wave impact problems, the implausible results for either the water–air shock tubes or plunging wave impact problems implies that some very important aspects of the numerical algorithms remain unresolved in these works. It is noted that standard density-based methods were adopted by all of them to solve wave impact problems no matter what kind of multiphase flow models (fully-conservative or quasi-conservative) were used. Apparently these authors ignored the fact that density-based methods are not suitable to solve nearly incompressible low-speed (low-Mach) flows without applying special treatments to overcome the system stiffness problem.

It is well-known that density-based methods were originally designed for high-speed compressible flows. In this context, the continuity equation is used to obtain the density field while the pressure field is determined from the equation of state. However for low-speed flows, pressure is weakly coupled or even decoupled from density. This causes convergence difficulties for density-based methods. With a non-convergent solution to steady flows, the variation of pressure between successive time steps may well exceed the whole range of the real physical pressures (Kadioglu et al., 2005). It will become even worse for unsteady low-speed flows, which require an accurate computation of the pressure field at every real time step.

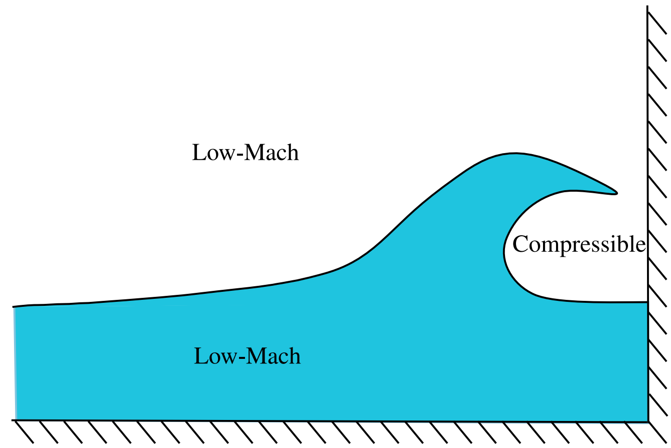

Fig. 1. A sketch of an air-enclosed plunging wave impact at a vertical wall. The flow domain consist of compressible and low-speed (nearly) incompressible regions.

As a matter of fact, the impact of a plunging wave with enclosed air pocket is rather complicated. The flow consists of a small but remarkable compressible region (the air pocket) and a much larger low-speed (low-Mach) incompressible area (the water and air outside the enclosed air pocket) as shown in Fig. 1. To properly handle this kind of problem, numerical methods must closely follow the physics by taking the following important factors into account:

- The flow is not single-phase but multi-phase.
- The flow includes incompressible and compressible regions.
- The enclosed air pocket repeatedly contracts and expands.
- Standard density-based methods are not suitable for low speed flows.

To solve low-speed flows with density-based methods, preconditioning of the governing equations by pre-multiplying time derivatives with a suitable matrix is a popular option to address the system stiffness problem. Unfortunately the resultant accuracy and convergence rate depend upon the choice of artificial parameters. Moreover this becomes extremely complicated especially when multiphase flows are considered (LeMartelot et al., 2013; Kadioglu et al., 2005). To the knowledge of the authors, a density-based conservative method suitable to solve air enclosed wave impacts with proper modifications to improve the low-speed region computation is not available yet. On the other hand, pressure-based methods, which sequentially solve the momentum equations and the pressure correction equation in an iterative process, can properly deal with low-speed incompressible flows. These methods have been successfully extended to incorporate compressibility for single-phase aerodynamic flows (van der Heul et al., 2003; Shterev and Stefanov, 2010; Chen and Przekwas, 2010) and multi-phase cavitating flows (Senocak and Shyy, 2002).

The objective of the present work is to investigate air-enclosed plunging wave impact loads on structures with an appropriate numerical method. This includes the following important aspects:

- the evolution of the free surface and enclosed air pocket,
- air cushion effects on the spatial distribution of pressure loadings,