Computers & Fluids 114 (2015) 218–231

Contents lists available at ScienceDirect

Computers & Fluids

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

Numerical simulation of hydrodynamic wave loading by a compressible two-phase flow method

Rik Wemmenhove [a,1], Roel Luppes [a], Arthur E.P. Veldman [a,*], Tim Bunnik [b]

[a] Institute for Mathematics and Computer Science, University of Groningen, P.O. Box 407, 9700 AK Groningen, The Netherlands
[b] MARIN, P.O. Box 28, 6700 AA Wageningen, The Netherlands

ARTICLE INFO

Article history:
Received 3 January 2014
Received in revised form 3 January 2015
Accepted 11 March 2015
Available online 20 March 2015

Keywords:
CFD
Sloshing
Wave loading
Two-phase flow
VOF-method
Spurious velocities

ABSTRACT

Hydrodynamic wave loading on and in offshore structures is studied by carrying out numerical simulations. Particular attention is paid to complex hydrodynamic phenomena such as wave breaking and air entrapment. The applied CFD method, ComFLOW, solves the Navier–Stokes equations with an improved Volume-of-Fluid method to track the movement of the free surface. A local height function keeps the surface sharp (no 'flotsam and jetsam'). Application of two different fluid models, single-phase (only liquid) and two-phase (liquid and compressible gas) is presented, the latter model being capable of simulating bubbles of entrapped gas.

Treatment of the density around the free surface is found highly critical for obtaining an accurate fluid distribution and velocity field. A newly-developed gravity-consistent density averaging method is applied to prevent spurious velocities around the free surface. The convective terms are approximated by a compressible, symmetry-preserving second-order upwind discretization. Time integration, using second-order Adams–Bashforth, is carried out with a generalization of the familiar pressure-correction method, in which the full acoustical part of the flow equations is treated implicitly.

Numerical results are validated against experimental data for two test cases. As an example of internal wave loading, liquid sloshing dynamics are validated with experimental results for a 1:10 scale LNG tank section. In particular, the experimental pressure signal during a moment of air entrapment is compared with one-phase and two-phase flow simulations. The simulation of external wave loading is validated with data from an experiment with wave run-up against a 1:50 scale semi-submersible offshore structure. The test cases show that modeling of two-phase effects can be beneficial for simulating hydrodynamic wave loading.

1. Introduction

Offshore environments are known for their large variability in wind, waves and currents. Ships and offshore structures should be able to withstand environmental loads during violent weather conditions, even in case of steep, extreme, waves [13,18]. Awareness of the magnitude of these loads is important during design and operation [4]. In order to estimate wave loads on offshore structures by means of a numerical method, both the wave field and the structural geometry have to be modeled accurately and with sufficient detail. Linear methods, based on potential flow theory, to analyze the indicated extreme events are not capable of predicting wave loads to an acceptable level of accuracy. Especially near the objects studied, the physical phenomena accompanying these events are highly non-linear in relation to the occurring wave elevations, and require more complete models as a basis for describing wave dynamics and loads. The review paper [42] and the monograph [11] give well-balanced descriptions of the strengths and limitations of linear flow models.

Furthermore, in complex cases, modeling the dynamics of both water and the surrounding compressible air is helpful to estimate loads more accurately. The dynamics of the water and compressible air phases are well visible for complex free surface problems like green water loading, slamming and tank sloshing. In particular, around the free surface a complex interaction between water and air may occur. Spray, consisting of many small droplets, is observed above the free liquid surface, while clouds of small bubbles (entrained air) are present just below it. Wave overtopping, either due to wave steepness or the presence of an offshore structure, may lead to the entrapment of a much larger air pocket. Air pockets can have a cushioning effect on peak pressure levels during wave

* Corresponding author. Tel.: +31 503633939; fax: +31 503633800.
E-mail addresses: rwemmenhove@slb.com (R. Wemmenhove), r.luppes@rug.nl (R. Luppes), a.e.p.veldman@rug.nl (A.E.P. Veldman), t.bunnik@marin.nl (T. Bunnik).
[1] Present address: Schlumberger, P.O. Box 234, 1372 Asker, Norway.

http://dx.doi.org/10.1016/j.compfluid.2015.03.007
0045-7930/

impacts [33]. The size of these air pockets varies greatly, but they generally have a short lifetime [7]. The life cycle of air pockets is also influenced by aeration of the water, making the physics more complex with even smaller length scales and compressibility effects. This part of the physics lies outside the scope of the present paper: the water phase will be considered strictly incompressible. Also capillary effects from surface tension are not considered, but can be added if desired [30].

The entrapment of air pockets and the entrainment of bubbles is not only important for wave loading on ships and offshore structures (external wave loading), but also for internal wave loading cases, such as the fluid motion in fuel tanks or anti-roll tanks on board of a ship. Model test campaigns to predict the air–water interaction during hydrodynamic wave loading are rather costly and time-consuming. Therefore, there is a great need for numerical simulation tools that can predict the impact loading on and the flow around offshore structures during hydrodynamic wave loading.

To model two-phase flow effects for wave-type problems, a number of choices have to be made with respect to the numerical method. The description of the fluid flow of both phases is based on the Navier–Stokes equations. These equations can be applied to both phases separately, but in the current method the liquid and gas phase are described as one aggregated fluid with varying properties. Another important aspect is the choice of the computational grid. The grid should be constructed such that the free surface is described as 'sharp' as possible in order to model its dynamical behavior sufficiently accurate.

Instead of using a Cartesian grid, as in the present method, another option would be to construct an unstructured grid using a Lagrangian approach. However, aligning the grid with a moving interface results in a less transparent grid and is very difficult for highly distorted and rapidly moving free surfaces, as is the case in many offshore problems. An alternative for grid-based methods could be the use of Smoothed Particle Hydrodynamics (SPH) and related methods, e.g. [12,16,24,31]. These meshless methods put a large number of particles in the flow, each with their own mass and velocity [21]. They are however computationally expensive and lead to a less accurate pressure prediction, which is a major drawback when forces on offshore structures have to be computed.

In the method described in this paper, the Navier–Stokes equations are solved for compressible two-phase flow with an incompressible liquid phase and a compressible air phase. This occurs in such a way that the free surface is kept sharp even for violent flow conditions. The description of the interface is based on the Volume of Fluid (VOF) method, which has been introduced by Hirt and Nichols [19], extended with a local height function for improved accuracy [26,43]. The improved Volume of Fluid (iVOF) method is able to keep the interface sharp, while it allows use of a rather coarse grid to limit computation times.

Especially on staggered computational grids, two-phase flow models can suffer from serious errors near the free surface in the form of so-called spurious (or parasitic) velocities, see e.g. [14,17,38,39]. The cure is often sought in a more accurate treatment of reconstruction and/or advection of the free surface. In the present paper, however, we focus on the discrete compatibility between the pressure gradient, the density and the gravity force. A 'gravity-consistent' averaging of the density between cell centers and cell faces is found to play an essential role.

Generalizing the pressure Poisson equation for incompressible flow, in our compressible flow model the acoustical part is treated in an implicit way. The continuity equation (with the density implicitly coupled to the pressure) and the momentum equation are combined into a hyperbolic wave-type equation from which the pressure can be calculated. The convective terms are discretized with a compressible symmetry-preserving upwind method [20,44,49].

The numerical method, called ComFLOW, has been developed initially to simulate one-phase flow. Earlier applications were in the simulation of sloshing on board spacecraft [15,43,47], in medical science [28,29] and in μ-gravity biology [32]. Currently, the method is used to solve engineering problems in the maritime and offshore industry, e.g. [3,6,22,23,25–27]. Extensive validation with model experiments that are relevant for the offshore industry is an important aspect in the development of the numerical method. An overview of the current status of ComFLOW can be found in e.g. [45,46] and from the ComFLOW website [5] (containing also some experimental data).

The basics of the numerical method are presented first, after which attention is paid to the numerical refinements around the free surface. The method is validated with the results of two series of model experiments. The sloshing fluid motion inside a partially-filled LNG (liquefied natural gas) tank has been measured to validate the fluid flow inside closed domains. To test the simulation of impact phenomena and wave propagation, experiments concerning wave run-up against a semi-submersible offshore structure have been carried out. For the latter flow case an extensive assessment of numerical and experimental uncertainty is included.

2. Mathematical model

2.1. Aggregated governing equations

The flow of two phases is described as the flow of one aggregated fluid with varying properties, which can be described by one continuity and one momentum equation. This approach leads to a smooth velocity field around the free liquid surface [41]. Mass conservation is applied on an arbitrary part Ω of the flow domain with boundary S and an outward directed normal vector $\mathbf{n}$:

$$\int_\Omega \frac{\partial \rho}{\partial t}\,\mathrm{d}\Omega + \oint_S (\rho \mathbf{u})\cdot \mathbf{n}\,\mathrm{d}S = 0, \tag{1}$$

with $\mathbf{u} = (u, v, w)^T$ the velocity, ρ the density and t the time. Momentum conservation is given by

$$\int_\Omega \frac{\partial(\rho\mathbf{u})}{\partial t}\,\mathrm{d}\Omega + \oint_S (\rho\mathbf{u}\cdot\mathbf{n})\mathbf{u}\,\mathrm{d}S + \oint_S p\mathbf{n}\,\mathrm{d}S - \oint_S \{\mu(\nabla\mathbf{u} + \nabla\mathbf{u}^T) - \frac{2}{3}\mu(\nabla\cdot\mathbf{u})\mathbf{I}\}\cdot\mathbf{n}\,\mathrm{d}S - \int_\Omega \rho\mathbf{F}\,\mathrm{d}\Omega = 0, \tag{2}$$

with pressure p, dynamic viscosity μ and external force $\mathbf{F}$. For later use, we will reformulate these equations in other formats:

$$\frac{\partial\rho}{\partial t} + \nabla\cdot(\rho\mathbf{u}) = 0 \iff \frac{D\rho}{Dt} + \rho(\nabla\cdot\mathbf{u}) = 0 \quad\text{and}\quad \frac{\partial\mathbf{u}}{\partial t} + \frac{1}{\rho}\nabla p = \mathbf{R}, \tag{3}$$

where $\mathbf{R}$ contains the convective and diffusive terms as well as the body force. The form of the momentum equation in (3) has been chosen because the quotient $\rho^{-1}\nabla p$ is constant for a hydrostatic pressure in two-phase flow, although ρ as well as ∇p are discontinuous between the two phases. That means that one can 'safely' calculate discrete derivatives of this expression, as will be needed in the sequel.

Given the variable density ρ in the mass and momentum equations, an additional equation of state $\rho = \rho(p)$ has to be applied to describe the compressibility of the air phase. Currently, the polytropic equation of state is used, i.e.

$$\frac{\rho}{\rho_{ref}} = \left(\frac{p}{p_{ref}}\right)^{1/\gamma} \quad \text{with} \quad \gamma = 1.4. \tag{4}$$

In this equation, the initial (atmospheric) reference values for pressure and density are used [51].