Computers and Fluids 170 (2018) 41–52

Contents lists available at ScienceDirect

Computers and Fluids

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

Analysis of breaking and re-closure of a bubble near a free surface based on the Eulerian finite element method

Z.L. Tian, Y.L. Liu*, A.M. Zhang*, S.P. Wang

College of Shipbuilding Engineering, Harbin Engineering University, Harbin 150001, China

ARTICLE INFO

Article history:
Received 17 January 2018
Revised 6 March 2018
Accepted 17 April 2018
Available online 20 April 2018

Keywords:
Bubble dynamics
Free surface
Eulerian finite element method
Volume of fluid (VOF) method
Underwater explosion

ABSTRACT

The nonlinear interaction between a bubble and a nearby free surface is an important phenomenon, which becomes more complex when the bubble breaks and re-closes at the free surface. In this paper, an axisymmetric bubble dynamics model based on the Eulerian finite element method with the interface tracked by the volume of fluid (VOF) method is established to numerically investigated the breaking and re-closure of a bubble near a free surface. An experimental validation of the numerical model reveals its good accuracy. The motion, breaking, and re-closure of small and large bubbles at a free surface are simulated. Complex phenomena are observed in the simulation after the break of the bubble. Under the effects of inward gas flow and the pressure difference between its inside and outside, a broken bubble finally recloses. The convergence and impact of a spike wall generate an upward water spray higher than that from an intact bubble. Furthermore, the critical depth at which the bubble period reaches its peak has a negative relationship with the weight of the explosive charge used to generate the bubble.

1. Introduction

Bubbles with a large pressure difference between the internal gas and the ambient flow are widely observed, such as bubbles generated by underwater explosions threatening the safety of warships, bubbles caused by cavitation causing propellers erosion, and bubbles released by air guns to exploring the geology of the deep ocean. Much research has revealed the presence of nonlinear interactions when a bubble moves near a free surface [1–8]. The bubble is repelled by the free surface, and a liquid jet emerges from the latter, penetrating the bubble during the collapse phase. At the same time, a spike appears on the free surface and exhibits complex dynamics varying with the initial conditions. These complex characteristics are highly nonlinear because of the geometric nonlinearity caused by the large deformation of interfaces. Boundary element method (BEM) is one of the most widely used methods in non-spherical bubble dynamics research. As a significant pioneer, Blake systematically analyzed the mechanisms through which bubbles interact with different boundaries including free surfaces, and introduced the Blake criterion to predict the overall motion of such bubbles [1,9,10]. The doubly connected flow field that appears after a bubble has been penetrated by a jet violates the basic assumptions of potential flow in BEM. Wang et al. [6,11] and Pearson et al. [5] introduced different approaches to deal with this problem, thereby improving the application of the BEM to bubble dynamics. Zhang et al. [12] extended the application of the vortex ring model proposed by Wang to the cases of breaking of a toroidal bubble. Wang et al. [13] used the BEM to investigate the bursting of an underwater explosion bubble at a free surface and showed that a water pillar would emerge and rise to a great height even when the bubble breaks soon after formation. Because of the efficient way in which it deals with interfaces, the BEM has great advantages for the simulation of moving-boundary problems in infinite flow fields, specifically those involving bubble dynamics. However, when the topology of the fluid field changes in a complicated manner, such as in the case of jet penetration or the bursting of a bubble at a free surface, the need for mesh surgery and the presence of a multivalued velocity potential cause great difficulties for numerical simulations. For this reason, most previous work in this area based on the BEM has focused on bubble dynamics in the early stages of oscillation.

Computational fluid dynamics (CFD) methods with an Eulerian mesh have been successfully applied in multiphase hydrodynamics simulations. Interface tracking methods are crucial for the representation and simulation of multiphase flows. The front tracking method [14–16] advances the interface with a prescribed velocity field through an Eulerian mesh, which, like the BEM, can preserve sharp interfaces, but suffers from the same difficulties when faced with topology change. The level-set method [17–21] and the volume of fluid (VOF) method [22–26] both introduce auxiliary func-

* Corresponding authors.
E-mail addresses: yunlong_liu@hrbeu.edu.cn (Y.L. Liu), zhangaman@hrbeu.edu.cn (A.M. Zhang).

https://doi.org/10.1016/j.compfluid.2018.04.028
0045-7930/

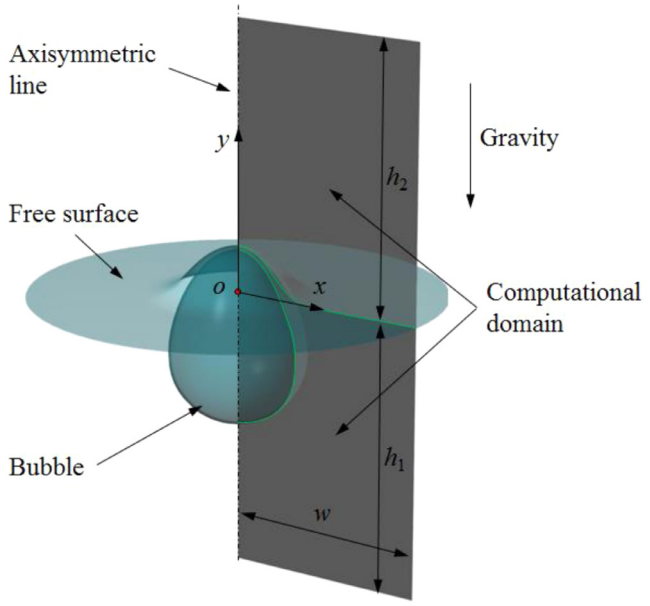

Fig. 1. Configuration for the interaction between a bubble and a free surface.

tions to determine the location of the interface and advect them with the transfer equation. These methods have the necessary flexibility to deal with changes in interface topology and thus both of them have been widely used in bubble dynamics simulations.

The present paper employs the Eulerian finite element method to solve an axisymmetric problem in which a bubble moves near a free surface. The VOF method is adopted to deal with the multiphase problem. Both small and large bubbles moving and bursting near a free surface are simulated and analyzed.

2. Theoretical and numerical models

The problem studied here is sketched in Fig. 1. A spherical high-pressure gas bubble is initially placed beneath the free surface in still water at a depth d. Then, the bubble starts expanding and interacting with the free surface. To simplify the numerical model, environment factors, such as the surface waves, the ocean current and nearby boundaries except the free surface and the bubble, are all ignored. Thus, with the axisymmetric initial and boundary conditions, the flow field is also assumed to be axisymmetric, although the symmetry may be broken when small-scale disruptions of the interface are considered. This assumption limits the present numerical model into an idealized problem. Thus, following the approach adopted in previous work [1,4,11,27,28], a cylindrical coordinate system is set up with the origin o at the free surface right above the bubble and with oy taken as the symmetry axis. x represents the radial coordinate, while y represents the axial coordinate pointing upward. The computational domain is taken as a rectangle of width w and heights h_1 and h_2 beneath and above the free surface, respectively.

2.1. Eulerian finite element method for bubble dynamics

It has been shown that the viscosity of the fluid is not important for bubbles generated by underwater explosion or cavitation with a high Reynolds number [3,27,29–31]. The stress tensor of the fluid can then be simplified as a scalar representing the pressure. It is assumed in this paper that the Mach number of the gas within the bubble is sufficiently high that the compressible Eulerian equations are appropriate. It is clear that the details of the disruption of the interface and the motion of small droplets thereby formed will be dominated by surface tension effects. However, this paper is concerned only with the overall flow pattern. Thus, given the length scales of the liquid jet and the free surface spike, the surface tension effects are ignored following previous published papers [1,4,6,46].

The Eulerian finite element method is an efficient approach to solve structural or fluid dynamics problems involving large deformations. Its detailed formulation can be found in the literature [32–34]. In this paper, an operator split technique is used to separate the calculation into two phases within a single time increment. The first phase is solved using the explicit finite element method from a Lagrangian perspective in which the mesh moves along with the material. At the end of the time increment, the deformed mesh is moved back to its original position, and the convection between elements is calculated; this is usually called the Eulerian phase.

In the Lagrangian phase, the conservation of momentum can be expressed as

$$\frac{d\rho\dot{\mathbf{u}}}{dt} + \nabla p = \rho\mathbf{g}, \tag{1}$$

where ρ and p are the density and pressure of the fluid, $\boldsymbol{g}$ is the gravity, and $\boldsymbol{u}$ is the displacement vector. By multiplying both sides of Eq. (1) with an arbitrary weight function ϕ and integrating over the control volume V, the weak form of Eq. (1) can be rewritten as

$$\iiint_V \left(\frac{d\rho\dot{\mathbf{u}}}{dt} + \nabla p\right)\phi \, dv = \iiint_V \rho\mathbf{g}\phi \, dv. \tag{2}$$

The term $x\phi\nabla p$ is then integrated by parts and the formula is expanded in the cylindrical coordinate system as

$$\iint_\Omega x(\rho\ddot{\mathbf{u}}\phi - p\nabla\phi)\, d\Omega = -\int_\Gamma xp\hat{\mathbf{n}}\phi \, d\Gamma + \iint_\Omega p\hat{\mathbf{e}}_1\phi \, d\Omega + \iint_\Omega x\rho\mathbf{g}\phi \, d\Omega, \tag{3}$$

where the volume integral has been simplified as an area integral over the discrete element Ω, and factors of 2π have been dropped on both sides. The first two integrations on the right-hand side are derived from the integration by parts and the Gauss–Green formula. Γ is the boundary of the integration domain Ω, $\hat{\mathbf{n}}$ is its unit normal vector, and $\hat{\mathbf{e}}_1 = \nabla x$ is the unit vector pointing in the radial direction. Taking Φ as the shape function of the element, Eq. (3) can be rewritten in the semi-discretized form

$$\int_\Omega (x\rho\Phi_M\Phi_N)\, d\Omega \, \ddot{\mathbf{u}}^N = \int_\Omega (x\rho\mathbf{g}\Phi_M - xp\nabla\Phi_M)\, d\Omega + \int_\Gamma xp\hat{\mathbf{n}}\Phi_M d\Gamma + \iint_\Omega p\hat{\mathbf{e}}_1\Phi_M d\Omega, \tag{4}$$

where the arbitrary weight function ϕ has been replace by the shape function Φ. The subscripts M and N indicate nodal variables at nodes M and N, respectively. The acceleration vector $\ddot{\mathbf{u}}$ is then calculated by solving Eq. (4). The nodal velocity and displacement are integrated explicitly with the second-order central difference method,

$$\dot{\mathbf{u}}^{n+1/2} = \dot{\mathbf{u}}^{n-1/2} + \ddot{\mathbf{u}}^n \Delta t, \tag{5}$$

$$\mathbf{u}^{n+1} = \mathbf{u}^n + \dot{\mathbf{u}}^{n+1/2}\Delta t, \tag{6}$$

where Δt is the time increment, and the superscript n indicates the number of the increment. After the nodal position has been