Applied Ocean Research 59 (2016) 183–192

Contents lists available at ScienceDirect

Applied Ocean Research

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

Pressure characteristics of bubble collapse near a rigid wall in compressible fluid

Wang Long-kan, Zhang Zhi-fan, Wang Shi-ping*

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

ARTICLE INFO

Article history:
Received 6 January 2016
Received in revised form 15 April 2016
Accepted 6 June 2016

Keywords:
Bubble jet
Shock wave
Pressure characteristics
Compressible fluid

ABSTRACT

High speed liquid jet and shockwave can be produced when a bubble collapses near a rigid wall, which may cause severe damage to solid structures. A hybrid algorithm was adopted to simulate bubble motion and associated pressures near a wall combining Level Set-Modified Ghost Fluid-Discontinuous Galerkin (LS-MGF-DG) method and boundary element method (BEM). Numerical results were compared with experimental data to validate the presented algorithm. Jet formation was simulated by BEM and the induced pressure on the wall was calculated with auxiliary function. The pressure at the point on the wall where the jet points to reaches its peak value after the jet penetrates the bubble. Bubble collapse and rebounding were simulated by the LS-MGF-DG method. Shock-wave is induced when the bubble collapse toroidally to a minimum volume and the pressure at wall center reaches the maximum due to shockwave superposition. A third pressure peak is found associated with the bubble rebounds and bubble splitting. In the case studied, a higher pressure was found due to collapse shockwave than bubble jet and affects a larger area of the wall. In addition, the three pressure peaks due to jet impact, collapse impact as well as bubble rebounding and splitting decrease with the increase of the standoff distance.

1. Introduction

Cavitation may lead to erosion of propellers. Bubble pulsating pressure and liquid jet may cause damage on warships [1]. The main reason of damage includes the high-speed jet generated when a bubble is attracted by the surface of a structure during collapse and the shockwaves induced when the bubble volume reaches the minimum. Experimental research [2–6] may still be the most effective and direct method to investigate bubble dynamics. Nevertheless, experimental research can be unrepeatable, dangerous, and less comprehensive in data collecting. Therefore, numerical simulation [7–11] is usually combined with experiments to study bubble dynamic characteristics. Rayleigh [12] developed a motion equation of a spherical bubble in an incompressible flow field and found that high-pressure pulses and shock waves were generated by a volume implosion when the bubble shrinks to its minimum volume. In Naude and Ellis's [13] research, results show that instead of a spherical collapse, a high-speed jet would form when the bubble is close to the rigid wall, which revealed the mechanisms of cavitation corrosion. Lauterborn *et al.* [14,15] investigated the dynamics of a single laser-produced bubble near a wall and cavitation erosion; the shock wave generated in the first collapse phase of the bubble was captured via high-speed photography. Klaseboer *et al.* [16] combined experiments and Boundary Element Method (BEM) to investigate the dynamics of an underwater explosion bubble near a resilient/rigid structure. Hsiao *et al.* [17] adopted a hybrid numerical approach based on an incompressible BEM and a compressible finite-difference method to obtain the impulsive pressures on material surface during bubble collapse phase. Li *et al.* [18] applied BEM to study bubble motion near the rigid wall as well as pressure characteristics and combined BEM with auxiliary function method to calculate the wall pressure.

Although bubble dynamics and jet characteristics have been experimentally and numerically studied by various researchers, few published papers discuss the shock wave pressure associated with bubble collapse. For experimental researches, it is difficult to measure the wall pressure due to a limited photographic technique. As the Mach number at the tip of a bubble jet could reach 0.3 or higher [19,20] during the collapse phase, fluid compressibility has a significant influence on bubble motion and should be considered. Although BEM is more computationally effective, some numerical techniques that come along with BEM may have negative influence on the accuracy of numerical results, such as the Vortex-Ring model, the Vortex sheet model and the numerical fairing in

* Corresponding author.
E-mail address: wangshiping@hrbeu.edu.cn (S.-p. Wang).

http://dx.doi.org/10.1016/j.apor.2016.06.003

the simulation of a bubble-jet formation. In addition, the impact of shock-wave on structures such as a rigid wall cannot be simulated with BEM. The LS-MGF-DG method which takes fluid compressibility into consideration is desirable to accurately capture emission and propagation of shockwaves in the process of bubble motion. Therefore, the numerical simulation in this paper combines BEM and LS-MGF-DG methods and benefits from better efficiency and shockwave capturing accuracy. The combined method is adopted to investigate the dynamics and pressure characteristics of bubble oscillation near a rigid wall. In addition, the numerical results are compared with the experimental data to verify the effectiveness of the hybrid numerical model.

2. Numerical and experimental methods

The fluid can be assumed inviscid and irrotational for the underwater explosion bubble with high Reynolds number simulated in this paper. Due to a small Mach number during bubble expansion and contraction phase the compressibility can be ignored while it should be considered when the Mach number reaches 0.3 or higher during later stages. Therefore, the BEM based on the incompressible potential flow theory is applied to simulate the process of bubble expansion and contraction while LS-MGF-DG method considering the compressibility is used to simulate the formation of bubble jet. The verification of these two methods can be found from Refs. [21,22], in which numerical results were compared with experimental data.

2.1. Boundary element method

The numerical model of bubble dynamics based on a high-efficiency BEM can be found in Wang's [23] and Zhang's [8,10,18] researches. The surrounding fluid is assumed to be incompressible in the simulation of underwater explosion bubble using BEM. Hence, the fluid satisfies the Laplace Equation, which can be given by

$$\nabla^2 \Psi = 0 \tag{1}$$

where Ψ is the velocity potential.

According to Green Equation, the velocity potential at any point in the fluid filed Ω can be expressed as a function of the potential at

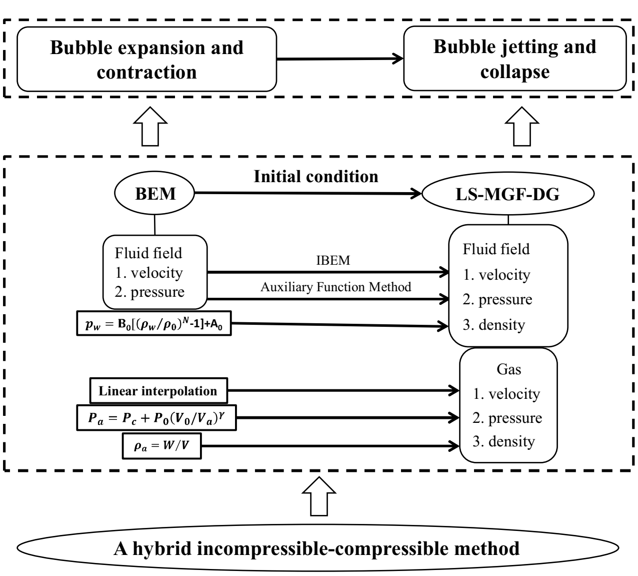

Fig. 1. Flowchart of a hybrid incompressible-compressible method.

the boundary S and its normal derivative. The boundary integration equation can be expressed as

$$\lambda\Psi(\mathbf{p}) = \iint_s \left(\frac{\partial\Psi(\mathbf{q})}{\partial n} G(\mathbf{p},\mathbf{q}) - \Psi(\mathbf{q}) \frac{\partial}{\partial n} G(\mathbf{p},\mathbf{q}) \right) dS \tag{2}$$

where λ, S and n denote the solid angle, the boundary of the fluid field and the normal vector pointing out of the fluid field, respectively; p and q denote field and source points on the boundary, respectively.

Eq. (2) is solved in an axisymmetric coordinate as follows. The bubble surface was discretized into N elements with M nodes, and the discretized form of Eq. (2) is

$$\sum_{j=1}^{M} (W_{ij} \frac{\partial \Psi_j}{\partial n}) = \sum_{j=1}^{M} (M_{ij}\Psi_j) - \varepsilon(i)\Psi_i \tag{3}$$

where W_{ij} and M_{ij} are influence coefficients. The solution can be found in Wang [23].

The bubble surface satisfies the kinematic boundary condition and the dynamic boundary condition, written as

$$\frac{d\boldsymbol{r}}{dt} = \nabla\Psi \tag{4}$$

$$\frac{d\Psi}{dt} = \frac{|\nabla\Psi|^2}{2} + \frac{p_\infty}{\rho} - \frac{p}{\rho} - gz \tag{5}$$

where $\boldsymbol{r}$ is the position vector of a node on the bubble surface; p_∞ denotes the pressure infinitively far away from the initial charge center in the horizontal direction; p is the pressure inside the bubble; ρ is the fluid density.

The pressure P at any point in the fluid domain can be obtained through the Bernoulli equation [18,24], given by

$$P = P_\infty - \rho(\frac{\partial\Psi}{\partial t} + \frac{1}{2}|\nabla\Psi|^2 + gz) \tag{6}$$

where the velocity $\nabla\Psi$ can be solved by indirect boundary element method (IBEM) and $\partial\Psi/\partial t$ can be evaluated by using auxiliary function method.

The velocity potential $\Psi(i)$ at point i in the fluid domain is given by

$$\Psi(i) = \iint \frac{\sigma(j)}{|r-R|} ds_j = G \cdot \sigma(j) \tag{7}$$

where i and j denote the points in the fluid domain and on the bubble surface respectively; $\sigma(j)$ is the density of the distributed sources on the bubble surface.

We assume that the point i is located on the bubble surface. Therefore, $\sigma(j)$ can be obtained through Eq. (7) as

$$\sigma(j) = G^{-1} \cdot \Psi(i) \tag{8}$$

Thus, the velocity at point i in the fluid domain can be obtained by

$$\nabla\Psi(i) = \nabla \iint \frac{\sigma(j)}{|r-R|} ds = \int \sigma(j) r \nabla_i (\int_0^{2\pi} \frac{1}{|r-R|} d\theta) dl \tag{9}$$

The auxiliary function method is applied to solve the term $\partial\Psi/\partial t$ in the fluid domain, in which the auxiliary function K is defined as

$$\mathrm{K} = \frac{\partial\Psi}{\partial t} \tag{10}$$

The auxiliary function K satisfies the Laplace equation in the fluid domain due to similar mathematical properties between the auxiliary function K and the velocity potential Ψ. Then we have

$$\nabla^2 \mathrm{K} = 0 \tag{11}$$