Theoretical & Applied Mechanics Letters 7 (2017) 231–234

Contents lists available at ScienceDirect

Theoretical & Applied Mechanics Letters

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

Letter

A numerical model for cloud cavitation based on bubble cluster

Tezhuan Du *, Yiwei Wang, Chenguang Huang, Lijuan Liao

Key Laboratory for Mechanics in Fluid Solid Coupling Systems, Institute of Mechanics, Beijing 100190, China
School of Engineering Science, University of Chinese Academy of Sciences, Beijing 100049, China

HIGHLIGHTS

- The structure of cloud cavitation is needed to accurately predict collapse pressure.
- A cavitation model was developed through dimensional analysis and CFD simulation.
- Bubble number density was used to characterize the internal structure of cavitation.
- This model is implemented on cavitating flow over a projectile with conical head.
- The results show that the collapse pressure is affected by bubble number density.

ARTICLE INFO

Article history:
Received 28 April 2017
Received in revised form 17 July 2017
Accepted 19 July 2017
Available online 30 August 2017
*This article belongs to the Fluid Mechanics

Keywords:
Cavitation model
Bubble number density
Bubble cluster
Collapse

ABSTRACT

The cavitation cloud of different internal structures results in different collapse pressures owing to the interaction among bubbles. The internal structure of cloud cavitation is required to accurately predict collapse pressure. A cavitation model was developed through dimensional analysis and direct numerical simulation of collapse of bubble cluster. Bubble number density was included in proposed model to characterize the internal structure of bubble cloud. Implemented on flows over a projectile, the proposed model predicts a higher collapse pressure compared with Singhal model. Results indicate that the collapse pressure of detached cavitation cloud is affected by bubble number density.

Cloud cavitation, as one of the most common type of cavitation for the marine vessels and projectile, consists of a large amount of small bubbles. The pressure pulse generated by the bubble collapse in the cloud cavitation is usually considered to be the major cause of the structure failure and the noise radiation. Numerical simulations are commonly used to study cavitating flows in many applications, such as hydrofoils, projectiles, turbo machines, etc. Most of the researches were focused on the homogeneous flow modeling, which is based on the Navier–Stokes equations of mixture phase [1–7]. However, the homogeneous cavitation modeling is only capable of providing macro-solutions of vapor volume fraction while the micro mechanism of cavitation evolution remains unknown. For example, bubble clusters with different distributions may lead to different collapse pressures because of the interaction of bubbles under identical vapor volume fractions. Therefore, a numerical model considering the internal structure of cloud cavitation is essential to understand bubble evolution and improve the accuracy of predicting bubble collapse pressure. Evans et al. [8] developed a population balance model to predict bubble size distribution in the wake region below a ventilated gas cavity. Du et al. [9] proposed an evolution model of bubble number density is proposed to simulate bubble breakup and transportation.

In this letter, a cavitation model based on bubble cluster is deduced by dimensional analysis and direct simulation. A numerical strategy is established to solve flow with cloud cavitation by combining the developed model with the multi-phase Reynolds averaged Navier–Stokes (RANS) equations and transport equation of bubble number density. This strategy is applied on simulation of cavitating flow around a projectile. These solutions are compared with experimental observations. The comparisons of proposed model and Singhal model are discussed.

The proposed numerical model consists of multi-phase RANS equations of the mixture phase, transport equation of bubble number density [9], modified re-normalized group (RNG) k–ε model [10], continuity equation of the vapor phase and cavitation model based on bubble cluster.

The condensation rate based on bubble cluster is constructed by dimensional analysis and direct numerical simulation. All bubbles are assumed to be spherical and in the state of equilibrium at

* Corresponding author.
E-mail address: dutezhuan@imech.ac.cn (T. Du).

http://dx.doi.org/10.1016/j.taml.2017.08.001

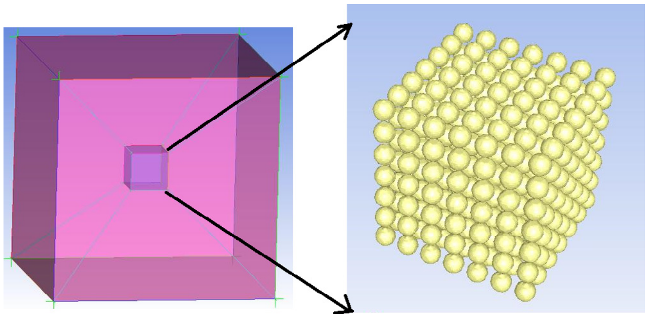

Fig. 1. Flow field and numerical model.

the beginning and then collapse under ambient pressure p_∞. The averaged variation rate of total bubble volume can be expressed as:

$$\dot{V} = f\left(R_0, a; n; \rho_{\mathrm{L}}, \mu_{\mathrm{L}}, S, \rho_{\mathrm{A0}}, \mu_{\mathrm{A}}, \gamma; p_\infty, p_{\mathrm{B}}\right), \tag{1}$$

where R_0 is initial bubble radii, a is characteristic length of bubble cluster, n represents population of bubbles per unit volume. ρ_{L} and μ_{L} are density and viscosity of liquid, respectively. ρ_{A0} and μ_{A} are reference density and viscosity of gas, respectively. γ is ratio of specific heats of gas, S is surface tension. p_∞ and p_{B} are surrounding pressure and pressure inside bubbles, respectively. Taking ρ_{L}, R_0, $p_\infty - p_{\mathrm{B}}$ as a unit system produces dimensionless form:

$$\frac{\dot{V}}{\frac{4}{3}\pi R_0^3/T_{\mathrm{C}}} = f\left(\alpha;\ N;\ \frac{\mu_{\mathrm{L}}}{\rho_{\mathrm{L}}R_0^2/T_{\mathrm{C}}}, \frac{S}{\rho_{\mathrm{L}}R_0^3/T_{\mathrm{C}}^2}, \frac{\rho_{\mathrm{A0}}}{\rho_{\mathrm{L}}}, \frac{\mu_{\mathrm{A}}}{\rho_{\mathrm{L}}R_0^2/T_{\mathrm{C}}}, \gamma, \frac{p_\infty - p_{\mathrm{B}}}{p_\infty}\right), \tag{2}$$

where $T_{\mathrm{C}} = 0.915R_0\sqrt{\frac{\rho_{\mathrm{L}}}{p_\infty - p_{\mathrm{B}}}}$ represents collapse time of single bubble. The collapse of bubbles is mainly controlled by pressure difference and inertial force, which provides the possibility of ignoring of viscosity and surface tension. For given material $\frac{\rho_{\mathrm{A0}}}{\rho_{\mathrm{L}}}$ and γ, Eq. (2) can be reduced to:

$$\dot{V}' = f\left(\alpha;\ N; p'\right), \tag{3}$$

where $p' = (p_\infty - p_{\mathrm{B}})/p_\infty$ represents non-dimensional driving pressure, $\dot{V}_0 = \frac{4}{3}\pi R_0^3/T_{\mathrm{C}}$ represents volume variation rate of single bubble, therefore $\dot{V}' = \dot{V}/\dot{V}_0$ is non-dimensional volume variation rate of bubble cluster. According to the π theorem, we can assume that:

$$\dot{V}' = c\alpha^{k_1}N^{k_2}p'^{k_3}. \tag{4}$$

The empirical parameters c, k_1, k_2, k_3 can be determined by direct simulation of collapse of bubble clusters.

Consider a cubic bubble cluster with primitive cubic lattice distribution, in which bubbles are arranged in order as shown in Fig. 1. The initial pressure inside bubble cluster is set to be p_{B}, and the pressure at boundary is p_∞. The liquid is incompressible water, and the gas inside the bubbles is air modeled as ideal.

Volume of fraction (VOF) method and large eddy simulation (LES) are adopted in the numerical simulations. The parameters are listed as follows:

volume faction α : 0.15, 0.27, 0.50,
population of bubbles N : 3^3, 4^3, 5^3, 6^3,
non-dimensional pressure p' : 0.6, 0.7, 0.8, 0.9.

The results show that bubble cluster collapses layer by layer from outside to inside (Fig. 2).

Combined with the influence of α, N, and p', Eq. (4) becomes :

$$\dot{V}' = -\alpha^{-0.2}N^{0.68}p'^{0.61}. \tag{5}$$

Equation (5) gives the no-dimensional collapse velocity of bubble cluster. We can get the condensation rate as follows:

$$\dot{m}^- = -C_{\mathrm{C}}\frac{\rho_{\mathrm{L}}\rho_{\mathrm{v}}}{\rho_{\mathrm{m}}}\left(\frac{n^{0.013}}{V_{\mathrm{m}}^{0.32}\alpha^{0.53}}\right)\alpha\left(\frac{p-p_{\mathrm{B}}}{p}\right)^{0.61}\sqrt{\frac{2\left(p-p_{\mathrm{B}}\right)}{3\rho_{\mathrm{L}}}}, \tag{6}$$

where V_{m} is the volume of a mesh cell, and $C_{\mathrm{C}} = 0.057$. According to Singhal model, the evaporation rate is set to be:

$$\dot{m}^+ = C_{\mathrm{e}}\rho_{\mathrm{L}}\frac{\rho_{\mathrm{L}}\rho_{\mathrm{v}}}{\rho_{\mathrm{m}}}\frac{\sqrt{k}}{\sigma}\sqrt{\frac{2\left(p_{\mathrm{B}}-p_\infty\right)}{3\rho_{\mathrm{L}}}}\left(1-\alpha_{\mathrm{v}}-\alpha_g\right). \tag{7}$$

The present model is then implemented on the cavitating flow over a cylindrical body with 90° conical head. The diameter of cylinder is $d = 37.5$ mm, and the free-stream velocity is $v = 18$ m/s. The corresponding cavitation number and Reynolds number are $\sigma = 0.612$, $Re = 6.75 \times 10^5$, respectively. The 2-D axisymmetric geometry is utilized in the simulation, with a 350 × 150 structural mesh, depicted in Fig. 3. The numerical results are validated by the experimental data based on the split Hopkins pressure bar (SHPB) launch system [11].

Figure 4 illustrates the outline of cavitation and the distribution of bubble number density n. The black curves represent the iso-surface of vapor fraction with the value of 0.1, and the white zones in the cavitation indicate the high bubble number density area ($n \geq 10^9$). It can be seen that cloud cavitation area is consistent with the region swept by the re-entrant jet, which suggests that the re-entrant jet breaks bubbles into small ones, and forms the cavitation cloud.

The performance of the present model and Singhal model on predicting pressure focusing effect are compared in this letter. Results show that present model presents a higher condensation

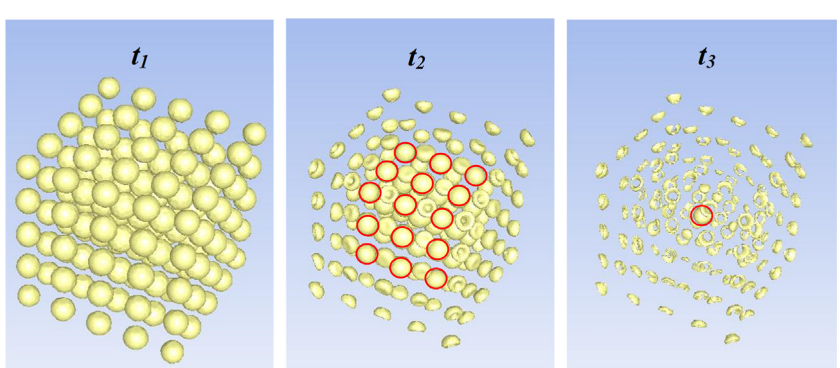

Fig. 2. Collapse of bubble cluster ($\alpha = 0.15, N = 5^3, p' = 0.8$).

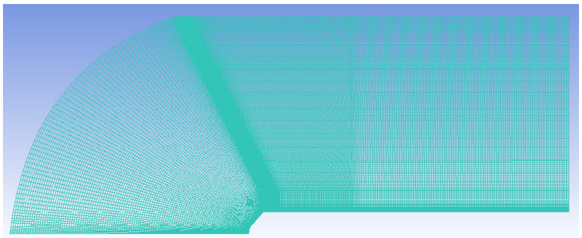

Fig. 3. Computational domain.

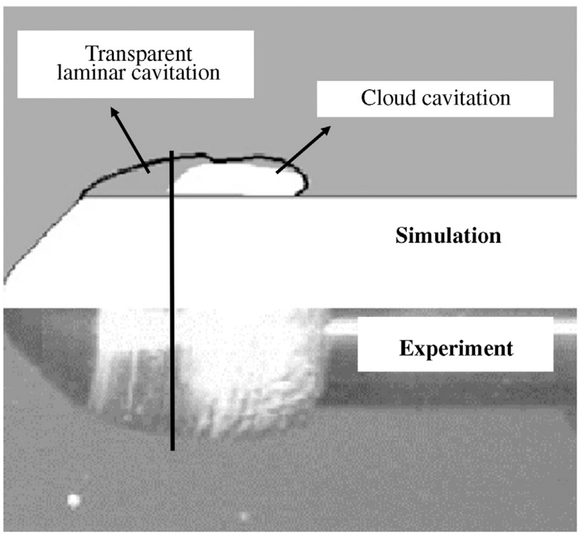

Fig. 4. Formation of cloud cavitation.

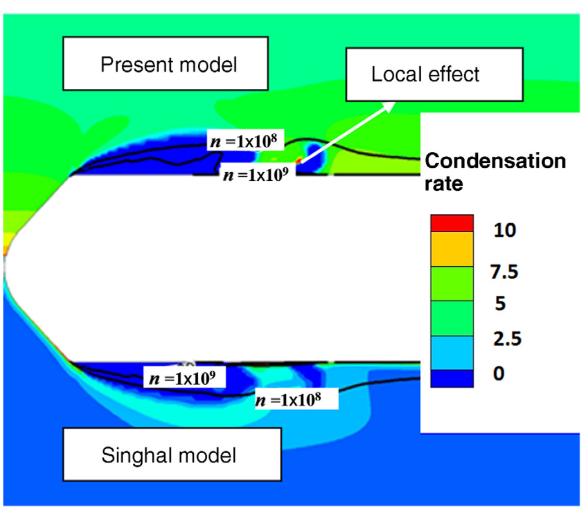

Fig. 5. Comparison of condensation rates.

rate and collapse pressure than Singhal model (Figs. 5 and 6). Furthermore, the condensation rate of Singhal model is quite smooth while present model shows local effects of collapse. The region of cavitation cloud with higher bubble number density collapses more fiercely than other else. That is an important characteristic named geometry focusing of collapse pressure in the collapse process of bubble cluster. The pressure waves emitted by collapse

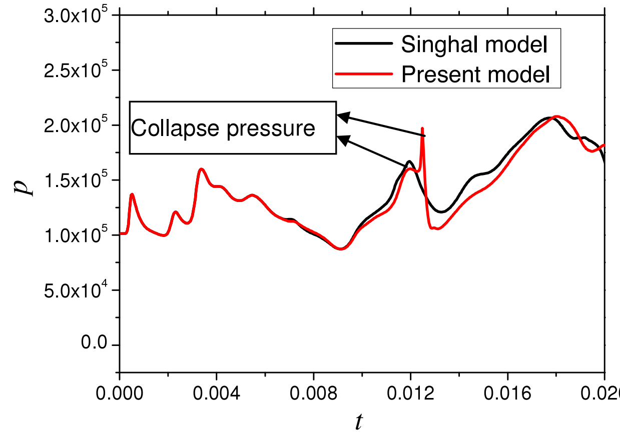

Fig. 6. Comparison of collapse pressure (at $x/d = 2$).

bubbles will propagate inward which leads to a higher magnitude of collapse pressure [12].

In this letter, a numerical model for cloud cavitation based on bubble cluster is proposed. The condensation rate is constructed through dimension analysis and direct simulation of collapse of bubble cluster. Bubble cluster collapses layer by layer from outside to inside. The geometry focusing effect leads to a higher magnitude of collapse pressure. This model is implemented on cavitating flow over a projectile with conical head. The simulation results show that present model can predict the evolution of cavitation and the distribution of bubble number density. The performance of present model and Singhal model are compared. Proposed model predicts a higher collapse pressure than Singhal model due to the geometry focusing effect. The results show that the collapse pressure of detached cavitation cloud is affected by bubble number density. Non-uniform distribution of bubble number density may lead to different collapse velocities. The region of cavitation cloud with higher bubble number density collapses more fiercely than other else.

Acknowledgment

The authors acknowledge the financial support from the National Natural Science Foundation of China (11402276).

References

[1] A. Kubota, H. Kato, Unsteady structure measurement of cloud cavitation on a foil section, J. Fluids Eng. 111 (1989) 204–210.
[2] C.L. Merkle, J. Feng, P.E.O. Buelow, Computational modeling of the dynamics of sheet cavitation, in: Proceedings of 3rd International Symposium on Cavitation, Grenoble, France, 1998.
[3] R.F. Kunz, D.A. Boger, D.R. Stinebring, et al., A preconditioned Navier–Stokes method for two-phase flows with application to cavitation prediction, Comput. & Fluids 29 (2000) 849–875.
[4] A.K. Singhal, M.M. Athavale, H. Li, et al., Mathematical basis and validation of the full cavitation model, J. Fluids Eng. 124 (2002) 617–624.
[5] Y.W. Wang, C.G. Huang, X. Fang, et al., Cloud cavitating flow over a submerged axisymmetric projectile and comparison between two-dimensional RANS and three-dimensional large-eddy simulation methods, Trans. ASME, J. Fluids Eng. 138 (2016) 061102.
[6] X.X. Yu, C.G. Huang, T.Z. Du, et al., Study on characteristics of cloud cavity around axisymmetric projectile by large eddy simulation, ASME J. Fluids Eng. 136 (2014) 051303.
[7] B. Ji, X. Luo, X. Peng, et al., Numerical analysis of cavitation evolution and excited pressure fluctuation around a propeller in non-uniform wake, Int. J. Multiph. Flow 43 (2012) 13–21.
[8] G.M. Evans, P.M. Machniewski, A.K. Bin, Bubble size distribution and void faction in the wake region below a ventilated gas cavity in downward pipe flow, Chem. Eng. Res. Des. 82 (2004) 1095–1104.

[9] T.Z. Du, Y.W. Wang, L.J. Liao, et al., A numerical model for the evolution of internal structure of cavitation cloud, Phys. Fluids 28 (2016) 3–20.
[10] J.L. Reboud, B. Stutz, O. Coutier, Two-phase flow structure of cavitation: experiment and modeling of unsteady effects, in: the 3rd International Symposium on Cavitation, Grenoble, France, 1998.
[11] Y.P. Wei, Y.W. Wang, X. Fang, et al., A scaled underwater launch system accomplished by stress wave propagation technique, Chin. Phys. Lett. 28 (2011) 024601.
[12] Y.C. Wang, Distribution on the dynamics of a spherical cloud of cavitation bubbles, J. Fluids Eng. 121 (1999) 881–886.