PAPER • OPEN ACCESS

Numerical simulation of unsteady cloud cavitation: a comparative study of compressible mixture models

To cite this article: G Peng *et al* 2019 *IOP Conf. Ser.: Earth Environ. Sci.* **240** 062043

Numerical simulation of unsteady cloud cavitation: a comparative study of compressible mixture models

G Peng[1], M Mori, T Tazaki, Y Oguma

Department of Mechanical Engineering, College of Engineering, Nihon University
Tamura-machi, Koriyama, Fukushima 963-8642 Japan
[1] Email: peng@mech.ce.nihon-u.ac.jp

Abstract. Focused on the unsteady behavior of intensively cavitating water jets numerical analysis is carried out by applying the compressible mixture flow method under assumptions of bubble cavitation and homogenous mixture. Submerged water jets issuing from a sheathed nozzle is treated when the cavitation number $\sigma \cong 0.1$. The periodically shedding of cavitation clouds in submerged water jet is captured acceptably and the core velocity distribution evaluated by numerical simulation agrees with experiment data of PIV approximately. Concerning the effect of flow compressibility estimation, comparison of computation results reveals that it is capable to capture the unsteady behavior of cavitating flow by both the simplified bubble cavitation model and the homogeneous mixture model. However, the homogenous model shows a tendency to estimate gas volume fraction of cavitation clouds excessively. It becomes necessary to evaluate the mixture compressibility by considering the effect of bubble dynamics in modeling intensively cavitating flow. The simplified isothermal bubble cavitation model is demonstrated to be a practical method for treating bubble-liquid flow with intensive cavitation.

1. Introduction

High-speed water jet injected into still water, which is called submerged water jet, has received much attention for its capacity of generating very high cavitation impact pressure in the collapse of cavitation bubbles, and widely applied to such as peening of metal materials, decomposition of toxic substances and purification of sewages, etc. [1]. Until now, many experimental studies have been made concerning jet driven pressure, shape and size of a nozzle, cavitation number etc. However, the flow structure of cavitating jet and the behavior of unsteady cavitation clouds are still unclear for the difficulty to observe the interior of cavitating flow [2]. For the purpose of performance prediction and optimum design of water jet devices numerical simulation of high-speed water jets with intensive cavitation becomes a task full of challenges [3].

Cavitation usually takes place in low-pressure regions of relative high velocity and cavitating flows in most industrial applications are turbulent. The flow dynamics at the interface formed between liquid and gas phases involves complex bubble-bubble and bubble-liquid interactions. These interactions are still not well understood in the closure region of cavities. Also, the near field of cavities reveals to be highly compressible due to the growth and collapse of bubbles while the far-field away from cavities is essentially incompressible. For the difficulty to consider all these different characteristics numerical simulations of cavitating flow have been conventionally performed by introducing certain simplifications on a special flow field concerned. Most of numerical simulations of turbulent cavitating flow are based on Unsteady Reynolds Averaged Navier-Stokes (URANS) equations and cavitation models applied may be mainly classified into two-fluid and two-phase mixture flow method. Differing to the two-fluid method the pseudo one-fluid mixture flow method treats the cavitating fluid media as a locally homogeneous fluid mixture by neglecting the velocity slip between the liquid and gas phases [4, 5]. The physical property of two-phase mixture is dependent on the volume fraction of gas phase, which varies greatly with oscillation of cavitation cloud. For evaluating the variation of gas volume fraction a supplementary equation estimating such as mass transfer between the two phases is usually introduced by employing a certain cavitation model. Thus, the computation of two-phase mixture models is much cheap since there is no need to treat the motion of a mass of bubbles separately. These methods are becoming more popular because they can be applied to turbulent flows encountered in most industrial applications [6].

Focused on the unsteady behavior of high-speed water jets numerical simulations are performed by applying the compressible mixture flow method under assumptions of bubble cavitation and homogenous mixture. The mean flow of two-phase mixture is calculated by the set of URANS equations for compressible flow and the intensity of cavitation is evaluated by the volume fraction of gas bubbles whose radius is estimated with a simplified Rayleigh-Plesset equation. High-speed submerged water jets issuing from a sheathed nozzle are treated and the temporal variation of cavitating flow is investigated. The periodically shedding of cavitation clouds in submerged water jet is captured acceptably and the core velocity distribution evaluated by numerical simulation agrees with experiment data of PIV approximately. Concerning the effect of flow compressibility estimation, comparison of computation results reveals that it is capable to capture the unsteady behavior of cavitating flow in some extent by both the simplified bubble cavitation model and the homogeneous mixture model. However, it becomes necessary to evaluate the fluid compressibility by considering the effect of bubble dynamics for evaluating the local gas volume fraction of cavitation clouds. The simplified isothermal bubble cavitation model is demonstrated to be a practical way for treating bubble-liquid flow accompanying intensive cavitation.

2. Compressible mixture flow bubble cavitation model

Experimental observations show that cavitation caused in high speed submerged water jet appears in the form of bubble cloud. Corresponding to the variation of surrounding liquid pressure bubble clouds expand and break up unsteadily. Thus, the density of two-phase mixture varies sharply with bubble oscillation and it becomes essential to consider the effect of fluid compressibility in the numerical simulation of intensive cavitating flows.

2.1 Compressibility of bubble-liquid mixture

The fluid media of cavitating flow are taken as a two-phase mixture of working liquid and cavitation bubbles in this work. The gas bubbles are supposed to uniformly disperse in the liquid phase and its volume faction is denoted as α_G. The liquid volume fraction is written to be α_L. Then, the density of two-phase mixture can be defined as follows by volume averaging.

$$\rho_M = \rho_G \alpha_G + \rho_L \alpha_L \tag{1}$$

where ρ denotes fluid density, the subscripts L and G do the liquid phase and the gas phase respectively, M represents does the two-phase mixture. Then variation of mixture density can be obtained by taking the differential of above equation and it is then arranged to the following form.

$$\frac{1}{\rho_M}\frac{d\rho_M}{dt} = \frac{\alpha_G}{\rho_G}\frac{d\rho_G}{dt} + \frac{\alpha_L}{\rho_L}\frac{d\rho_L}{dt} \tag{2}$$

According to the definition of compressibility the above equation is written as follows.

$$\frac{1}{\rho_M}\frac{d\rho_M}{dt} = \frac{1}{\rho_M c_M^2}\frac{dp_L}{dt} \tag{3}$$

where,

$$\frac{1}{\rho_M c_M^2} = \frac{\alpha_L}{\rho_L c_L^2} + \frac{\alpha_G}{\rho_G c_G^2}\frac{\partial p_G}{\partial p_L} \tag{4}$$

in which c denote the sonic speed in fluid referred. According to above equation we understand that the compressibility of bubbly mixture depends upon the volume fraction of gas phase as well as the variation of gas pressure, which is determined by bubble size.

Concerning the compressibility all fluids concerned are supposed to work exponentially and the equation of state for the pure liquid phase is expressed by adopting the Tait's equation.

$$\frac{p_L + \mathrm{B}}{p_{L,sat} + \mathrm{B}} = \left(\frac{\rho_L}{\rho_{L,sat}} \right)^{n_L} \tag{5}$$

where $p_{L,sat} = 2.34$ kPa and $\rho_{L,sat} = 998.2$ kg/m^3, represent the saturation pressure and density of liquid phase (water) at the reference state when temperature T= 293.15 K. Two fitted constants as given to be B = 3.049×10^8 Pa and N = 7.1. Then, the sonic speed in liquid phase is written as follows.

$$\rho_L c_L^2 = n_L (p_L + \mathrm{B}) \tag{6}$$

As for the gas phase it is assumed that the gas included in a bubble consists of vapor and non-condensation gas. That is, $\rho_G = \rho_g + \rho_v$, where subscripts g and v denote the non-condensation gas and the vapor respectively. The non-condensation gas is perfect one and its state equation is given in the following form.

$$\frac{p_g}{p_{g0}} = \left(\frac{\rho_g}{\rho_{g0}} \right)^{n_g} \tag{7}$$

where $p_{g0} = 101.3$ kpa and $\rho_{g0} = 1.2$ kg/m^3 denote pressure and density of idea gas at the reference state when temperature T= 293.15 K. Then, the sonic speed in gas media may be given as follows.

$$\rho_G c_G^2 = n_g p_g \tag{8}$$

The pressure of non-condensation gas in a bubble is dependent upon bubble radius, R_b.

$$p_g = p_{g0} \left(R_{b0} / R_b \right)^{3n_g} \tag{9}$$

As stated above, the compressibility of gas phase varies with bubble radius. Of course, the oscillation of bubble radius with surrounding liquid pressure may be solved by the Rayleigh-Plesset equation or similar ones. But the calculation of these equations is time consuming for the high frequency of bubble oscillation. Here the cavitation bubbles are treated as quasi-still one and an estimation of bubble radius is obtained by solving a simplified Rayleigh-Plesset equation approximately [7].

$$\frac{dR_b}{dt} = Sign(\Delta P) \sqrt{\frac{2}{3} \frac{|\Delta P|}{\rho_L}} \tag{10}$$

where $\Delta P = p_g + p_v - p_L$, denotes the pressure difference acting on the bubble surface.

With above equation the bubble radius may be estimated according the liquid pressure and then the sonic speed in the bubbly mixture may be evaluated by equation (4). If the effect of fluid interface is negligible then we reach to $p_G = p_L$ and $dp_G/dt = dp_L/dt$. This is the case of homogeneous two-phase mixture and calculation of equation (4) may be simplified to the same one given by Brenenn [8].

Figure 1 shows, the average sonic speed in bubble-liquid mixture evaluated by equation (4) under different gas volume fraction. The solid line with circles demonstrates the case of bubble cavitation, where a bubble-water mixture including micro-bubbles of $\alpha_G = 0.001$ is considered and the initial bubble radius is given to be R_0 = 5×10^{-6} m under the atmospheric pressure when T = 293.15 K. The surrounding liquid pressure is decreased by imposing a negative pressure pulse and the variation of bubble radius is then estimated by equation (10). Under the assumption of no bubble coalescing and breaking the variation of gas volume fraction is evaluated and the mean sonic speed at different bubble

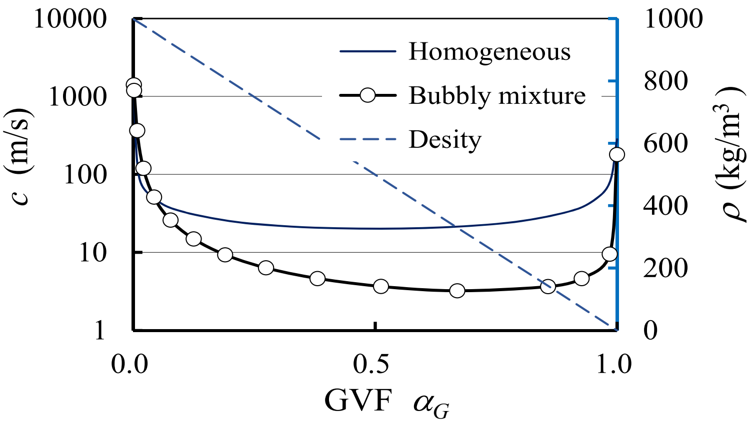


Figure 1. Average sonic speed in cavitating fluids.

radius is calculated. For comparison the sonic speed in homogenous mixture is also denoted by the solid line, where the gas volume fraction increases with bubble number density. When $\alpha_G = 0.58$ the sonic speed decreases to $c_M \approx 25$ m/s, which agrees to the experiment data for quasi-still bubble-liquid mixture [9]. The dashed line denotes the mixture density. The figure demonstrates that the sonic speed in cavitating bubble-liquid mixture deceases much intensively. So, it should be essential to estimate the mixture compressibility by considering the effect of bubble dynamics in the numerical simulation of intensively cavitating flow with the mixture flow method.

2.2 Governing equations for turbulent cavitating mixture flow

In consideration of the effect of fluid compressibility the URANS equations for compressible fluid are adopted as main flow governing equations. The variation of temperature caused by cavitation is thought to be very small in the whole flow field and the conservation equation of energy is omitted. Conservation equations of mass and momentum for the two-phase mixture are written as follows.

$$\frac{\partial \rho_M}{\partial t} + \boldsymbol{u} \cdot \nabla \rho_M = -\rho_M \nabla \cdot \boldsymbol{u} \tag{11}$$

$$\rho_M \left(\frac{\partial \boldsymbol{u}}{\partial t} + (\boldsymbol{u} \cdot \nabla) \boldsymbol{u} \right) = -\nabla p_L + \nabla \cdot \boldsymbol{\tau} + \boldsymbol{g} \tag{12}$$

where $\boldsymbol{u}$ denotes the mean velocity of the mixture, which is treated by neglecting the difference of liquid and gas velocities. $\boldsymbol{g}$ denotes the gravity, and $\boldsymbol{\tau}$ does the stress tensor written as follows.

$$\tau_{ij} = \mu_{M,eff} \left[\left(\nabla \boldsymbol{u} + \nabla \boldsymbol{u}^T \right) - \frac{2}{3} (\nabla \cdot \boldsymbol{u}) \boldsymbol{I} \right] \tag{13}$$

where $\boldsymbol{I}$ is the unit tensor. $\mu_{M,eff} = \mu_M + \mu_{M,turb}$ denotes effective viscosity that includes both the molecular viscosity μ_M and the turbulent viscosity $\mu_{M,turb}$ of the fluid mixture. The turbulence of the two-phase flow is evaluated by adopting k-epsilon model with a modification on cavitation effect [10].

In order to close above equations, equation (3) relating the mixture density and liquid pressure is taken into equation (11) and the following equation governing the transportation of pressure is obtained.

$$\frac{\partial p_L}{\partial t} + \boldsymbol{u} \cdot \nabla p_L = -\rho c_M^2 \nabla \cdot \boldsymbol{u} \tag{14}$$

Furthermore, the mass conversation of gas phase is employed to estimate the variation of gas volume fraction.

$$\frac{\partial}{\partial t}(\rho_G \alpha_G) + \boldsymbol{u} \cdot \nabla(\rho_G \alpha_G) = -(\rho_G \alpha_G) \nabla \cdot \boldsymbol{u} \tag{15}$$

where $(\rho_G \alpha_G)$ denotes the mass fraction of gas including in the mixture. Here evaporation and condensation caused by cavitation are supposed to be negligible for the purpose of simplification.

In high-speed water jets with intensive cavitation bubbles may expand greatly and the density of fluid mixture varies sharply in cavitation regions. Thus, strong compressible bubbly flow regions coexist with the weak compressible liquid flow region. For dealing with the great variation of compressibility the set of above flow governing equations are solved by using the CIP-CUP (Cubic-Interpolated Propagation/Combined and Unified Procedure) method based on the time splitting technique [11].

3. Computation results and discussions

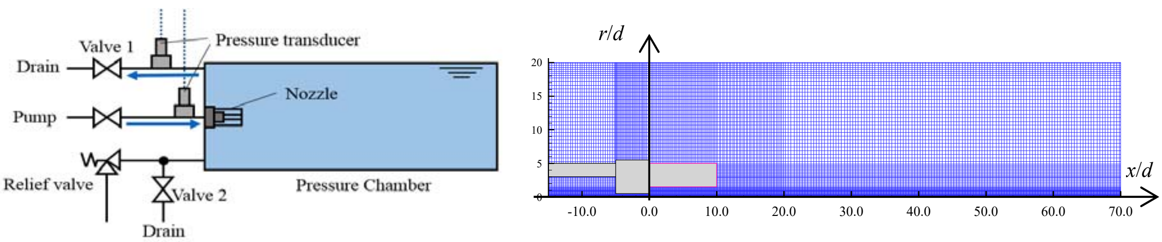
Figure 2 (a) shows the submerged water jet system to be concerned, where a sheathed orifice nozzle is set up at the bottom of a closed cylindrical chamber full of tap water (deposited one night). Pressured tap water supplied by a plunger pump is injected into the water chamber and a submerged water jet is generated at the given condition. The gauge pressure within the chamber can be adjusted to a given level up to 2.0MPa and the output pressure of the plunger pump can be adjusted within its maximum of 21.0MPa in gauge pressure according to experiment requirements. The throat diameter of the nozzle d = 1.0 mm and the length of nozzle throat is 5.0d. A pipe-like sheath is mounted at the nozzle exit. The inner diameter of the pipe-like sheath is 3.0d and its length is 10.0d.

Figure 2 (b) shows the computational domain and the structure mesh adopted for the numerical simulation. Focused on the upstream axisymmetric structure the assumption of axisymmetric flow is adopted. The computation domain is taken from 15.0d upstream of the nozzle inlet to 70.0d downstream of the nozzle exit. The width of computational domain is 25.0d in the radial direction. The computation domain is then discretized with quadrilateral structure grid. As for the boundary conditions, a pressure condition is imposed at the inlet according to the given total pressure and the intensity of turbulence was given to be 1.0% of the inflow velocity. At the outlet, a given static pressure is imposed and the Neumann condition was applied to other flow variables such as velocity etc. All the wall boundaries such as the nozzle and the sheath geometries are treated as no-slip walls by applying a universal wall function. As an index for the similarity of cavitation dynamics, the cavitation number σ for the present water jet device is defined as follows.

$$\sigma = \frac{p_o - p_v(T_\infty)}{P_i - p_o} \tag{16}$$

where P_i denotes the injection pressure, p_o the static surrounding pressure at the nozzle exit and p_v the saturated vapor pressure under the reference temperature T_∞.

Figure 3 shows, as an example, instantaneous distributions of gas volume fraction obtained via numerical simulation when the injection pressure $P_i = 1.1$ MPa and the discharge pressure p_o = 0.1 MPa. The cavitation number $\sigma \cong 0.1$ and the Reynolds number $Re \cong 4.8 \times 10^4$ for the concerned case. The red colour denotes gas phase and the blue one do the liquid phase. The area demonstrated by red and green colours present axial section of cavitation clouds, where the value of α_G is relative high. The dark



(a) Scheme of water jet device (b) Computational domain and grids

Figure 2. Scheme of submerged water jet and computation domain of numerical simulation.

vectors denote the mean velocity in local flow field. The result shows that cavitation occurs at the entrance of nozzle throat and cavitation cloud expands and develops while flowing downstream along the boundary layer. Developed cavitation clouds split into small blocks and sheds downstream periodically near the sheath exit. As shown in figure 3(a), heavy cavitation cloud denoted by red colour starts to stretch from its minimum length and gradually expands to the out of the pipe-like sheath as shown in figure 3(b). Then, developed cavitation cloud splits into small blocks with collapsing of bubbles and finally shed downstream as shown in figure 3(c). In this way, cavitation clouds denoted by red-green colours stretch out of the sheath and shed downstream periodically. The capability of present model to capture the unsteady shedding of cavitation cloud is demonstrated [6].

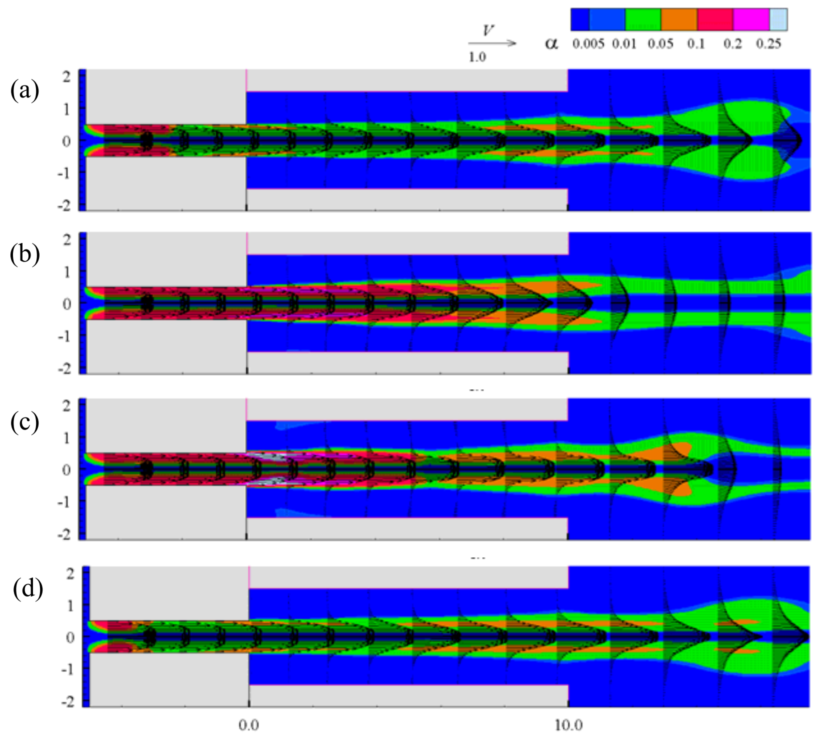


Figure 3. Instantaneous flow distribution of cavitating jet in time sequence: (a) $t = 23.2$ ms, (b) $t = 23.6$ ms, (c) $t = 23.9$ ms and (d) $t = 24.1$ms.

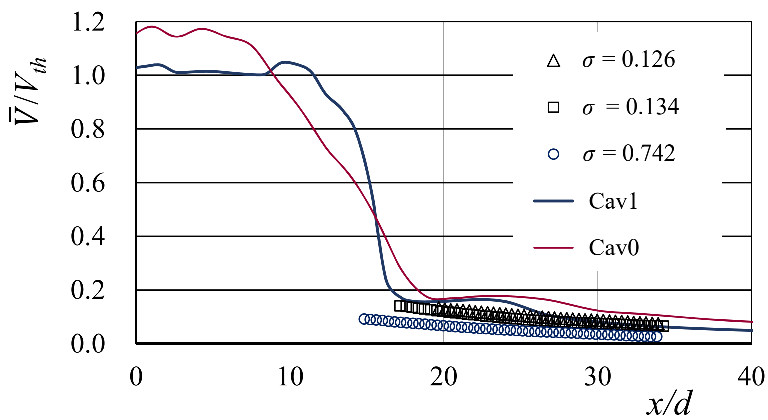
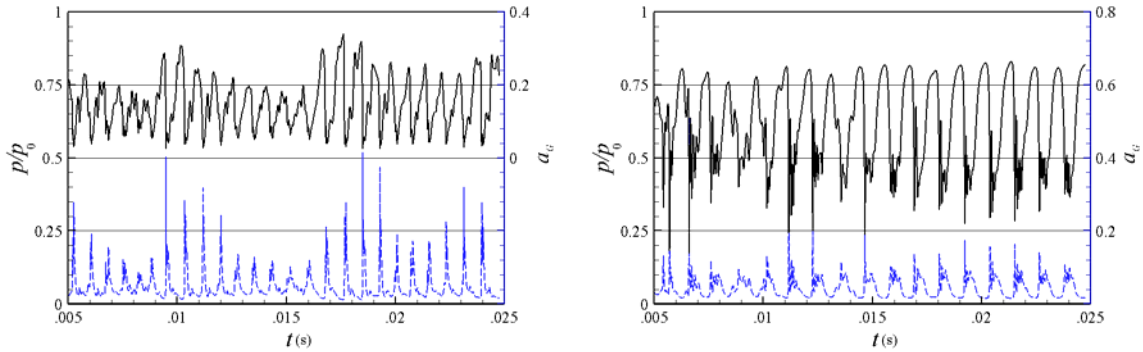


Figure 4. Decay of average velocity along central axis: comparison of numerical results (lines) to experiment data (symbols).



(a) Results of bubble cavitation model (b) Results of homogeneous mixture model

Figure 5 Temporal variations of pressure and GVF at the reference position ($x/d = -4.5$, $r/d = 0.45$) near the wall just behind the nozzle throat entrance.

To clarify the effect of mixture compressibility estimation under different conditions as shown in figure 1 simulations are performed also by employing the homogenous model where the effect of bubble interface is neglected [12]. Figure 4 shows, the distribution of timely averaged velocity along the central axis, where the horizontal axis *x*/*d* denotes the no-dimensional distance from the nozzle exit. The thick line, Cav1, presents the result with the simplified bubble-cavitation model, and the thin line, Cav0, does the result by the homogeneous mixture model when the effect of bubble surface is neglected. For comparison, experiment data measured by PIV when $\sigma = 0.126, 0.134,\ 0.74$ are denoted with symbols Δ, □ and ○ respectively. It should be mentioned that cavitation clouds with heavy bubble concertation appear within the nozzle sheath and it near downstream when the cavitation number is decreased to 0.2 and below. It is very difficult to observe the interior of bubble clouds and it is nearly impossible to evaluate its velocity field for scattering reflection of light on bubble surfaces even by using a strong sheet laser light. Therefor the shooting area of PIV was set apart from the exit of nozzle sheath and only velocity data in the downstream of *x*/*d* =15~35 have obtained. The figure shows that computation results agree with experiment data approximately and the reliability of present computations has been demonstrated. According to the results we understand that the core velocity of water jet keeps to a high level within the sheath and then decays quickly at the near downstream of sheath exit. It can be also noticed that the core velocity within the sheath predicted by the homogeneous mixture model is higher than that by the bubble cavitation model.

Figure 5 show the temporal variations of GVF and pressure and at a given monitor position ($x/d = -4.5, r/d = 0.45$) located at the entrance of nozzle throat near the wall, where (a) presents computation results of the bubble cavitation model and (b) does results of the homogeneous model. The black solid line denotes the variation of GVF, α_G, and the blue dashed line does the variation of liquid pressure, p, in absolute value. The horizontal axis denotes time. Figure 5 (a) shows that the dimensionless pressure decreases to the level of 0.05 and GVF increase to 0.2 approximately. GVF and pressure fluctuate periodically almost at the same frequency. Also, it may be noticed that GVF deceases much fast compared to its increase process and an impulse pressure is released every time while GVF reaches to its minimum. That is to say, the effect of bubble collapsing is captured in some extent by the present method.

Comparing figure 5 (a) and (b) we understand that pressures evaluated by both the methods are almost at the same level but the pressure pulsation shown in figure 5 (b) is higher. The average value of GVF estimated by the homogeneous model reaches to the level of 0.5 shown in figure 5 (b), which is much higher than that shown in figure 5 (a). As shown in Figure 4 the core velocity in the sheath (*x*/*d* < 10) evaluated by the homogeneous model is relative higher and the cavity region seems to be overestimated. Thus, the homogenous model reached by neglecting the effect of interface shows a tendency to estimate GVF excessively and pressure fluctuation inadequately. The result reveals that it becomes necessary to evaluate the fluid compressibility carefully by considering the effect of bubble dynamics in modeling bubble-liquid mixture with intensive cavitation.

4. Conclusions

Focused on the unsteady behavior of intensively cavitating jets numerical analyses were carried out by applying the simplified compressible mixture-flow bubble-cavitation model. The periodically shedding of cavitation clouds in submerged water jet is captured acceptably. The core velocity variation evaluated agrees with experiment data of measured by PIV approximately and the reliability of the present method is verified. Concerning the effect of flow compressibility estimation comparison of computational results reveals that both the simplified bubble cavitation model and the compressive homogeneous mixture model are capable to capture the unsteady behavior of cavitation clouds in some extent. However, the homogenous model shows a tendency to estimate gas volume fraction of cavitation clouds excessively. It becomes necessary to evaluate the fluid compressibility by considering the effect of bubble dynamics in modeling bubble-liquid mixture with intensive cavitation.

Acknowledgments

This work was partly supported by JSPS, Grant-in-Aid for Scientific Research (C) (No. 17K06169). The authors would like also to thank Mr. T Ito and Miss A Wakui for their assistance in experiments.

References

[1] Kalumuck K M and Chahine G L 2000 The use of cavitating jets to oxidize organic com-pounds in water, *J. Fluids Eng.* **122** 465-70
[2] Nishimura S et al. 2012 Similarity law on shedding frequency of cavitation cloud induced by a cavitating Jet, *J. Fluid Sci. & Tech.* **7**(3), 405-20
[3] Qin Z, Bremhorst K and Alehossein H 2007 Simulation of cavitation bubbles in a convergent–divergent nozzle water jet, *J. Fluid Mech.* **573**, 1-25
[4] Tran T D et al. 2015 Investigation of cavitation models for steady and unsteady cavitating flow simulation, *Int. J. Fluid Machinery and Systems*, **8**, 240-253.
[5] Iga Y and Konno T 2012 Numerical analysis of the influence of acceleration on cavitation instabilities that arise in cascade, *Int. J. Fluid Machinery and Systems*, **5**, 1-9
[6] Peng G, Yang C, Oguma Y and Shimizu S 2016 Numerical analysis of cavitation cloud shedding in a submerged water, *J. Hydrodynamics (B)*, **28**(6), 986-93
[7] Singhal A K, Athavale, M M, Li, H and Jiang, Y 2002 Mathematical basis and validation of the full cavitation model, *J. Fluids Eng*. **124**(3), 617–24
[8] Brennen, C E 1995 *Cavitation and bubble dynamics*, (New York: Oxford Univ. Press)
[9] Karplus, H B 1957 Velocity of sound in a liquid containing gas bubbles, *J. Acoustics Soc. America* **29**, 1261
[10] Peng G, Shimizu S and Fujikawa S 2011 Numerical simulation of cavitating water jet by a compressible mixture flow method, *J. Fluid Sci. & Tech*. **6**, 499-509
[11] Peng G, Ishizuka M and Hayama S, 2001 An improved CIP-CUP method for submerged water jet flow simulation, *JSME Int. J. (B)*, **44**(4), 497-504
[12] Park S and Rhee S H, 2015 Comparative study of incompressible and isothermal compressible flow solvers for cavitating flow dynamics, *J. Mech. Sci. Tech*. **29**(8), 3287~96