Numerical simulation of high-speed cavitating flows in submerged water jet

G Peng and S Shimizu
Department of Mechanical Engineering, College of Engineering, Nihon University Koriyama, Fukushima, 963-8643, JAPAN
E-mail: peng@mech.ce.nihon-u.ac.jp

Abstract. Properties of existing cavitation models are discussed and a compressible mixture flow method based on a simplified estimation of bubble cavitation is then presented for numerical simulation of high-speed water jets accompanied by intensive cavitation. Two-phase fluid media of cavitating flow are treated as a mixture of liquid and bubbles, and the mean flow is computed by solving RANS equations for compressible fluids considering the effect of bubble expansion and contraction. The intensity of cavitation is evaluated by the gas volume fraction, which is governed by the compressibility of bubble-liquid mixture corresponding to the status of mean flow field. Numerical results of cavitating water jet issuing from a submerged nozzle are presented and its applicability to intensively cavitating jets is demonstrated.

1. Introduction
High-speed water jets in air or water have been widely utilized for burry removal and cleaning of complicated mechanical products, cutting of solid materials, and surface improvement of materials in various industry fields [1]. Among them, high-speed water jet injected into still water, which is called submerged water jet, has been applied to peening of metal material by using cavitation impacts caused by the collapse of cavitation bubbles [2, 3]. It has received much attention also in environmental industry for the possibility of applications to decomposition of toxic substances and purification of sewages. Until now, many experimental studies on cavitating water jets have been made concerning jet driven pressure, shape and size of a nozzle, cavitation number etc. [4, 5]. However, the structure of cavitating jet and the behavior of unsteady cavitation bubbles are still unclear for the difficulty to observe the interior of cavitating flows [6]. With the purpose of performance prediction and efficient design of high-speed water jet devices much attention has been attracted to the numerical simulation of cavitating flows.

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, where a distinct interface may not exist in the case of high void-fraction micro bubble cloud. 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 carried out under certain simplifications such as inviscid and steady flows [7, 8].
Numerical methods used for treating cavitating flows may be classified into two categories: the interface capturing method and the continuum modeling method. Interface capturing methods assume that there is a clear and distinct interface between the liquid and gas phase, which can be determined via an iterative procedure. Their applications are limited to simpler problems where the cavity can be described as a well-defined closed volume of pure gas [9-11]. On the other hand, the continuum modeling methods do not make attempt to track the liquid-gas interface but treats the fluid media as a two-phase mixture whose density varies continuously between the liquid and vapor extremes. These methods are becoming more and more popular because they can be applied to turbulent flows often encountered in most industrial applications. The continuum methods are implemented using different approaches: one-fluid method, two-fluid method and their hybrid one [12].

The two-fluid method assumes that both phases co-exist at every point in the flow field and each phase is governed by its own set of conservation laws. So, the motions of liquid and bubbles are treated separately and two sets of governing equations derived for liquid and gas phases are solved interactively [13, 14]. Usually, the volume change of bubbles is described by the Rayleigh-Plesset equation or similar ones, and the exchanges of mass, momentum, and energy are estimated explicitly as transfer terms. So, these models can take into account the physical details occurring in the cavitation phenomenon. However, the transfer terms strongly depends upon the estimation of the interaction between the liquid and bubbles, which is very difficult since there is not a general physical model describing the interaction of liquid and bubbles. This kind of methods has been used for flows accompanying weak cavitation [15].

Differing from the two-fluid methods, one-fluid methods treats the fluid media of cavitating flows as a homogeneous fluid mixture by neglecting the velocity slip between the liquid and gas phases. The physical properties of the homogeneous mixture are determined by the volume fraction of gas phase. Therefore, the fluid is pseudo-single phase one whose density may vary sharply over a range from liquid to gas one. For evaluation of the mixture density, Delannoy and Kueny [16] proposed a method to link directly the density to the static pressure, and Iga et al. [17] improved it by using the equation of state for homogeneous compressible two-phase medium under equilibrium condition. In consideration for bubble dynamics, Kubota et al. [18] proposed a bubble two-phase cavitation model by coupling an incompressible flow solver with a simplified Rayleigh-Plesset equation describing the radial motion of bubble surface. Singhal et al. [19] developed a full cavitation model by estimating the temporal influence of mass transfer through an empirical formulation for the source term of gas void fraction transport. The one fluid approach cannot reproduce strong thermodynamic or kinetic non-equilibrium effects but, it is often used for its simplicity. Generally speaking, the computation of one-fluid homogeneous models is cheaper than that of two-fluid models because there is no need to treat the motion of a mass of bubbles separately.

Hybrid methods are intermediate ones between one-fluid and two-fluid methods. This kind of models has been developed by adding a mass equation for the gas or liquid density including a cavitation source term to the one-fluid methods [20-22]. The main difficulty of these models is related to the formulation of the cavitation source term and the tunable parameters involved in the vaporization and condensation process accompanying cavitation.

The dynamics of cavitating water jet depends strongly upon the behaviour of bubbles travelling together with jet flow. For clarifying the characteristics of growth, collapse, and rebound of cavitation bubble travelling along jet flow Alehossein and Qin [23] combined the computation of the Rayleigh–Plesset equation governing bubble growth and collapse with a density-based mixture flow simulation. However, the effect of bubble growth and collapse to the flow was neglected since the results of bubble dynamics computation were not taken back to the flow analysis. Peng et al [24] presented a hybrid method by combining the Rayleigh–Plesset equation with a compressible RANS computation, where the effect of bubble dynamics to the surrounding flow fields has been accounted by adding a source term related to the bubble radius to the RANS equations. The method demonstrated the possibility to predict the intensity of cavitation impact varying unsteadily together with flow field, but
its application was limited to the cases of weak cavitation for the difficulty to fit the flow calculation to a severe variation of bubble radii in the case of intensive cavitation.

In addition to modeling cavitation, modeling of turbulence has been demonstrated to be another important issue affecting the numerical result of cavitating flow simulations. Some three-dimensional time dependent computations obtained with Large Eddy Simulations (LES) or direct numerical simulations (DNS) have been reported but they are not yet tractable for general industrial applications. The RANS method is often applied by adopting a certain turbulence model to evaluate the unsteady turbulence effect according to the mean flow quantities. However, the standard eddy-viscosity models based on the linear Boussinesq relation are known to suffer from numerous weaknesses. In particular, the over-production of eddy-viscosity reduces the development of unsteadiness and modifies the flow topology. For unsteady cavitation flows the problem of limiting the turbulent viscosity becomes determinant to capture oscillations of cavitation. Usually, arbitrary limiters are introduced in the turbulent viscosity calculation to obtain unsteady flow characteristics such as cavity shedding and re-entrant jet [25, 26].

In this work, focusing on the numerical simulation of high-speed water jets accompanied by intensive cavitation we introduce a compressible mixture flow method based on a simplified estimation of bubble cavitation as well as its application to turbulent cavitating water jets. Numerical results of cavitating water jet issuing from a submerged orifice nozzle are presented and its applicability to intensively cavitating jets is demonstrated.

2. A compressible mixture flow method for modelling of turbulent cavitating water jets

The homogeneous one fluid method has been widely applied to high-speed cavitation flows often encountered in industrial applications. Both density and pressure-based methods have been used in conjunction with certain cavitation models.

Aiming at the numerical simulation of intensively cavitating water jets, here we present a pressure-based compressible mixture flow method. The fluid media of cavitating flow are taken as a two-phase mixture of working liquid and cavitation bubbles and the density of the mixture is defined as follows by volume averaging.

$$\rho_m = \rho_L \alpha_L + \rho_G \alpha_G$$  \hspace{1cm} (1)

where \(\rho\) denotes fluid density and \(\alpha\) the volume faction. The subscripts \(L\), \(G\) and \(M\) do the liquid phase, the gas phase and the two-phase mixture, respectively. Then, the variation of mixture density is written as follows by taking the differential of above equation.

$$\frac{1}{\rho_m} \frac{d\rho_m}{dt} = \frac{\alpha_L}{\rho_L c_L^2} \frac{dp_L}{dt} + \frac{\alpha_G}{\rho_G c_G^2} \frac{dp_G}{dt}$$  \hspace{1cm} (2)

where \(c\) denotes the sonic speed and \(p\) does the working pressure. Both the liquid and the gas included in bubbles are supposed to work exponentially and the state equations are used to relate their density and working pressure. Thus, the compressibility of liquid and gas phases is written below, similarly.

$$\rho_k c_k^2 = n_k (p_k + B_k)$$  \hspace{1cm} (3)

where the subscript \(\kappa = L, G\), denotes the liquid and gas phases, and \(n\) the ratio of specific heat. \(B\) is a constant given to be \(3.049 \times 10^8\) Pa for liquid and 0 for gas. Considering engineering application the simplified relation, \(dp_L / dt \approx dp_G / dt\), is adopted by neglecting the instantaneous variation of bubble size. Then, the variation of mixture density is directly related to the liquid pressure.

$$\frac{1}{\rho_m} \frac{d\rho_m}{dt} = \frac{1}{\rho_m c_m^2} \frac{dp_L}{dt}$$  \hspace{1cm} (4)

So, the average sonic speed in the two-phase mixture is defined as follows approximately.

3
The variation of bubble radius is dependent upon the liquid pressure surrounding a bubble and the compressibility of bubble-liquid mixture varies greatly with expanding and contracting of bubble size. Thus, the compressibility of cavitating mixture varies much intensively compared to that of non-cavitating bubbly mixture.

As flow governing equations for the compressible two-phase fluid mixture, Reynolds Averaged Navier-Stokes equations for compressible turbulent flow are adopted and the temperature variation caused by cavitation is supposed to be very small. So, conservation equations of mass and momentum are employed together with the density-pressure relation derived previously. For convenience the subscript \( M \) denoting the mixture is omitted hereafter.

\[
\begin{align*}
\frac{1}{\rho_a c_a^2} \leq \alpha_c &+ \frac{\alpha_c}{\rho_c c_c^2} \\
\frac{\partial \rho}{\partial t} + \mathbf{u} \cdot \nabla \rho &= -\nabla \cdot \mathbf{u} \\
\frac{\partial \mathbf{u}}{\partial t} + \rho (\mathbf{u} \cdot \nabla) \mathbf{u} &= -\nabla p + \nabla \cdot \mathbf{\tau} + \mathbf{g}
\end{align*}
\]

where \( \mathbf{u} \) denotes the mean velocity and \( \mathbf{g} \) does the gravity. \( \mathbf{\tau} \) denotes the stress tensor and its components are given as follows.

\[
\tau_{ij} = 2\mu S_{ij} - \frac{2}{3} \mu (\nabla \cdot \mathbf{u}) \delta_{ij} - \rho u_i u_j
\]

in which \( i \) and \( j \) denote respectively three components of the coordinates. \( \delta_{ij} \) denotes the Kronecker delta and \( S_{ij} \) does the strain tensor. \( \mu \) does the mean viscosity of the two-phase mixture. \(-\rho u_i u_j\) does the Reynolds stress which is related to mean velocity filed via a turbulence model. Here, the RNG \( k-\varepsilon \) turbulence model for high Reynolds number flow is adopted to take account of the turbulence effect and the Reynolds stress is given as follows.

\[
\frac{\rho u_i u_j}{\rho} = 2\mu S_{ij} - \frac{2}{3} \mu (\nabla \cdot \mathbf{u}) \delta_{ij} + \rho k \delta_{ij}
\]

However, the turbulence energy of bubbly cavitating flow may be absorbed corresponding to expanding and contracting of cavitation bubbles, and the eddy viscosity in cavitating region becomes much weak compared to the liquid flow region. Thus, a modification of eddy viscosity show below is introduced to take account of the effect of cavitation on the flow turbulence [26].

\[
\mu = \chi \rho_a C_\mu k \varepsilon
\]

where \( \mu \) denotes the eddy viscosity, \( k \) and \( \varepsilon \) do the turbulence energy and the turbulence dissipation rate. \( C_\mu \) is a constant of turbulence model. \( \chi \) is a modification coefficient given as follows.

\[
\chi(\alpha_c) = \frac{1 + (c_\rho - 1)(1 - \alpha_c)}{\alpha_c + c_\rho (1 - \alpha_c)}
\]

in which \( c_\rho = \rho_l/\rho_g \). \( k \) and \( \varepsilon \) are evaluated by solving their transport equations together with the other governing equations with the CIP-CUP algorithm [27].

3. Application to high-speed cavitating water jets

Water jets issuing from a sharp-edged circular orifice are treated under different cavitation numbers. Geometry of the orifice are given to be that \( D/d = 2.88 \) and \( L/d = 5.0 \), where \( D \), \( d \), and \( L \) denote the inlet diameter, the throat diameter, and the orifice length, respectively. As an index for the similarity of cavitation dynamics, the cavitation number \( \sigma \) is usually defined as follows for a submerged water jet.
where $P_{in}$ denotes the jet driving pressure, $p_o$ the static surrounding pressure at the nozzle exit and $p_v(T_{\infty})$ the saturated vapor pressure at the reference temperature $T_{\infty}$.

Figure 1, as a sample, shows the development of a cavitating jet of which the cavitation number $\sigma = 0.1$ and the jet driven pressure $\Delta p \approx 1.0 \text{MPa}$. Figure 1 (a) to (d) present the contour maps of gas volume fraction in a sequence time, where $\alpha_G$ increases to 0.8 locally with developing of jet flow. As shown in figure 1 (a) cavitation takes place initially at the entrance of nozzle throat as well as the central area of starting vortex near the nozzle exit. According to figure 1 (b) to (d) we understand that $\alpha_G$ increases gradually and cavitation bubbles mainly concentrate in the shear layer between the main jet and surrounding water.

Figure 2 shows the contour maps of gas volume fraction in developed jets at different cavitation numbers. Comparing the results we may confirm that the maximum value of $\alpha_G$, which indicates the local intensity of cavitation, keeps almost the same when the cavitation number is decreased from 0.1 to 0.02. However, the area of strong cavitation, which is denoted by high value $\alpha_G$, increases with the decrease of cavitation number.

Figure 3 shows the variation of discharge coefficient with the cavitation number, where the vertical axis is normalized by the discharge coefficient $c_d$ of no-cavitating jet. The circles indicate computational results obtained by present method and the solid line refers to Nurick’s experiment data [28]. The figure confirms that the discharge coefficient decreases to 80.0% approximately compared to the case of no-cavitating jets when $\sigma$ is decreased to 0.02.

\begin{equation}
\sigma = \frac{P_{in} - p_v(T_{\infty})}{P_{in} - p_o}
\end{equation}
4. Summary

Expanding and collapsing of bubbles is an important matter needs to be carefully considered in the numerical analysis of cavitating flows. Properties of existing methods for modelling of cavitating flow including bubble dynamics effect and turbulence closures has been discussed, and then a compressible mixture flow method based on a simplified estimation of bubble cavitation has been presented focusing on the numerical simulation of high-speed water jets accompanied with intensive cavitation.

High-speed water jets issuing from a submerged orifice nozzle have been treated by the method and its applicability to intensively cavitating water jets has been confirmed. The results demonstrate that: (1) Cavitation takes place initially at the entrance of orifice when the cavitation number is decrease to a certain critical value and cavitation bubbles mainly concentrate in the shear layer surrounding the jet. (2) The average intensity of cavitation denoted by the mean value of gas volume fraction within the cavitation area increases generally with decreasing of the cavitation number.

Acknowledgments

This work was partly supported by JSPS, Grant-in-Aid for Scientific Research (C) (N0. 22560177).

References

[1] Shimizu S and Peng G 2009 Water jetting technology for LOHAS (Tokyo: International Academic Printing Co Ltd)
[2] Soyama H, Kusaka T and Saka M 2001 J. materials sci. Lett. 20 1263-65
[3] Soyama H and Takakuwa O 2011 J. Fluid Sci. & Tech. 6 510-21
[4] Yamaguti A and Shimizu S 1987 J. Fluids Eng. 109 442-7
[5] Foldyna J, Sitek L, Svehla B and Svehla S 2004 Ultrasonics Sonochemistry 11 131-7
[6] Gopalan G, Katz J and Knio O 1999 J. Fluid Mechanics 398 1-43
[7] Franc J P and Michel J M 1985 J. Fluid Mechanics 154 63-90
[8] Lemmonier H and Rowe A 1988 J. Fluid Mechanics 195 557–80
[9] Yu P, Ceccio S L and Tryggvason G 1995 Physics of Fluids 7 2608-16
[10] Sussman M 2003 J. Computational Physics 187 110-36
[11] Takahira H, Matsuno T and Shuto K 2008 Fluid Dyn. Res. 40 510-20
[12] Goncalves E and Patella R F 2009 Computers & Fluids 38 1682–96
[13] Saurel R and Lemeyayer O 2001 J. Fluid Mechanics 431 239-71
[14] Yano T, Egashira R and Fujikawa S 2006 J. Physical Society of Japan 75 104401
[15] Tamura Y and Matsumoto Y 2009 J. Hydrodynamics (B) 21 41-6
[16] Delannoy Y and Kueny J 1990 *Proc. cavitation & multiphase flow forum*, ASME FED-98 153-8
[17] Iga Y, Nohmi M, Goto A, Shin B R and Ikohagi T 2003 *J. Fluids Eng.* **125** 643-51
[18] Kubota A, Kato H and Yamaguti H 1992 *J. Fluid Mechanics* **240** 59-96
[19] Singhal A K, Athavale M M, Li H and Jiang Y 2002 *J. Fluids Eng.* **125** 643-51
[20] Kunz R F et al. 2000 *Computers & Fluids* **29** 849–75
[21] Iben U, Wrona F, Munz C-D and Beck M 2002 *J. Fluids Eng.* **124** 1011-17
[22] Peng G, Egashira R, Yano T and Fujikawa S 2007 *Proc. ASME FEDSM2007 (San Diego, California, USA, 30 July-2 August 2007)* 37420
[23] Alehossein H and Qin Z 2007 *Int. J. Numerical Methods in Eng.* **72** 780–807
[24] Peng G, Egashira R, Yano T and Fujikawa S 2007 *Proc. 1st Int. Coll. on Dynamics, Physics & Chemistry of Bubble & Gas-Liquid Boundaries (Hokkaido, Japan, 25-28 September 2007)* 2512
[25] Delgosha C O, Patella F R and Reboud J L 2002 *J. Turbulence* **3** 58
[26] Peng G, Shimizu S and Fujikawa S 2011 *J. Fluid Sci. & Tech.* **6** 499-509
[27] Peng G, Ishizuka M and Hayama S 2001 *JSME Int. J. (B)* **44** 497-504
[28] Nurick W H 1976 *J. Fluids Eng.* **98** 681-687