A simple cavitation model for unsteady simulation and its application to cavitating flow in two-dimensional convergent-divergent nozzle

Y Yamamoto¹, S Watanabe² and S I Tsuda²
¹ Graduate School of Engineering, Kyushu University, 744 Motooka, Nishi-ku, Fukuoka, Fukuoka 819-0395, Japan
² Department of Mechanical Engineering, Kyushu University, 744 Motooka, Nishi-ku, Fukuoka, Fukuoka 819-0395, Japan

E-mail: fnnabc@mech.kyushu-u.ac.jp

Abstract. In this paper, a simple cavitation model is developed under the framework of homogeneous one-fluid model, in which the perfect mixture of liquid and vapor phases is assumed. In most of conventional models, the vapor phase is considered as a dispersed phase against the liquid phase as a continuous phase, while in the present model, two extreme conditions are considered: for low void fraction, dispersed vapor bubbles in continuous liquid phase, while for high void fraction, dispersed droplets in continuous vapor phase. The growth of bubbles and droplets are taken into account in the mass transfer between vapor and liquid phases, and are switched according to the local void fraction. The model is applied for the simulation of cavitating flow in a two-dimensional convergent-divergent nozzle, and the result is compared with that using a conventional model. To enhance the unsteadiness of cavitation due to the instability at the cavity interphase, the turbulent shear stress is modified depending upon the continuous phases in combination with the proposed cavitation model, which drastically reduces the turbulent viscosity for high void fraction region. As a result, the unsteadiness of cavitation observed in experiments is well reproduced.

1. Introduction
The recent rapid progress of computer science has enabled us to simulate the cavitating flow and achieve qualitative, and in some extent, quantitative agreements with actual observed flows. However, such cavitation CFD (Computational Fluid Dynamics) still often fails to predict the cavitation performance even in simple cases of cavitating hydrofoil (Kato [1]). Moreover, still due to the limited computer resources, Reynolds-Averaged Navier-Stokes (RANS) simulation is often used to simulate the unsteady cavitating flow. However, the reproduction of the unsteadiness of cavitation is very limited, and key unsteady cavitation phenomena such as a re-entrant jet which develops beneath the sheet cavity and the formation of cloud cavity can never be simulated with RANS model developed based on incompressible flow. On the other hands, Reboud et al. [2]considered that the incompressible turbulence model overestimates the turbulent viscosity in the two-phase region, and has proposed the well-known Reboud correction to cut the turbulence viscosity in most of two phase flow region. With the tuned parameter, this correction method has successfully reproduced the major unsteadiness of cavitation in cases of a two-dimensional convergent-divergent nozzle [2], a two-dimensional hydrofoil [3], and a three-dimensional twisted hydrofoil [4].
The cavitating flows in two-dimensional convergent-divergent nozzles have been studied by many researchers, because of their simple structures but revealing important unsteady natures of cavitating flow such as re-entrant jet evolutions and vertical cloud cavity shedding. Stutz and Reboud [5], [6] have measured, by using a double optical probe, two-phase structures inside the cavity on the throat section of convergent divergent nozzle. Their nozzles have a common convergent angle of 43° but two different diverging angles of 4° and 8°. Keil et al. [7] have studied an aggressiveness of collapse of cloud cavitation in this kind of nozzle for the wide variety of Reynolds number. In our previous study [8], we investigated the unsteady character of cavitation in two-dimensional convergent-divergent nozzles with divergent angles of 4.0 and 8.4°. In both nozzles, we could not observe the re-entrant jet nor large scale cloud cavities, probably due to low Reynolds number flow compared to the other studies, while we observed small cloud cavity shedding from the trailing edge of sheet cavity as an essential character of cavitating flow in this kind of nozzle.

In the present study, we propose a simple cavitation model which considers virtually the interface between liquid and vapour phases in the framework of homogeneous one-fluid model, in which a subgrid-scale model of bubbles in the liquid phase and that of droplets in the vapour phase are employed. Mass source terms of liquid/vapour phases are considered through the growths and collapses of bubbles and droplets which are locally switched depending upon the local void fraction. In conjunction with this treatment of continuum phase, we also switch the eddy viscosity as well as the molecular viscosity depending upon whether the continuum phase is vapour or liquid. By doing so, it is expected to reproduce the strong unsteadiness of cavitating flow such as the generations of cloud cavities from the trailing edge of sheet cavity even if incompressible two-equation RANS turbulence model is employed. In this paper, the proposed model is applied for the simulation of cavitating flow in a simple two-dimensional convergent-divergent nozzle. The effects of the choices of the mass source terms for bubbles and droplets as well as the turbulence model modification on the global motion of unsteady cavitating flow are investigated.

2. Numerical method
A numerical simulation was carried out by using an open source software, OpenFOAM. Several cavitation models are implemented in OpenFOAM, among which we employ interPhaseChangeFoam (IPCF) as a base solver. This solver is an incompressible Navier-Stokes one with homogeneous cavitation model considering the phase change between liquid and vapour phases. Besides the momentum equation of the mixture and the transport equations of turbulence properties, IPCF solves the following continuity equation and the mass conservation of liquid phase as follows

\[ \frac{\partial \rho_j}{\partial t} + \nabla \cdot (\rho_j \mathbf{u}_j) = \frac{1}{\rho_l} \left( \dot{m}^l + \dot{m}^v \right) \quad (1) \]

\[ \frac{\partial \alpha_l}{\partial t} + \nabla \cdot (\alpha_l \mathbf{u}_j) = \frac{1}{\rho_l} \left( \dot{m}^l + \dot{m}^v \right) \quad (2) \]

where \( x \) is a Cartesian coordinate, \( u_j \) is velocity component in the \( x_j \) direction, \( \rho_l \) and \( \rho_v \) are densities of liquid and vapour phases, \( \alpha_l \) is a volume fraction of liquid phase. The source terms in the above equations, \( \dot{m}^l \) and \( \dot{m}^v \), are mass transfer rates between two phases due to evaporation and condensation, which are modelled in the following section.

2.1. Cavitation model

2.1.1. Schnerr-Sauer model (SS). Schnerr-Sauer (SS) model [9] implemented in OpenFOAM-2.2.1 is employed as a base model for this study, which has been used in many studies (for recent examples, [10] and [11]). In SS model, the vapour bubbles are always considered as dispersed phase in continuum
liquid phase even for very large void fraction region. The SS model is based on the Rayleigh equation as follows

\[ R \frac{d^2R}{dt^2} + \frac{3}{2} \left( \frac{dR}{dt} \right)^2 = \frac{p - p_v}{\rho_l} \]

(3)

where \( R \) is a bubble radius, \( p \) is pressure, \( p_v \) is saturation pressure. If we neglect the second derivative term, the bubble radius growth can be represented by the following equation.

\[ \frac{dR}{dt} = \text{sign}(p_v - p) \sqrt{\frac{2}{3} \frac{|p - p_v|}{\rho_l}} \]

(4)

In this model, the void fraction \( \alpha_v = 1 - \alpha_l \) represents the vapour volume per unit mixture volume and the bubble radius is derived from \( \alpha_v \) as follows.

\[ \alpha_v = \frac{n_{\text{mc}} 4\pi}{3} \frac{R^3}{1 + n_{\text{mc}} 4\pi} \]

(5)

\[ R = \frac{\alpha_v}{\sqrt{\frac{4}{3} \pi n_{\text{mc}} (1 - \alpha_v)}} \]

(6)

where \( n_{\text{mc}} \) is the number of bubbles per unit liquid volume. From those equations, the mass source terms of SS model can be obtained as

\[ \dot{m}^+ = C_e \frac{\rho_l \rho_v}{\rho} \alpha_v (1 - \alpha_v) \frac{3}{R} \sqrt{\frac{2}{3} \frac{(p - p_v)}{\rho_l}} \quad p > p_v \]

(7)

\[ \dot{m}^- = -C_e \frac{\rho_l \rho_v}{\rho} \alpha_v (1 - \alpha_v) \frac{3}{R} \sqrt{\frac{2}{3} \frac{(p - p_v)}{\rho_l}} \quad p < p_v \]

(8)

where \( C_e \) and \( C_m \) are condensation and evaporation coefficients and \( \rho = \rho_l (1 - \alpha_v) + \rho_v \alpha_v \) is the mixture density. In the present study, the basic parameters in this model are set as \( n_{\text{mc}} = 1.6 \times 10^{13} \text{m}^{-3} \), and \( C_e = C_m = 1.0 \).

2.1.2. Bubble-Droplet 1 model (BD1). Since applying the dispersed bubble models for large void fraction regions seems to be inappropriate, we consider another extreme case in which vapour phase contains more or less liquid droplets. In this study, this extreme condition near the void fraction is unity is taken into account in our model, at which we treat vapour phase as a continuum media. Figure 1 shows the conceptual drawing of our model.

This model virtually considers the interface between liquid and vapour phases as the iso-surface of void fraction \( \alpha_{v,\text{int}} \) and in this study \( \alpha_{v,\text{int}} \) is set to be 0.5 for simplicity. When the local void fraction satisfies \( \alpha_v < \alpha_{v,\text{int}} \), the vapour phase is treated as dispersed phase, while \( \alpha_v > \alpha_{v,\text{int}} \) the liquid phase is treated as dispersed phase. The mass transfer between vapour and liquid are dominated by that occurs at the surfaces of the bubbles/droplets, then the mass transfer rates \( \dot{m}^+ \) and \( \dot{m}^- \) are switched depending upon the local void fraction \( \alpha_v \). In the model “Bubble-Droplet 1 (BD1)”, the SS model, i.e. equation (7) and (8), is adopted for \( \alpha_v < \alpha_{v,\text{int}} \), while for \( \alpha_v > \alpha_{v,\text{int}} \) the phase change at the surface of droplet is considered using Schrage’s mass flux \( \dot{M} \) [12], expressed as follows
\[ M = \frac{k}{\sqrt{2\pi R_T}} (p_v - p) \]  

(9)

where \( k \) is an evaporate/condensate coefficient, \( R_T \) is a gas constant, and \( T \) is temperature. The value of \( k \) was set to 0.4 throughout this computation. By considering the phase change through the total surface area of droplets, the mass transfer rate can be obtained as

\[ m = m^+ + m^- = \frac{k}{\sqrt{2\pi R_T}} \frac{3(1 - \alpha_v)}{R_d} (p_v - p) \]  

(10)

where \( R_d \) is a radius of droplet, which can be calculated by

\[ R_d = \left(\frac{\frac{4}{3} \pi n_{nuc} (1 - \alpha_v)}{\frac{4}{3} \pi n_{nuc} \alpha_v}\right)^{\frac{1}{3}} \]  

(11)

Since we do not consider any collisions and fissions of droplets/bubbles and any nucleation and destructions, \( n_{nuc} \) is constant and common for bubbles and droplets.

Since the cavity interface can be virtually treated, the mass transfer at the cavity interface is supposed to be possibly treated, which remains for our future study.

2.1.4. Bubble-Droplet1 viscosity filtering model (BD1VF). Throughout the present computations, the standard \( k-\varepsilon \) model was used. However, it is known that key unsteady cavitation phenomena such as a re-entrant jet which develops beneath the sheet cavity and the formation of cloud cavity can never be simulated with incompressible RANS turbulence model. To enhance the unsteadiness due to the instability on the sheet cavity interface, we switch the eddy viscosity as well as the molecular viscosity by referring only to the continuum phase. This treatment may look similar to well-known Reboud correction [2], while in this study the turbulent and molecular viscosities are modified based...
on the fluid properties of continuum phase only, which is suitably applied in combination with BD1 model.

2.2. Computational method
Numerical simulations using above models were carried out for the unsteady cavitating flow in a two-dimensional convergent-divergent nozzle shown in Figure 2. The height of throat \( h \) is 5mm. The nozzle consists of a top straight wall and a bottom inclined wall with convergent and divergent angles of 43 and 8.4 degrees, respectively. This nozzle shape is similar to that used in our previous experiment [8], in which we observe a continuous cloud cavity shedding from the trailing edge of the sheet cavity.

As the boundary conditions, the velocity is fixed at the inlet \( (u_{\text{inlet}} = 1.5 \text{ m/s}) \), and the static pressure is fixed at the outlet. The non-slip flow condition is applied on the upper and lower wall of the nozzle. The number of nodes is \( 1,200 \) (along the wall) \( \times 60 \) (perpendicular to the wall). For the convection scheme, second order upwind scheme is basically used except the void fraction; for the convection scheme in equation (2), TVD scheme [13] is used to well capture the cavity interface. As for the time integration, implicit Euler scheme is employed with the variable small time step satisfying local CFL number less than 0.8.

Figure 2. Computational domain (main flow direction is from left to right)

3. Result and discussion
Figure 3 shows typical cavity shapes at four instants with the time interval of 4ms, simulated by using four models described before. The cavitation number defined using the area-averaged nozzle throat velocity \( U_{\text{throat}} \) as \( \sigma = 2(p_{\text{in}} - p_{\text{v}})/\rho_{\text{v}} U_{\text{throat}}^2 \) takes similar value of 1.63 in all cases except SS model (0.49), resulting in the time averaged cavity length of \( L_{c}/h \approx 11 - 16 \). The similar cavitation observed in the previous experiment [8] is also shown in figure 3 (a). By comparing the results between SS, BD1 and BD2 models, it is found that the sheet cavity is slightly longer for BD1 and BD2, but the unsteady behavior of cavitation is very similar: there are no cloud cavities observed but the sheet cavity slightly fluctuates. However, by applying the viscosity filtering model along with BD1 (BD1VF), we can clearly see the cloud cavity shedding similar to that observed in the experiment. These results suggest that the treatment of the turbulent shear stress is important rather than the cavitation model, whereas the concept of the viscosity filtering treatment in this study is conceptually suitable to the proposed cavitation model BD1 and BD2.

Figure 4(a) shows the results of the FFT analysis of pressure fluctuation measured 23 \( h \) downstream on the upper wall in the case with \( L_{c}/h \approx 11 - 15 \). In this figure, the strong peak around 40Hz (60Hz in SS model) and its harmonics can be clearly seen for SS, BD1 and BD2. This is associated with the cavity volume change observed in figure 3 (b)-(d), which numerically causes surge-like oscillation of liquid column downstream of the sheet cavity. For the BD1VF model, the every frequency component is significantly larger than those for SS, BD1 and BD2, but we can clearly see the broad-banded frequency peaks around 45Hz, which is caused by the continuous cloud cavity shedding from the trailing edge of the sheet cavity. Similar tendency could be found for the other conditions with various sheet cavity lengths.

Figure 4 (b) shows the comparisons of Strouhal number \( St = f h/U_{\text{throat}} \) of the cloud cavity shedding predicted by BD1VF model with that obtained by the experiment. Although the numerical
data are limited to longer sheet cavities, the tendency of the predicted Strouhal number against the cavity length is very similar to the experiment. Therefore, we believe that the present model can well simulate the unsteadiness of the cloud cavity shedding in this kind of nozzle flow.

\[ \text{Strouhal number, } St = \frac{f h}{U_{\text{throat}}} \]

\[ L_c/h = 11-16 \]

Figure 3. Comparisons of cavity behaviours among four numerical models and experiment

Figure 4. Results of FFT analysis of pressure fluctuation caused by unsteady cavitation

4. Concluding remarks

In this paper, a simple cavitation model was developed under the framework of homogeneous one-fluid model. The model is constructed considering two extreme conditions; for low void fraction, dispersed vapour bubbles in continuous liquid phase, while for high void fraction, dispersed droplets in continuous vapour phase, and according to the local void fraction, the growth of bubbles and droplets are taken into account in the mass transfer between vapour and liquid phases. To enhance the unsteadiness of cavitation due to the instability at the cavity interphase, the turbulent shear stress is modified depending upon the continuous phases in combination with the proposed cavitation model. It is found that the proposed model can well predict the major unsteadiness of cavitation, i.e. continuous cloud cavity shedding from the sheet cavity, in a two-dimensional convergent-divergent nozzle flow.
References

[1] Kato C 2011 Proc. ASME-JSME-KSME Joint Fluids Eng. Conf. (Hamamatsu, Shizuoka, Japan, 24-29, July, 2011) AIK2011-06084

[2] Reboud J. L., Stutz B, and Coutier O 1998 Proc. 3rd Int. Sym. on Cavitation (Grenoble, France, 7-10, April, 1998) 203-208

[3] Coutier-Delgosha O, Fortes-Patella R and Reboud J L 2003 ASME J. Fluids Eng. 125 38-45

[4] Ji B, Luo X W, Arndt R E A and Wu Y L 2014 Ocean Eng. 87 64-77

[5] Stutz B and Reboud J L 1997 Physics of Fluids 9 3678-86

[6] Stutz B and Reboud J L 2000 Experiments in Fluids 29 545-52

[7] Keil T, Pelz P F, Cordes U and Ludwig G 2011 Proc. WIMRC 3rd International Cavitation Forum 2011(Warwick, UK, 4 – 6 July 2011)

[8] Watanabe S, Enomoto K, Hara Y and Furukawa A 2012 Proc. 5th Int. Symp. on Fluid Machinery and Fluid Eng. (Jeju, Korea, 24-27 Oct 2012.)

[9] Schnerr G and Sauer J 2001 Proc. 4th International Conference on Multiphase Flow (New Orleans, USA, 27 May - 1 June 2001)

[10] Young Y L and Savander B R 2011 Ocean Engineering 38 1368-81

[11] Roohi E, Zahiri A P, and Passandideh-Fard M 2013 Applied Mathematical Modelling 37 6469-88

[12] Schrage R W 1953 Theoretical Study of Interphase Mass Transfer (Columbia: Columbia University Press)

[13] Bram V L 1974 J. of Computational Physics 14 361-70