Cavitation analysis of a Delft twisted hydrofoil using multi-process cavitation model

Takahito Miki¹, Keita Fujiyama¹, Shin-ichi Tsuda², and Tomohiro Irie¹

1. Software Cradle Co. Ltd., 3-4-5 Umeda, Kita-ku, Osaka, Japan
2. Department of Mechanical Engineering, Kyushu University, 744 Motooka, Nishi-ku, Fukuoka, Japan

E-mail: takahito.miki@cradle.co.jp

Abstract. Cavitation is essentially formed of a combination of multiple processes. Most of the conventional cavitation models consider these in a single model, and the conservation equation for void fraction of cavitation bubbles is solved. To understand the relationship between cavitation and turbomachinery performance, not only void fraction, but also states of cavitation bubbles such as number and radius may become useful data. The multi-process cavitation model proposed by Tsuda and Watanabe is one of the solutions. This model has been implemented into commercial CFD software scFLOW. To validate the implemented cavitation model, cavitation around a Delft twisted hydrofoil was analyzed, and the result was compared with past studies. From a parametric study of the bubble growth coefficient in this cavitation model, it was found that 0.2 for expansion and 0.05 – 0.1 for shrinkage are appropriate. Cavitation shedding frequency and cavity volume agreed well with the past studies by using those coefficients. Since the multi-process cavitation model provides detailed cavitation bubble states, a new index is proposed to detect bubbly regions and its distribution compares well with experimental data. This implies that the multi-process cavitation model will help us to predict cavitation noise and erosion risk as well as to understand the mechanism of the cavitation phenomenon.

1. Introduction

If cavitation occurs around fluid machinery dealing with liquid, such as a screw propeller, its performance is degraded because the propeller is covered by bubbles and will be idling. Moreover, the propeller can be damaged by a shockwave generated when the bubble collapses. Cavitation, therefore, is an important phenomenon in fluid machine design and cannot be ignored. For this reason, the mechanism of cavitation has been studied, and many cavitation prediction models have been proposed. Cavitation is essentially formed from a combination of multiple processes, for example inception/collapse, expansion/shrinkage, coalescence/break-up, evaporation/condensation, etc. Most of the Eulerian cavitation models [1, 2, 3] consider these processes in a single model, and an equation for the void fraction (or quality) of cavitation bubbles is solved. However, to understand the relationship between the cavitation phenomenon and turbomachinery performance, cavitation noise, and erosion risk, not only void fraction distribution, but also detailed bubble states such as number and radius are important. In fact, cavitation can be classified into several types, e.g. sheet cavitation and cloud cavitation, and each type shows different characteristics. Cavitation models based on the Lagrangian method [4, 5] solve bubble radius, but the number of bubbles is limited by computing resources.
The multi-process cavitation model, which is an Eulerian cavitation model proposed by Tsuda and Watanabe [6], is one of the solutions to overcome this issue. This model solves six equations regarding the mean state of the cavitation bubble and treats each of the cavitation processes individually. With these unique characteristics and perspective, the multi-process cavitation model has been implemented in commercial CFD software scFLOW V2020 (Software Cradle, Osaka, Japan). The validation of the multi-process cavitation model has been performed for two-dimensional geometry [6], but it has not been validated for complicated geometry. Propeller geometry is inherently three-dimensional. In this paper, we apply the multi-process cavitation model to cavitation analysis of a Delft twisted hydrofoil [7] for practical validation. The twisted hydrofoil was originally invented to study cavitation around a marine propeller for a non-rotating condition [8]. At last, by using the unique characteristics of the multi-process cavitation model, an example of its application can be demonstrated.

2. Method

2.1. Multi-process cavitation model

The basic idea of the multi-process cavitation model is derived from the moment method, and a total of six equations are solved as follows:

\[
\frac{DM_i}{Dt} = iC_g G(P_v, R)M_{i-1} + R_{\text{max}} J_s + R_i (S_b - S_c) \quad \text{(for } i = 0, 1, 2, 3) \tag{1}
\]

\[
\frac{D(\rho_v M_3)}{Dt} = \dot{m}_v (4\pi M_2) M_3 + \rho_v \frac{DM_3}{Dt} \tag{2}
\]

\[
P_v = \rho_v R_v T \tag{3}
\]

where \(M_0\) is bubble number density \([1/m^3]\), \(M_1\) is bubble radius density \([m/m^3]\), \(M_2\) is bubble surface density \([m^2/m^3]\), \(M_3\) is bubble volume density \([m^3/m^3]\), and \(\rho_v M_3\) is bubble mass density \([kg/m^3]\). Let us call equations for \(M_i\) as bubble-state equations in this paper. \(R_{\text{max}}\) is the maximum radius of the cavitation bubble nucleus and \(R\) is the mean radius of cavitation bubbles. \(P\) is pressure, \(\rho\) is density, \(T\) is temperature, and subscripts \(v\) and \(l\) stand for vapor phase and liquid phase. \(R_v\) is the gas constant of water vapor and its value is 461.5 J kg\(^{-1}\) K\(^{-1}\). Void fraction, \(\alpha\) and quality, \(Y\) in the multi-process cavitation model are defined by the following equations:

\[
\alpha = \frac{4}{3} \pi M_3 \left( \frac{1}{1 + \frac{4}{3} \pi M_3} \right) \tag{4}
\]

\[
Y = \frac{\rho_v \frac{4}{3} \pi M_3}{\rho_l + \rho_v \frac{4}{3} \pi M_3} \tag{5}
\]

Each cavitation process is taken into consideration in the source term of the bubble-state equations. \(J_s\) is inception/collapse rate, \(S_b\) is break-up rate, \(S_c\) is coalescence rate, and \(G\) is bubble expansion/shrinkage rate. \(\dot{m}_v\) is a mass flux rate on the bubble surface corresponding to evaporation/condensation. The arbitrary modeling method is acceptable to represent each process and mostly the same modeling methods defined in the original paper [6] are adopted in this study. However, some modifications have been applied. Firstly, \(S_b\) and \(S_c\) are omitted because Tsuda et al. [9] reported that the effect of these processes could be ignored when the formulae proposed in the original paper were used. Secondly, since the modified Rayleigh-Plesset equation proposed by Kubota et al. [10] and applied for the modeling of \(G\) in the original paper had a mesh dependency issue, we use the following equation derived from the simplified Rayleigh-Plesset equation for the calculation of \(G\):

\[
G(P_v, R) = \frac{dR}{dt} = \text{sgn}(P_v - P_\infty - 2\sigma_s/R) \sqrt{\frac{2}{3} \frac{|P_v - P_\infty - 2\sigma_s/R|}{\rho_l}} \tag{6}
\]
where $\sigma_s$ is surface tension coefficient, which is 0.07275 N m$^{-1}$ for a water-vapor interface. $P_\infty$ is originally far-field liquid pressure, but local fluid pressure is used in our modeling. The growth rate coefficient, $C_g$ in the bubble-state equation exists because some terms are omitted from the complete Rayleigh-Plesset equation and also because the equation of single bubble dynamics is applied to the phenomenon of multiple bubbles. Since the purpose of this cavitation model is not to precisely simulate the growth of a single bubble dynamics but to calculate the macroscopic cavitation flow whose characteristic scale is the chord length, the simplified Rayleigh-Plesset equation, which is widely used in other cavitation models [2, 3], satisfies our objective. Equation 6 is also applied in the calculation of $J_s$, and its distribution is only dependent on the pressure distribution because $J_s$ only obeys to the square root of pressure difference with the constant parameter of $R_{\text{max}}$ in inception.

2.2. Cavitation simulation in scFLOW

Commercial CFD software scFLOW, which employs the finite volume method, solves the following equations for a cavitation flow considering fluid compressibility:

$$\frac{\partial p}{\partial t} + \frac{\partial p u_i}{\partial x_i} = 0 \quad (7)$$

$$\frac{\partial p u_i}{\partial t} + \frac{\partial p u_i u_j}{\partial x_j} = \frac{\partial \tau_{ij}}{\partial x_j} - \frac{\partial P}{\partial x_i} \quad (8)$$

$$\rho = \frac{P(P + P_c)}{K(1 - Y)P(T + T_0) + R_vY(P + P_c)T} \quad (9)$$

where $u$ is flow velocity and $\tau$ is the share stress tensor. scFLOW adopts the homogeneous approach for cavitation analysis, in which vapor phase and liquid phase are assumed to share the same motion. In this paper, the SIMPLEC scheme is chosen to couple the mass conservation equation (Eq. 7) and the momentum equation (Eq. 8). Equation 9 is the equation of state (EOS) derived from combining the ideal-gas EOS and the Tammann EOS by quality, $Y$ [11]. $K$, $P_c$, and $T_0$ are constants in the Tammann EOS, and those values are 472.27 J kg$^{-1}$ K$^{-1}$, 1.9446 $\times$ 10$^9$ Pa, and 3,837 K for water, respectively. Since water cavitation at an ordinary temperature can be assumed to be an isothermal process, the energy equation is not solved in a cavitation analysis of scFLOW. Therefore, temperature, $T$ in the EOS becomes a specified condition.

2.3. Implementation of the multi-process cavitation model in scFLOW

A calculation flowchart of scFLOW including the multi-process cavitation model is shown in Fig. 1. Since pressure is the trigger of the cavitation generation, it is essential to tightly couple pressure and void fraction, $\alpha$ using a balancing loop at each time-step in a transient cavitation analysis. It is also necessary to iteratively solve bubble-state equations locally to obtain reasonable $\alpha$ for stable computation because the CFD solver essentially needs $\alpha$, not $M_i$. However, a full balancing loop for the bubble-state equations is time-consuming.

To solve stably and to reduce computation time, we have developed a preconditioning technique for the multi-process cavitation model. In this preconditioning, the Lagrangian time derivative $\frac{D}{Dt}$ in the bubble-state equation is replaced by the Eulerian time derivative $\frac{d}{dt}$ to make the equation independent from neighboring elements. The specific calculation procedure is shown on the right side of Fig. 1. As the exact $\frac{\partial S_i}{\partial x_j}$ is challenging to calculate, it can be simplified; instead, it is solved iteratively until convergence. Note that this iterative calculation is necessary only for the cavitating region because converged solutions can be obtained immediately in the non-cavitating region. As a result, the proper initial conditions for the subsequent bubble-state equation can be obtained efficiently and the full balancing loop for the bubble-state equation can be reduced or skipped.
2.4. Visualization of the multi-process cavitation model

In cavitation experiments, we generally recognize a bubbly region as a bright white region. If we can predict this region from the bubble states calculated in the multi-process cavitation model and visualize it like experimental images, it will be useful not only to compare numerical results with experimental data, but also to accurately estimate cavitation noise or erosion risk by numerical analysis. On the first attempt, we introduce the bubble intensity, $I_B$, calculated from the following equation to detect the bubbly region:

$$I_B = \frac{\alpha}{R}$$

(10)

$I_B$ is designed to indicate a high value in the bubbly region, and according to its definition, $I_B$ becomes higher as void fraction increases and as bubbles get smaller. The dimension of $I_B$ is $m^{-1}$ as is the dimension of $M_2$. Since the multi-process cavitation model employs the concept of an extended volume which allows an overlap of cavitation bubbles [6], a high $M_2$ may be observed in the region of large bubbles which are overlapped. Here, as we want to detect the region which consists of many small bubbles, the above alternative index is introduced. The mean bubble radius, $\bar{R}$ is calculated from $M_1$, which gives a value biased toward the mode value for the bubble radius.

3. Validation conditions

We employed a Delft twisted hydrofoil 11N [7], with angle-of-attack equal to -2 degrees at both edges, for validation of the multi-process cavitation model. The span length, $l_S$ of the hydrofoil is 300 mm and the chord length, $l_C$ is 150 mm. Figure 2 shows the dimensions of the computational domain in this study. Since the hydrofoil is symmetric at the center, only one side of the domain is computed. The total number of calculation elements was 4,103,956. Inflow velocity, $U_{in}$ was set to 6.97 m s$^{-1}$ and pressure was fixed at the outlet boundary in all calculations. The free-slip boundary condition was applied to the symmetry wall and the no-slip boundary condition was set on the hydrofoil surface and on the domain wall.

First of all, a non-cavitating flow was analyzed in a steady-state calculation. Incompressible water with a density of 998.2 kg m$^{-3}$ was assumed. The shear-stress transfer (SST) k-omega model was applied for the RANS (Reynolds Averaged Navier–Stokes) turbulence calculation.
After the non-cavitating flow analysis, cavitation analysis was carried out. The cavitation number, $\sigma$ of this analysis is 1.07, calculated from a saturation pressure of 2,970 Pa and an absolute outlet boundary pressure of 29,000 Pa. It is known that large eddy simulation (LES), detached eddy simulation (DES), or RANS with eddy viscosity modification is necessary to simulate cavitation shedding around the twisted hydrofoil [12, 13]. We applied DES based on the SST k-omega model [14]. The compressibility of cavitation flow was considered by Eq. 9. The 2nd-order implicit scheme was used for time-marching, with a time-step of $10^{-4}$ s. The steady flow field of the non-cavitating condition was applied as the initial conditions of the cavitation analysis. The temperature was fixed at 24°C. The number of cavitation nuclei in the non-cavitating region was $6 \times 10^9$ m$^{-3}$, corresponding to the initial value of $M_0$. $R_{max}$ in Eq. 1 was $200 \, \mu\text{m}$, which was also the initial value for $R$. From these conditions, $J_s$ becomes highest at the cavitation inception point, where the value will be approximately $5 \times 10^{10}$ m$^{-3}$ s$^{-1}$ in the case of $\sigma = 1.07$.

In the previous paper on the multi-process cavitation model [6], it was reported that 0.1 was appropriate for the growth rate coefficient, $C_g$ to compute cavitation around NACA0015. NACA0015 is two-dimensional geometry, whereas the twisted hydrofoil is three-dimensional geometry, and known to have complex cavitation behavior. To check $C_g = 0.1$ is still appropriate for three-dimensional geometry, we tested various $C_g$ values and compared results. In some cavitation models [1, 3], the coefficient for cavitation expansion, $C_e$ and shrinkage, $C_s$ are separated. We have also followed that approach. For this validation, 0.05, 0.1, and 0.2 were chosen for $C_e$ and $C_s$, and all possible combinations, a total of 9 cases, were tested. For each case, the time-histories of lift force, $F_l$ and total vapor volume, $V_v$ were logged.

4. Results and discussion

4.1. Non-cavitating flow

The pressure distribution around the hydrofoil for the non-cavitating condition was compared with experimental data [7]. Figure 3 shows the distribution of pressure coefficient, $C_p$ along chord direction at three different span locations. In this figure, two experimental data sets are indicated by crosses; one is from Delft University of Technology and another is from École polytechnique fédérale de Lausanne (hereafter, we call them Delft and EPFL). Note that the EPFL hydrofoil is half the size of the hydrofoil in this study and half the size of the Delft hydrofoil, so data for the corresponding Reynolds number is chosen.

We have obtained good agreement with the experimental data of EPFL and there are small differences compared to the Delft data. The lift coefficient, $C_l$ of the present study is 0.426, which is a bit smaller than the 0.448 value of Delft. On the other hand, the $C_l$ values of other numerical studies [12, 13] are around 0.425, which is in good agreement with our result. A
Figure 3. $C_p$ distribution around the hydrofoil for the non-cavitating condition. Span 50% is the center of the hydrofoil.

Figure 4. Time-averaged $C_p$ distribution around the cavitating hydrofoil at 50% span position

particular $C_p$ bump around $x/l_C = 0.3$ indicates separating flow in the Delft data, but this is not observed here or in other numerical studies [13, 15]. This implies that something is not correctly modeled in the numerical studies, such as surface roughness, flow velocity vibration in the experiment, etc.

4.2. Cavitating flow

To validate appropriate $C_e$ and $C_s$ in the multi-process cavitation model, the results obtained in this study were compared with experimental data and some other numerical results. Time-averaged lift coefficient, $\overline{C_l}$ around the hydrofoil, maximum and minimum of total cavity volume, $V_v$, and the Strouhal number, $St$ of cavitation shedding are summarized in Table 1. The shedding frequency, $f$ is calculated from FFT (Fast Fourier Transform) of time-history of $C_l$ and excellent agreement has been obtained with experimental data in all cases. Comparing the cavity volume for various cases, $C_e$ seems to be the dominant parameter to determine cavity volume. In addition to this, the comparison between P7 and P9 shows that a larger cavity volume is obtained when $C_e > C_s$. Also, cases P7 and P8 gave cavity volume values closer to past studies. It is interesting that other cavitation models [3, 16] also recommend $C_e > C_s$ for the coefficient of cavitation source term. Another notable result obtained in this study is that $V_v$ was changed drastically by changing $C_e$ and $C_s$, whereas $St$ and $\overline{C_l}$ were not. Additionally, $\overline{C_l}$ was underestimated in comparison with other numerical results, even though excellent agreement was obtained for $St$ and $V_v$. These results indicate that the tail part of the cavitation region was more separated from the hydrofoil than in other studies, and $C_e$ and $C_s$ were only affecting the thickness and density of cavitation. Figure 4 shows the time-averaged pressure coefficient, $\overline{C_p}$ around the hydrofoil for P3, P5, and P7. We have confirmed that the distribution of $\overline{C_p}$ becomes closer to experimental data as the total cavity volume increases (note that cavity volume is P3 $<$ P5 $<$ P7), but $-\overline{C_p}$ is still underestimated. It may be necessary to not only adjust cavitation model parameters, e.g. number and radius of cavitation nuclei, but also necessary to control mesh size, turbulence modeling, etc. to get better results. However, it is pointed out by some researchers [17] that there is a fundamental difficulty in predicting cavitation length correctly by the homogenous approach. Since we have bubble pressure and bubble radius distributions in the multi-process cavitation model, this model has an advantage for efficient implementation.
Table 1. Time-averaged lift coefficient, $C_l = \frac{F_l}{\frac{1}{2} \rho U_{in}^2 l_s l}$, the range of total cavity volume, $V_v$, and Strouhal number, $St = f l_s / U_{in}$ for various $C_e$ and $C_s$

| Case | Reference | $C_l$ | $V_v$ (min – max $\times 10^{-6}$ m$^3$) | $St$ |
|------|-----------|------|------------------------------------------|------|
| —    | Foeth (Experiment) [7] | 0.517 | — | 0.69 |
| —    | Whitworth (DES) [12] | 0.427 | 4.5 – 10.7 | 0.75 |
| —    | Vas et al. (DDES) [13] | 0.43 | 6.6 – 14 | 0.60 |
| P1   | Present ($C_e = 0.05, C_s = 0.05$) | 0.384 | 0.8 – 1.8 | 0.68 |
| P2   | Present ($C_e = 0.05, C_s = 0.10$) | 0.383 | 0.5 – 1.8 | 0.68 |
| P3   | Present ($C_e = 0.05, C_s = 0.20$) | 0.384 | 0.4 – 1.2 | 0.63 |
| P4   | Present ($C_e = 0.10, C_s = 0.05$) | 0.386 | 2.0 – 6.9 | 0.68 |
| P5   | Present ($C_e = 0.10, C_s = 0.10$) | 0.386 | 1.8 – 6.8 | 0.71 |
| P6   | Present ($C_e = 0.10, C_s = 0.20$) | 0.382 | 2.9 – 4.8 | 0.71 |
| P7   | Present ($C_e = 0.20, C_s = 0.05$) | 0.389 | 4.1 – 11.0 | 0.66 |
| P8   | Present ($C_e = 0.20, C_s = 0.10$) | 0.390 | 3.3 – 11.0 | 0.68 |
| P9   | Present ($C_e = 0.20, C_s = 0.20$) | 0.389 | 3.7 – 9.9 | 0.68 |

Figure 5. Cavitation appearance in experiment (top) and numerical result for $C_e = 0.2$, $C_s = 0.05$ (bottom). In the numerical result, the translucent isosurface is $\alpha = 10\%$, the red isosurface is $\alpha = 40\%$, and the blue isosurface is $I_B = 1.5 \times 10^3$ m$^{-1}$.

in a non-homogenous approach, such as the two-fluid model. For the parameters tested in this study, $C_e = 0.2$, $C_s = 0.05 – 0.1$ was appropriate for application of the multi-process cavitation model to the twisted hydrofoil. Furthermore, while it is not detailed in this paper, $C_e = 0.2$, $C_s = 0.1$ was tested in a cavitation analysis of NACA0015 and results were confirmed to be consistent with the Schnerr-Sauer model [2] and the Singhal model [3], which are implemented in scFLOW.

Figure 5 shows cavitation appearance in experiments and numerical results for $C_e = 0.2$, $C_s = 0.05$. The isosurface of 10\% void fraction is shown as translucent in the numerical results. The cavitation shape of the numerical result compares well with the experimental data. Additionally, the isosurface of $\alpha = 40\%$ is shown in red on the left side of the hydrofoil, and the isosurface of bubble intensity $I_B = 1.5 \times 10^3$ m$^{-1}$ is shown in blue on the right side. A high $I_B$ region tends to appear at the back of the cavitating area, whereas a high $\alpha$ region appears at the front and
in the core of the cavitating region. The high $I_B$ region qualitatively agrees with the bright white areas in the experimental data that assumed to be the bubbly region. Also, we must note that the high void fraction area is not necessarily equivalent to the bubbly region because this area may be a sheet cavitating region. From the experimental images and from experience, sheet cavitation is observed at the cavitation inception region, and cloud cavitation is seen in the collapse region, which agrees with our numerical result. These results show that the multi-process cavitation model is capable of representing cavitation behavior and also suggest that the model has great potential to predict various cavitation phenomena such as cavitation noise, erosion risk, etc. in turbomachinery.

5. Concluding remarks

We have implemented the multi-process cavitation model into a commercial CFD software using a preconditioning technique. The implemented model was validated using a three-dimensional Delft twisted hydrofoil and the following results were obtained:

- It was found that different values should be used for the coefficient of expansion rate and shrinkage rate. $C_e = 0.2$, $C_s = 0.05 – 0.1$ was appropriate for application to the twisted hydrofoil.
- Good agreement with experimental data was obtained for total cavity volume and the Strouhal number of cavitation shedding frequency, but lift force was underestimated compared to experimental data. Further study will be necessary to get a good agreement for the lift force.
- Bubble intensity, $I_B$ was introduced for the multi-process cavitation model and utilized in the visualization of results. The image of $I_B$ showed comparatively good agreement with the experimental image for the detection of bubbly regions.

References

[1] Kunz R F, Boger D A, Stinebring D R, Chyczewski T S, Lindau J W, Gibeling H J, Venkateswaran S and Govindan T R 2000 Comput. Fluids 29 849–75
[2] Schnerr G H and Sauer J 2001 Physical and numerical modeling of unsteady cavitation dynamics Proc. Fourth Int. Conf. on Multiphase Flow (New Orleans, United States)
[3] Singhal A K, Athavale M M, Li H and Jiang Y 2002 J. Fluids Eng. 124 617–24
[4] Farrell K J 2003 J. Fluids Eng. 125 46–52
[5] Ochiai N, Iga Y, Nohmi M and Ikohagi T 2010 J. Fluid Sci. Tech. 5 416–31
[6] Tsuda S and Watanabe S 2015 Application of multi-process cavitation model for cavitating flow in cold water and in liquid nitrogen around a hydrofoil Proc. ASME/JSME/KSME 2015 Joint Fluids Eng. Conf. (Seoul, South Korea)
[7] Foeth E J 2008 The structure of three-dimensional sheet cavitation Ph.D. thesis Delft Univ. of Technology
[8] Foeth E J and van Terwisga T 2006 An attached cavity on a three-dimensional hydrofoil Proc. Sixth Int. Symp. on Cavitation (Wageningen, The Netherlands)
[9] Tsuda S, Shigeta D, Hirai T and Nohmi M 2014 Turbomachinery 42 256–63
[10] Kubota A, Kato H and Yamaguchi H 1992 J. Fluid Mech. 240 59–96
[11] Saito Y, Takami R, Nakamori I and Ikohagi T 2007 Comput. Mech. 40 85
[12] Whitworth S 2011 Cavitation prediction of flow over the delft twist 11 foil Proc. Second Int. Symp. on Marine Propulsors (Hamburg, Germany)
[13] Vaz G, Lloyd T and Guanasundaram A 2017 Improved modelling of sheet cavitation dynamics on delft twist11 hydrofoil Proc. Seventh Int. Conf. on Comput. Methods in Marine Engineering (Nantes, France)
[14] Strelets M 2001 Detached eddy simulation of massively separated flows Proc. 39th AIAA Aerospace Sciences Meeting and Exhibit (Reno, United States)
[15] Melissaritis T, Bulten N and van Terwisga T 2019 J. Fluids Eng. 46 101104
[16] Zwart P J, Gerber A G and Belamri T 2004 A two-phase flow model for predicting cavitation dynamics Proc. Fifth Int. Conf. on Multiphase Flow (Yokohama, Japan)
[17] Iga Y 2014 Numerical analysis of unsteady cavitating flow by using a modification based on an assumption of apparent phase equilibrium IOP Conf. Series: Environ. Earth Sci. vol 22 p 052010