

Article

Cavitation Evolution around a NACA0015 Hydrofoil with Different Cavitation Models Based on Level Set Method

An Yu ^{1,*}, Qinghong Tang ² and Daqing Zhou ¹¹ College of Energy and Electrical Engineering, Hohai University, Nanjing 210029, China; zhouqdhhu@163.com² College of Water Conservancy and Hydropower Engineering, Hohai University, Nanjing 210029, China; yuath2@163.com

* Correspondence: yu_an@hhu.edu.cn; Tel.: +86-15190490883

Received: 3 January 2019; Accepted: 15 February 2019; Published: 21 February 2019



Abstract: Cavitation is a complex multiphase flow phenomenon that is usually involved in marine propulsion systems, and can be simulated with a couple of methods. In this study, three widespread cavitation models were compared using experimental data and a new modified simulation method. The accuracy of the three cavitation models was evaluated regarding their steady and unsteady characteristics, such as the flow field, re-entrant jet, vortex-shedding, and so on. Based on the experimental data and numerical results, the applicability of different cavitation models in different conditions was obtained. The Kunz model can accurately capture both the adverse pressure gradient and the action of the re-entrant jet in sheet cavitation, while the full cavitation model (FCM) has an accurate prediction for the flow field structure and the shedding characteristic of cloud cavitation. Through comparing the results, the optimal selection of cavitation models for further study at different conditions was obtained.

Keywords: cavitation model; level set method; re-entrant jet; bubbles shedding

1. Introduction

Cavitation is the process and phenomenon of phase transition between liquid and vapor, which appears in the liquid or at the interface of liquids and solids due to hydrodynamic factors [1–5]. It is a complex hydrodynamic phenomenon, which has obvious unsteady characteristics, and that widely occurs in hydraulic machinery, propellers, and hydraulic engineering. Cavitation occurs when the pressure of a local liquid is reduced to a certain threshold value; for example, cavitation often occurs near turbine blades and pump blades. Cavitation often results in a reduction in efficiency and the vibration of the unit in hydraulic turbines and pumps. Thus, cavitation has become a research hotspot recently, and many researchers have been devoted to investigating cavity's occurrence, breakup, shedding, collapse, and influence factor.

Due to the high costs and technical restriction of experimental investigations, in the last few years, CFD (computational fluid dynamics) have been applied to the research of cavitation. Many methods have been presented and used in cavitation investigation, and valuable achievements have been made. Most of these methods were based on the homogeneous fluid approach, i.e. the homogeneous equilibrium flow model (HEM). The two phases of vapor and liquid are considered to be a mixed fluid with variable density, and the relative velocity between the two phases is neglected. HEM considers the weighted average values of the corresponding parameters of the vapor-liquid as the parameters of the mixed fluid. Therefore, the continuity equation and momentum equation for the mixed fluid is established. There are two main methods to evaluate the variable density at present: the barotropic

equation model (BEM) [6,7] and the transfer equation-based model (TEM) for a void ratio with the source terms modeling the mass transfer due to cavitation [8–10].

Most of these homogeneous methods didn't consider the acting force between the two phases, which is not coincident with the law of physics. Huang et al. [11] introduced the level set method into cavitation simulation. The level set method takes the surface tension into account, and many valuable achievements have been made. Thus, this method would be an effective way to treat the interface between vapor and water, and should be used in future research. Also, research has shown that the choice of the turbulence model plays an important role in obtaining a faithful simulation of unsteady behaviors of cloud cavitation since the flow is three-dimensional and contains many different length scales [12]. The direct numerical simulation (DNS) may get a precise result by considering all these scales, but this would require a high computing power capacity. Other available approaches used in engineering prediction are Large eddy simulation (LES), Detached eddy simulation (DES) and Reynolds Average Navier-Stokes (RANS). LES is an increasingly popular method in cavitation simulations, and lots of work has been done using this method [13–18]. However, the LES method also needs large computational resources. Thus, researchers developed the DES model, which combined the RANS and LES approaches. However the switch between RANS and LES is defined by the length of the largest cell size, and it is grid-sensitive. The widely used standard two-equation turbulence models (e.g. $k-\epsilon$ class) usually have a tendency to overpredict the turbulent viscosity in the region of cavity closure, and the models can hardly capture the shedding dynamics. For this reason, different modifications have been presented to reduce the turbulent viscosity for cavitating flow simulation. One of those modifications is to introduce a filter-based model (FBM) for better estimation of the viscosity [19].

Adaptive mathematical models immediately determined the accuracy of the numerical simulation, and the specific cavitation model may have advantages in this specific condition. So, that is important for us to compare the performance of different cavitation models in different patterns of cavitation. The comparison between the different cavitation models based on RANS and the conventional homogeneous method has been done, but little research has been done with a more accurate method. Since most numerical investigations would be done with more accuracy and a modified method, the objective of this research was to verify the influence of the cavitation model on the accuracy and applicability of the numerical predictions of cavitation around a NACA0015 hydrofoil with a more accurate turbulence model and multiphase method. Thus, three cavitation models—the Zwart model [8], the Full Cavitation model, [9] and the Kunz model [10]—were compared with the FBM turbulence model based on the level set method. These three cavitation models involve tunable parameters, which were tuned by means of an optimization strategy used by Mitja Morgut [20]. With the experimental data and numerical results, the applicability of these three cavitation models in steady and unsteady cavitation was summarized, and valuable information was obtained for the further study of model selection.

2. Mathematical Model

2.1. Basic Equations

The proposed simulation method is based on a homogeneous assumption, where the mixture of water and vapor is a flowing fluid. With the conventional simulation method, the acting force between the interfaces is usually ignored. However, in cavitating flow, surface tension was proved to play an important role during the cavitation evolution [11]. Thus, the level set method was adopted in this study. The basic governing equations for homogeneous flow are listed as follows:

$$\frac{\partial \rho_m}{\partial t} + \nabla \cdot (\rho_m \mathbf{U}) = 0 \quad (1)$$

$$\frac{\partial \rho_m u_i}{\partial t} + \frac{\partial \rho_m u_i u_j}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[(\mu_t + \mu_m) \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + \sigma_s \kappa \delta(\phi) \mathbf{n} \quad (2)$$

where ρ , u , and p represent the fluid density, velocity, and pressure, respectively. The subscripts m and v mean the mixture and vapor, respectively. μ_m and μ_t are the molecular and turbulent viscosity.

The last term of Equation 2 is the surface tension force acting on the interface, with σ_s representing the surface tension coefficient, κ representing the surface curvature, and $\mathbf{n}$ representing the interface normal vector pointing from the primary fluid to the second fluid (calculated using the volume fraction gradient). In order to solve the surface tension term, a level set function [11] i.e. δ is adopted.

Cavitating flow is a multi-scale turbulent flow. So, in order to simulate different turbulent scales accurately, the filter-based model [19] (FBM) was adopted. The FBM uses different methods to define the turbulent viscosity μ_t in different turbulent scales, so the FBM is more superior than the k - ε model, and the results that conclude by using FBM are more precise. The turbulent viscosity μ_t in FBM is defined as follows:

$$\mu_t = \frac{C_\mu \rho_m k^2}{\varepsilon} F_{\text{FBM}} \quad (3)$$

$$F_{\text{FBM}} = \min \left(1, C_3 \frac{\lambda \cdot \varepsilon}{k^{3/2}} \right) \quad (4)$$

In the equation, λ is the filter scales, F_{FBM} is the filter function, and $C_\mu = 0.09$, $C_3 = 1.0$.

2.2. Cavitation Models

2.2.1. Zwart Model

The Zwart model is given by the simplified Rayleigh-Plesset equation, which is used for bubble dynamics. In the Zwart model, condensation and evaporation source terms are shown as follows:

$$\begin{aligned} \dot{m}^- &= -F_e \frac{3r_{\text{nuc}}(1-\alpha_v)\rho_v}{R_B} \sqrt{\frac{2}{3} \frac{P_v - P}{\rho_l}} & \text{if } P < P_v \\ \dot{m}^+ &= F_c \frac{3\alpha_v\rho_v}{R_B} \sqrt{\frac{2}{3} \frac{P - P_v}{\rho_l}} & \text{if } P > P_v \end{aligned} \quad (5)$$

where P_v represents the vapor pressure, r_{nuc} represents the nucleation site volume fraction, R_B represents the radius of a nucleation site, F_e represents the empirical calibration coefficients for the evaporation, and F_c represents the empirical calibration coefficients for the condensation. In the commercial software Ansys CFX 13.0, we set the above-mentioned coefficients as follows: $r_{\text{nuc}} = 5.0 \times 10^{-4}$, $R_B = 1.0 \times 10^{-6} \text{ m}$, $F_e = 300$, and $F_c = 0.03$.

2.2.2. Full Cavitation Model

This cavitation model was known as the full cavitation model originally when it was proposed by Singhal et al, but it will be referred to from now on as FCM. In this model, the growth and collapse of a bubble cluster are based on the reduced form of the Rayleigh-Plesset equation for bubble dynamics. The expressions of FCM are as follows:

$$\begin{aligned} \dot{m}^- &= -C_e \frac{\sqrt{k}}{T} \rho_l \rho_v \sqrt{\frac{2}{3} \frac{P_v - P}{\rho_l}} (1 - f_v) & \text{if } P < P_v \\ \dot{m}^+ &= C_c \frac{\sqrt{k}}{T} \rho_l \rho_v \sqrt{\frac{2}{3} \frac{P - P_v}{\rho_l}} f_v & \text{if } P > P_v \end{aligned} \quad (6)$$

where f_v represents the vapor mass fraction, k is the turbulent kinetic energy, T is the surface tension, and the two empirical coefficients are set as follows: $C_e = 0.40$, $C_c = 2.3 \times 10^{-4}$.

For convenience, in this study, we did not use the above expressions for numerical simulation, but in CFX, we used the expressions derived by Huuva in which the vapor mass fraction f_v is replaced by the vapor volume fraction α_v [21].

2.2.3. Kunz Model

Unlike the above-mentioned models, in Kunz cavitation model, the condensation $\dot{m}^-$ and evaporation $\dot{m}^+$ of the liquid are based on two different strategies. The evaporation source term $\dot{m}^+$ is computed as being proportional to the amount by which the pressure is below the vapor pressure and the condensation source term $\dot{m}^-$; otherwise, it is based on a third-order polynomial function of water volume fraction γ . The formulations of the Kunz model are as follows:

$$\begin{aligned}\dot{m}^- &= \frac{C_{\text{dest}} \rho_v \gamma \min[0, P - P_v]}{\frac{1}{2} \rho_l U_\infty^2 t_\infty} \\ \dot{m}^+ &= \frac{C_{\text{prod}} \rho_v \gamma^2 (1 - \gamma)}{t_\infty}\end{aligned}\quad (7)$$

where U_∞ is the free-stream velocity, and $t_\infty = L/U_\infty$ is the mean flow time scale, where L is the characteristic length scale of hydrofoil. Two empirical coefficients, C_{dest} and C_{prod} , are set as follows: $C_{\text{dest}} = 4100$, $C_{\text{prod}} = 455$.

The value of all the empirical coefficients mentioned in above three cavitation models are tuned to an optimal value by means of an optimization strategy mentioned by Mitja Morgut [20].

3. Grids and CFD Set-Up

In the present study, rectangular domains around the hydrofoil NACA0015 were used to simulate the cavitating flow. Figure 1 shows the boundary conditions and shape of the computational domain that was used for the NACA0015. For NACA0015 hydrofoil, the inlet boundary was placed three chord lengths ahead of the leading edge, while the outlet boundary was placed 5.5 chord lengths behind the trailing edge, and the span length was $0.3c$. The top and bottom boundaries were placed 2.5 chord lengths from the hydrofoil interface. The chord length c of the NACA0015 hydrofoil is 0.07 m.

A free-slip wall condition was used on the solid surface of the top and bottom, while the no-slip condition was used on the surface of hydrofoil. For the outer boundary, static pressure was set. When the static pressure $P_{\text{ref}} = 31,325$ Pa, it belongs to the sheet cavitation condition. When the static pressure $P_{\text{ref}} = 20,325$ Pa, it belongs to the cloud cavitation condition. On the inlet boundary, the values of the free-stream velocity components and turbulence quantities were fixed. The water volume fraction was set equal to one and the vapor volume fraction was set to zero on the inlet boundary. The free-stream velocity U_∞ was set equal to 7.2 m/s. Moreover, in the computation, the water and vapor density were kept constant, and they were set as follows: $\rho_l = 998$ kg/m³, $\rho_v = 0.023$ kg/m³. Different cavitating flow regimes were related to the cavitation number σ , which was defined as follows:

$$\sigma = \frac{P_{\text{ref}} - P_v}{\frac{1}{2} \rho_l U_\infty^2} \quad (8)$$

where P_v represents the saturation pressure. In this research, σ is equal to 1.075 when the value of saturation pressure $P_v = 31,325$ Pa, i.e. a sheet cavitation condition, and σ is equal to 0.065 when the value of saturation pressure $P_v = 20,325$ Pa, i.e. a cloud cavitation condition. In the numerical simulation, the pressure coefficient C_p was used and defined as follows:

$$-C_p = \frac{P_{\text{ref}} - P}{\frac{1}{2} \rho_l U_\infty^2} \quad (9)$$

where P is the local absolute pressure.

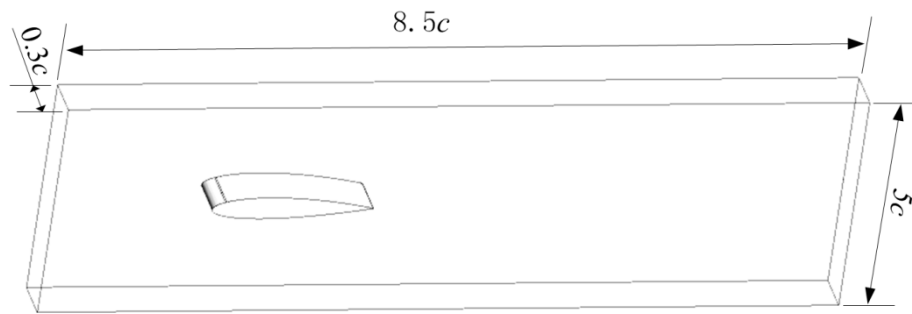


Figure 1. Computational domain and boundary conditions for NACA0015.

Mesh was hexa-structured and generated using the commercial software ANSYS-ICEM meshing tool. Local mesh refinement was performed in the surrounding area of the hydrofoil wall, which made the computational results more accurate. Figure 2 shows the mesh around the NACA0015 hydrofoil and the close view of the mesh along the leading trailing edge.

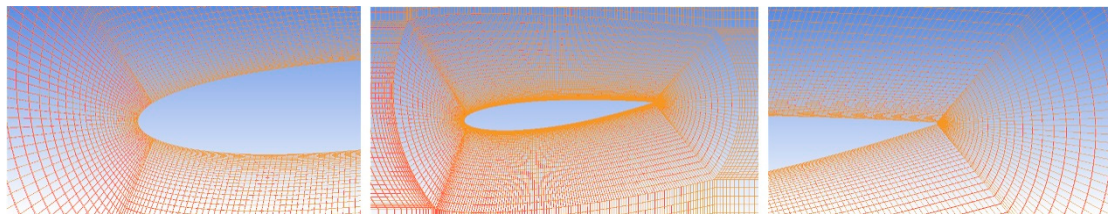


Figure 2. Mesh around NACA0015 at an angle of attack $\alpha = 6^\circ$ (middle); close view of the leading edge region (left); close view of the trailing edge region (right).

The simulations were conducted by using ANSYS CFX 13.0. The time-dependent governing equations were discretized both in space and time. The velocity was assigned at the domain inlet, and the static pressure was assigned at the outlet according to the cavitation number.

4. Results and Discussion

4.1. Flow Characteristics of Sheet Cavitation

4.1.1. Steady Characteristics

Figure 3 shows the cavity structures observed by the experimental and numerical results, which were calculated by using the three different cavitation models. Note that the experimental data were obtained from YU and LUO [22]. Comparing the experimental data and numerical results from Figure 3, it is indicated that the Zwart model and Kunz model can predict similar distributions of vapor bubbles around the hydrofoil with the experimental data using the modified simulation method. The structure of the vapor appears to be attached to the hydrofoil NACA0015. The calculated cavity length is about $0.36c$ for the Kunz model and $0.37c$ for the Zwart model, while in the experiment, it was about $0.39c$. The major difference between the Zwart model and the Kunz model is in the closure region of the cavities. The trailing edge of the cavity calculated by the Zwart model is narrow and long. The cavity structure predicted by the full cavitation model (FCM) is a very thin layer vapor attached to the leading edge of the hydrofoil, which is quite different from the cavity structure observed in the experiment. It is indicated that the full cavitation model (FCM) cannot predict the formation of sheet cavitation accurately.

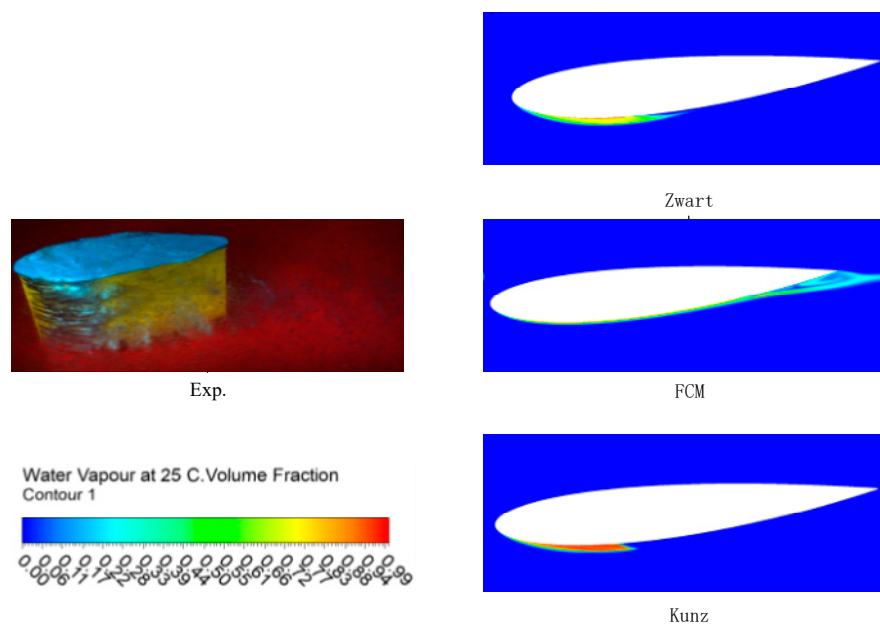


Figure 3. Cavity structures of sheet cavitation around NACA0015 hydrofoil ($\sigma = 1.075$, $\alpha = 60^\circ$).

Figure 4 shows the distribution of the evaporation and condensation rate by using different cavitation models. It can be found from Figure 4 that the leading edge of the hydrofoil is the evaporation area, and the second half of the cavity is the condensation area. The value of the evaporation source term reflects the degree of the transition from liquid to vapor in the flow field region. The vapor phase is generated at the distribution area of the evaporation source term. Likewise, the value of the condensation source term reflects the degree of the transition from vapor to liquid in the flow field region. It is indicated that both the Zwart model and the Kunz model simulate the evaporation and condensation areas clearly. Meanwhile, the full cavitation model only simulates the small range of the evaporation area attached to the leading edge of the hydrofoil, and does not calculate the existence of the condensation area. Combined with Figure 3, it can be found that the evaporation region is about half of the entire cavity in the Zwart model. In the Kunz model, the evaporation region accounts for about two-thirds of the length of entire cavity. The difference between the two distributions is due to the different definitions of the evaporation and condensation source terms in the Zwart and Kunz models. Both the Zwart model and the Kunz model use the same method to define the evaporation source term. The evaporation source term $\dot{m}^-$ of the Zwart model is proportional to half of the power difference between the bubble pressure P_v and the flow field pressure p , while the evaporation source term in the Kunz model is proportional to the difference between the bubble pressure P_v and the flow field pressure p . Therefore, the results show that Zwart model and Kunz model predict similar evaporation regions. The difference between the two models is mainly in the condensation area. The Zwart model performs the same method to define the evaporation and condensation source terms, and so the calculated evaporation area and condensation area account for about half of the cavity. The Kunz model uses different methods to define the evaporation and condensation source terms, and the condensation source term $\dot{m}^+$ is a cubic polynomial based on the liquid volume fraction. Therefore, the Zwart and Kunz models have different calculation results for the condensation area of the cavity tail, which is the main reason for the difference in the cavity tail structure predicted by the two models.

It can be seen from Figure 3 that the cavitation region predicted by the Kunz model is a high-purity vapor phase area, and the vapor–liquid interphase is clearer than the Zwart model; that is, the vapor–liquid interphase is thin. The cavitation area predicted by the Zwart model is a low-density vapor–liquid mixed region, and the prediction of the vapor–liquid interface is relatively fuzzy; that is, the vapor–liquid interface is thick. Figure 5 shows the suction side wall pressure

coefficient ($-C_p$) distributions of the experimental data and the results from the three cavitation models. It indicates that there is an adverse pressure gradient at the trailing edge of the hydrofoil near the wall. As shown in the $-C_p$ curve of the Kunz model in Figure 5, the $-C_p$ curve drops sharply in the interval from the leading edge of the hydrofoil ($0.4\text{--}0.45)c$, which is closer to the experimental data. The difference between the Zwart model and Kunz model is that the pressure coefficient curve of the former changes gently in the cavity tail near the wall, which means a small adverse pressure gradient. It can be found from Figure 5 that in the full cavitation model (FCM), the pressure coefficient curve begins to rise gradually at about $0.1c$ from the leading edge of the hydrofoil, which is different from the other models. The full cavitation model can only predict cavitation formation at the region $(0\text{--}0.1)c$, but it cannot predict the whole sheet cavitation structure successfully, which is inconsistent with the experimental phenomena. Thus, the Kunz model can get a more accurate adverse pressure gradient than the other models at the sheet cavitation condition around the NACA0015 hydrofoil.

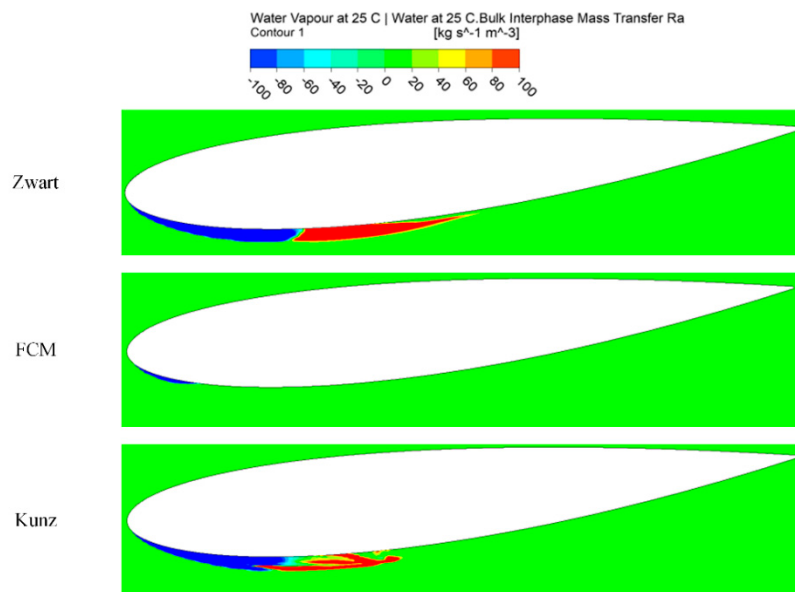


Figure 4. Interphase mass transfer rate distribution of different cavitation models ($\sigma = 1.075$, $\alpha = 60^\circ$).

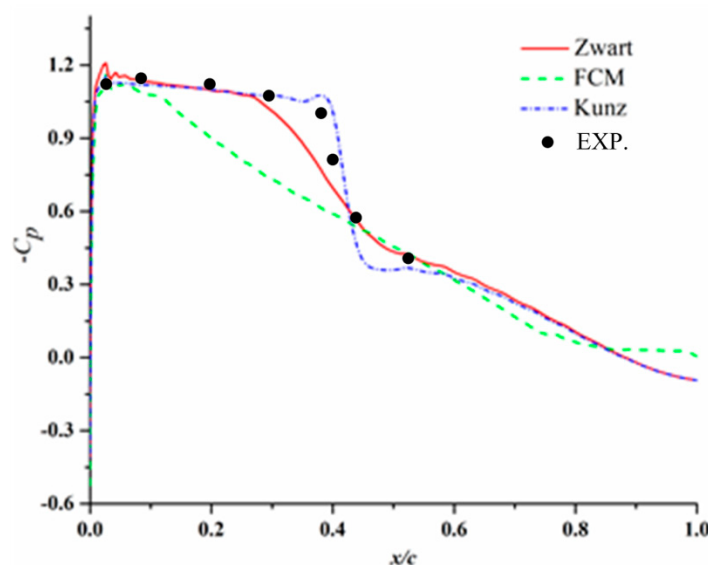


Figure 5. Suction side wall pressure distributions calculated by using three different cavitation models ($\sigma = 1.075$, $\alpha = 60^\circ$).

4.1.2. Unsteady Characteristics

The unsteady periodic variation of sheet cavitation is the main reason for the pressure fluctuations on the hydrofoil surface. The shearing action generated by the re-entrant jet is the root cause of the change in the structure of the bubble tail. This section analyzes the unsteady characteristics of sheet cavitation based on the numerical results, and evaluates the prediction performance of three cavitation models at the same time.

Figure 6 shows the time evolution of sheet cavitation for numerical results and the experimental data of a whole cycle. At the time of $0 T$, a groove in the cavity is formed by the action of the re-entrant jet at the cavity tail. The re-entrant jet continuously moves forward over time, and until $0.7 T$, the wall separation of the re-entrant jet due to the adverse pressure gradient shears the cavity tail, which leads to the shedding of microscale bubbles at the time of $0.9 T$. From 0.9 to $1.0 T$, the attached bubble becomes large, and the separation point between the cavity tail and the wall moves back to complete a periodic variation. It can be found from Figure 6 that the Zwart model cannot predict the unsteady characteristics of the cavity attached to the suction side; it does not simulate the shedding of the microscale bubbles in the cavity tail. The full cavitation model (FCM) fails to predict the cavity structure of sheet cavitation. At the end of the cavity, the full cavitation model (FCM) predicts the shedding of bubbles whose scale is large. It indicates that the FCM overpredicts the shedding of bubbles, which is inconsistent with experimental phenomena. The full cavitation model (FCM) takes the turbulent energy k into account for the definition of the condensation source term. So, the correlation between the phase transition and turbulent flow field is enhanced, and the unsteady characteristics of the flow field are enhanced, which leads to the shedding of large bubbles in the cavity tail. Obviously, it can be found that the periodic fluctuations of the cavity structure with time evolving, as predicted by the Kunz model, is consistent with experimental phenomena. Through the above analysis, the results predicted by the Kunz model are the closest to the experiment.

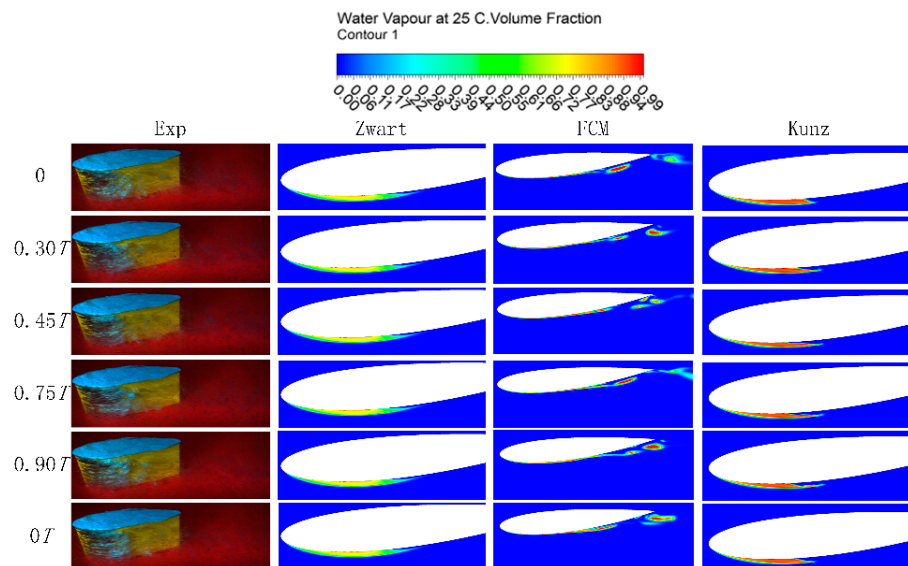


Figure 6. Time evolution of sheet cavitation for numerical results and experimental pictures ($\sigma = 1.075$, $\alpha = 60^\circ$).

Figure 7 shows the distribution of the velocity vector field predicted by the three cavitation models. We find that the Zwart model does not predict the formation of the re-entrant jet and the vortex. The attached cavity becomes narrow and long under the action of inertia force, so the Zwart model can simulate the effect of inertia force on the elongation of the bubble. Meanwhile, the full cavitation model (FCM) predicted the formation of the re-entrant jet and the vortex shedding of bubbles, but the structure of the attached cavity is inconsistent with the experimental observation.

So, the FCM results are not accurate. The Kunz model predicts the formation and movement of the re-entrant jet accurately. It predicts the shearing action on the attached cavity tail and the shedding of microscale bubbles, which is consistent with the experimental observations.

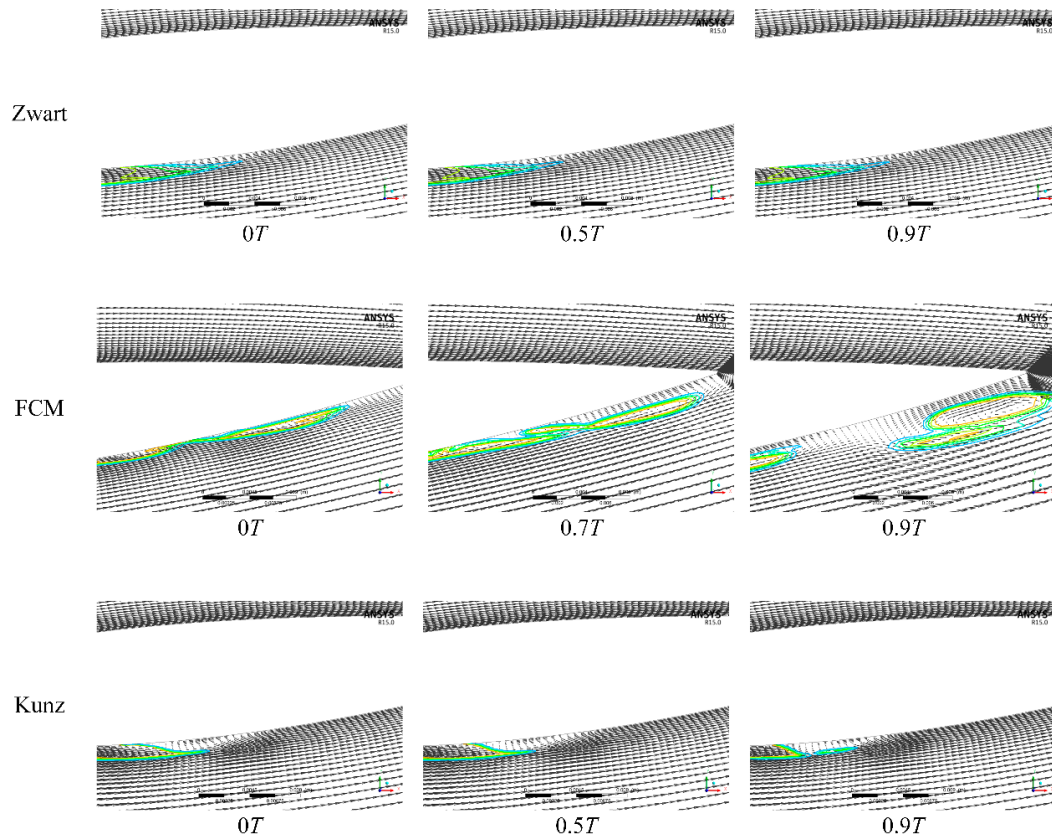


Figure 7. Velocity vector field predicted by three cavitation models ($\sigma = 1.075$, $\alpha = 60^\circ$).

4.2. Flow Characteristics of Unsteady Cloud Cavitation

In the above section, the steady and unsteady characteristics of sheet cavitation are mainly analyzed. In this section, the investigations are focused on the unsteady characteristics of cloud cavitation. The cloud cavitation is a complex physical phenomenon with obvious strong unsteady characteristics.

Figure 8 shows the evolution of the cavity structure over time, with the experimental data and numerical results. Three cavitation models clearly captured the periodic process of the initial, development, shedding, and collapse of the bubbles, which is consistent with the experimental phenomena. The periodic evolution of the cavity structure can be divided into two processes. The first is the process of bubble development from t_0 to $t_0 + 12.5$ ms. During this process, bubbles attached to the leading edge of the hydrofoil and grew linear to the maximum length. The second is the process of the formation and development of the re-entrant jet from $t_0 + 12.5$ ms to $t_0 + 42.5$ ms. When the attached bubbles grow to their maximum length, the re-entrant jet begins to appear at the cavity tail, near the wall of the hydrofoil. As the re-entrant jet moves along the suction side toward the leading edge of the hydrofoil, the attached cavity is sheared and then separated from the suction side. When the re-entrant jet reaches the leading edge of the hydrofoil, the attached cavity breaks into two parts, which are named the separated cavity body and the attached cavity body. The separated cavity body moves along the mainstream in the form of a vortex. During this process, the attached cavity body stops growing, and disappears suddenly. Then, the separated cavity body moves downstream and collapses in the high-pressure area.

By comparing the calculated results of the three cavitation models, it can be found that the different cavitation models predict different cycle scales. The cycle calculated by the Kunz model is

short; it is equal to 47.5 ms ($t_0 = 47.5$ ms). The cycle calculated by the other two models is about 55 ms ($t_0 = 55$ ms), which is agree well with the experimental data. The Kunz model predicts the fracture and shedding of bubbles prematurely.

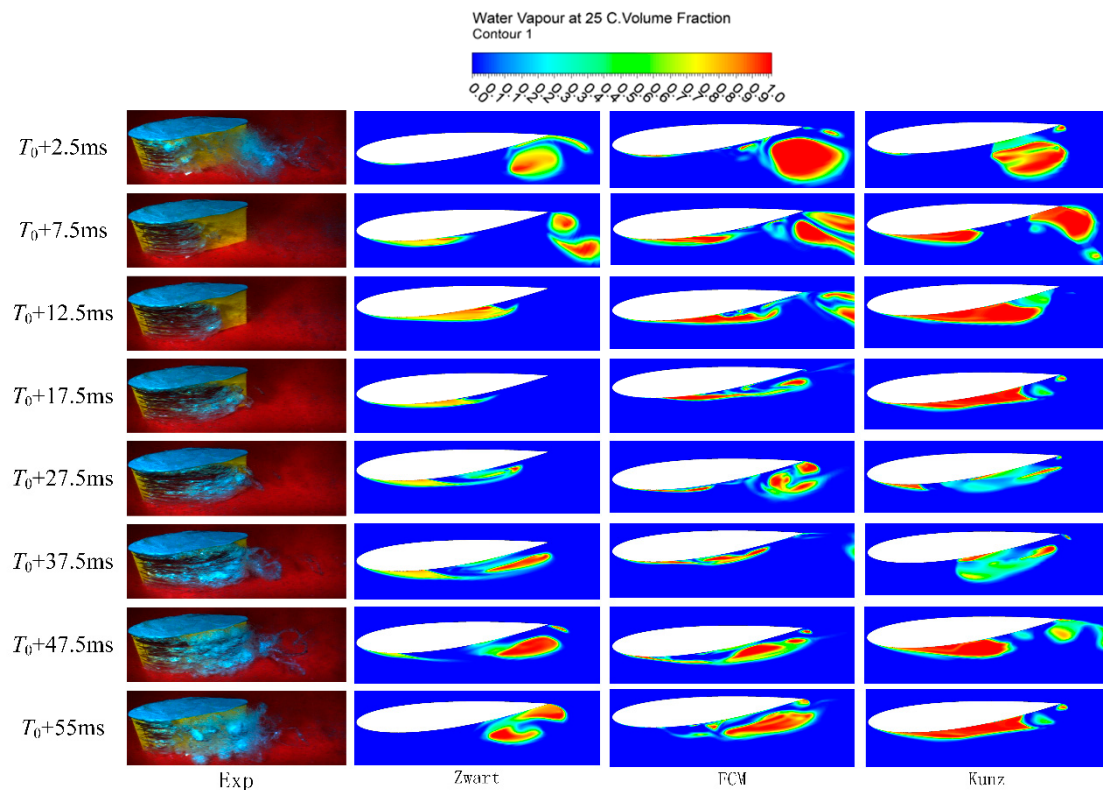


Figure 8. Time evolution of cloud cavitation for the numerical and experimental results ($\sigma = 0.65$, $\alpha = 60^\circ$).

The Zwart model successfully predicts the unsteady details of the shearing action of the re-entrant jet on the attached cavity. However, it is not obvious enough to simulate the vortex characteristics for the cavity tail. For the full cavitation model (FCM), the cavity thickness that it predicted was thinner than the other two models, and the cavitation region had a higher vapor content. It simulated the interface of vapor and liquid more clearly, and accurately captured the unsteady details regarding the vortex motion and re-entrant jet, which is consistent with the experimental phenomena. The Kunz model calculated the maximum length and thickness of the bubbles out of the three cavitation models. However, the Kunz model could not predict the shearing action of the re-entrant jet on the attached cavity; it could only predict the shedding vortex of the cavity tail and the retraction of the attached cavity. Also, the Kunz model predicted the shedding of bubbles prematurely.

Different cavitation models consider different mechanisms and methods, so different results were obtained. The Zwart model defined the evaporation and condensation source terms from the perspective of the difference between the vapor pressure P_v and the flow field pressure p . It could not reflect the degree of turbulence in the flow field only from the perspective of pressure difference, so the Zwart model does not predict the vortex motion of the cavity tail. The Kunz model uses the cubic polynomial of the liquid volume fraction to define the condensation source term, which leads to the excessive prediction of the condensation potential. Thus, cavity bubbles separated in advance, and the cycle was short.

In order to have a deeper understanding of the re-entrant jet as it progresses and the shedding mechanism with the three cavitation models, Figure 9 gives the formation and evolution of the re-entrant jet calculated by using the three different cavitation models for a whole cycle. Meanwhile, Figure 10 gives the re-entrant jet and vortex shedding results that were observed in the experiment.

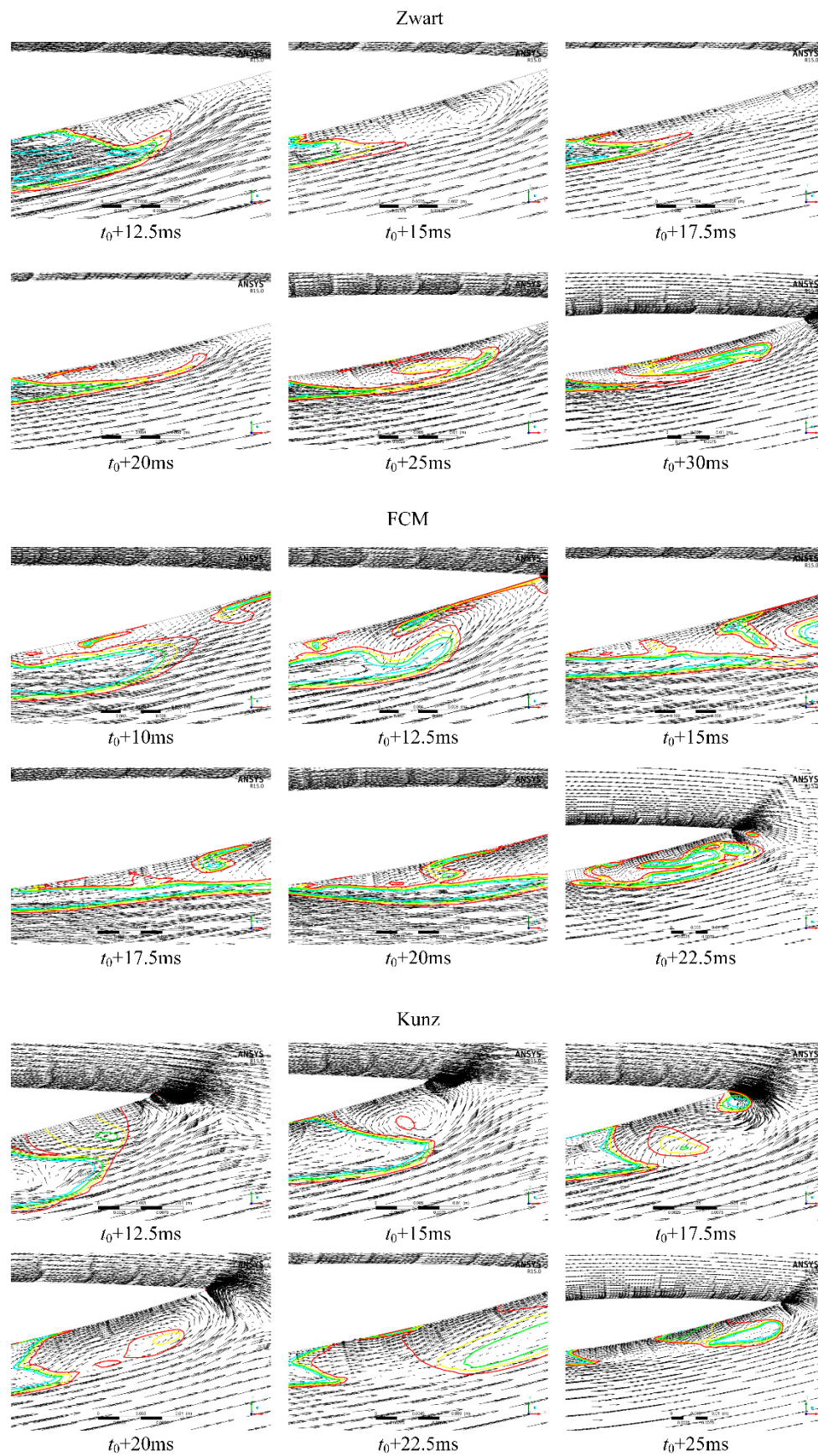


Figure 9. The formation and evolution of the re-entrant jet calculated by three different cavitation models ($\sigma = 0.65$, $\alpha = 60^\circ$).

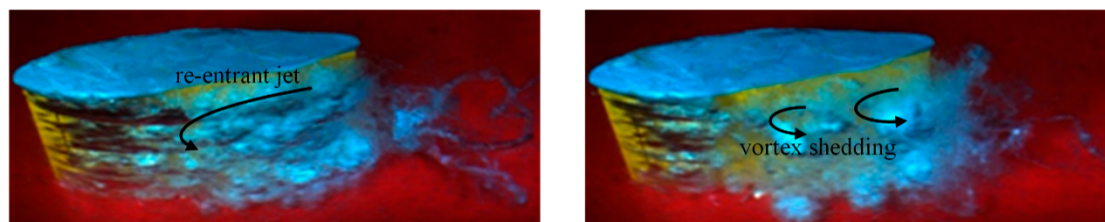


Figure 10. Re-entrant jet and vortex shedding observed in the experiment ($\sigma = 0.65$, $\alpha = 60^\circ$).

From the results of the Zwart model, it can be seen that the mixture of water and vapor at the cavity tail moves toward the leading edge near the wall under the action of local high pressure. The vortex is generated in the re-entrant jet area and expands with the forward movement of the re-entrant jet, and then moves downstream with the shedding bubbles. It is not obvious enough for the Zwart model to simulate the vortex motion inside the cavitating area. The full cavitation model (FCM) not only predicts the formation of the re-entrant jet in the cavity and the separation of attached bubbles caused by re-entrant jet, it also predicts the vortex flow.

The Kunz model also predicts the vortex motion inside the cavitation area. However, the vortex cannot move to the lower and rear part of the hydrofoil. Moreover, the Kunz model does not predict the shearing action of the re-entrant jet on the attached cavity. The attached cavity is compressed and then shears under the action of the tail vortex; then, it is further compressed by the re-entrant jet, which is inconsistent with experimental phenomena. Considering the three cavitation models, the full cavitation model (FCM) successfully predicts the formation of the re-entrant jet and its shearing on the attached cavity. In addition, the FCM predicts the vortex shedding of bubbles, which is consistent with the experimental observations.

5. Conclusions

Cavitating flows that occur in a variety of practical cases can be modeled with a wide range of methods. Based on the FBM turbulence model and level set method, the applicability of the Kunz, FCM, and Zwart cavitation models in sheet and cloud cavitation around a NACA0015 Hydrofoil were investigated. Several conclusions can be drawn as follows.

At the condition of sheet cavitation, the Kunz model has an accurate prediction for the steady and unsteady cavitation phenomena. It can accurately capture both the adverse pressure gradient and the action of the re-entrant jet. The Zwart model can predict the structure of the attached cavity accurately, but it cannot predict the adverse pressure gradient and the formation of the re-entrant jet. Meanwhile, the full cavitation model (FCM) overpredicts the unsteady characteristics of sheet cavitation, and cannot capture the structure of the cavity bubble.

At the condition of cloud cavitation, the full cavitation model (FCM) has a more accurate prediction for the flow field structure and reflects the unsteady phenomena that are observed in the experimental tests. Meanwhile, with the Zwart model, the vortex is generated in the tailing edge of the cavity, and expands with the forward movement of the re-entrant jet. However, it is not obvious enough for the Zwart model to simulate the vortex motion near the cavitating area. The Kunz model does not predict the shearing action of the re-entrant jet on the attached cavity. The variation of the cavity structure is inconsistent with the experimental observations. Moreover, for the cavitation around a hydrofoil, the selection of cavitation models seems to have a good general validity.

Author Contributions: Conceptualization, A.Y.; methodology, A.Y. and D.Q.Z.; software, Q.H.T.; validation, A.Y., Q.H.T. and D.Q.Z.; formal analysis, A.Y.; investigation, A.Y.; writing—original draft preparation, A.Y. and Q.H.T.; writing—review and editing, A.Y.; funding acquisition, A.Y.

Funding: This work was financially supported by the National Natural Science Foundation of China (Project Nos. 51806058), and the Fundamental Research Funds for the Central Universities (Project No. 2019B15014).

Conflicts of Interest: The authors declare no conflict of interest.

References

1. Zhang, B.; Wang, G.Y.; Huang, B.; Yu, Z.Y. Numerical and experimental studies on unsteady shedding mechanisms of cloud cavitation. *Chin. J. Theor. Appl. Mech.* **2009**, *41*, 651–659.
2. Arndt, R.E.A. Some Remarks on Hydrofoil Cavitation. *J. Hydrodyn.* **2012**, *24*, 305–314. [[CrossRef](#)]
3. Pan, S.S.; Peng, X.X. *Physical Mechanism of Cavitation*, 1st ed.; National Defense Industry Press: Beijing, China, 2013.
4. Gao, Y.; Huang, B.; Wu, Q.; Wang, G.Y. Experimental investigation of the vibration characteristics of hydrofoil in cavitating flow. *Chin. J. Theor. Appl. Mech.* **2015**, *47*, 1009–1016.
5. Prosperetti, A. Vapor Bubbles. *Annu. Rev. Fluid Mech.* **2017**, *49*, 221–248. [[CrossRef](#)]
6. Barre, S.; Rolland, J.; Boitel, G.; Goncalves, E.; Fortes Patella, R. Experiments and modeling of cavitating flows in venture: Attached sheet cavitation. *Eur. J. Mech. B Fluids* **2009**, *28*, 444–464. [[CrossRef](#)]
7. Coutier-Delgosha, O.; Reboud, J.L.; Delannoy, Y. Numerical simulation of the unsteady behavior of cavitating flows. *Int. J. Numer. Methods Fluids* **2003**, *42*, 527–548.
8. Zwart, P.; Gerber, A.G.; Belamri, T. A two-phase model for predicting cavitation dynamics. In Proceedings of the ICMF International Conference on Multiphase Flow, Yokohama, Japan, 30 May–4 June 2004.
9. Singhal, A.K.; Athavale, M.M.; Li, H.; Jiang, Y. Mathematical basis and validation of the full cavitation model. *J. Fluid Eng.-Trans. ASME* **2002**, *124*, 617–624. [[CrossRef](#)]
10. Kunz, R.F.; Boger, D.A.; Stinebring, D.R.; Chyczewski, T.S.; Lindau, J.W.; Gibeling, H.J.; Venkateswaran, S.; Govindan, T.R. A preconditioned Navier-Stokes method for two-phase flows with application to cavitation prediction. *Comput. Fluids* **2000**, *29*, 849–875. [[CrossRef](#)]
11. Huang, B.; Wang, G.; Zhang, M. Level set method for simulation of cavitating flows. *J. Ship Mech.* **2011**, *3*, 207–216.
12. Ji, B.; Luo, X.W.; Arndt, R.E.A. Numerical simulation of three dimensional cavitation shedding dynamics with special emphasis on cavitation–vortex interaction. *Ocean Eng.* **2011**, *87*, 64–77. [[CrossRef](#)]
13. Wang, G.; Ostojca-Starzewski, M. Large eddy simulation of a sheet cloud cavitation on a NACA0015 hydrofoil. *Appl. Math. Model.* **2007**, *31*, 417–447. [[CrossRef](#)]
14. Luo, X.W.; Ji, B.; Peng, X.X. Numerical simulation of cavity shedding from a three-dimensional twisted hydrofoil and induced pressure fluctuation by large-eddy simulation. *J. Fluids Engineering-Trans. ASME* **2012**, *134*, 041202. [[CrossRef](#)]
15. Ji, B.; Luo, X.W.; Peng, X.X. Three-dimensional large eddy simulation and vorticity analysis of unsteady cavitating flow around a twisted hydrofoil. *J. Hydrodyn.* **2013**, *25*, 510–519. [[CrossRef](#)]
16. Roohi, E.; Zahiri, A.P.; Passandideh-Fard, M. Numerical simulation of cavitation around a two-dimensional hydrofoil using VOF method and LES turbulence model. *Appl. Math. Model.* **2013**, *37*, 6469–6488. [[CrossRef](#)]
17. Huang, B.; Zhao, Y.; Wang, G.Y. Large eddy simulation of turbulent vortex-cavitation interactions in transient sheet/cloud cavitating flows. *Comput. Fluids* **2014**, *92*, 113–124. [[CrossRef](#)]
18. Ji, B.; Luo, X.W.; Arndt, R.E.A. Large eddy simulation and theoretical investigations of the transient cavitating vortical flow structure around a NACA66 hydrofoil. *Int. J. Multiph. Flow* **2015**, *68*, 121–134. [[CrossRef](#)]
19. Yu, A.; Ji, B.; Huang, R.F. Cavitation shedding dynamics around a hydrofoil simulated using a filter-based density corrected model. *Sci. China Technol. Sci.* **2015**, *58*, 864–869. [[CrossRef](#)]
20. Mitja, M.; Enrico, N.; Lgnacijo, B. Comparison of mass transfer models for the numerical prediction of sheet cavitation around a hydrofoil. *Int. J. Multiphas Flow* **2011**, *37*, 620–626.
21. Huuva, T. Large Eddy Simulation of Cavitating and Non-Cavitating Flow. Ph.D. Thesis, Chalmers University of Technology, Gothenburg, Sweden, 2008.
22. Yu, A.; Luo, X.W.; Yang, D.D. Experimental and numerical study of ventilation cavitation around a NACA0015 hydrofoil with special emphasis on bubble evolution and air-vapor interactions. *Eng. Comput.* **2018**, *35*, 1528–1542. [[CrossRef](#)]

