Numerical analysis of the interactions of sheet cavitation and cloud cavitation around a hydrofoil

D Z Pan, D S Zhang, H Y Wang, W D Shi and L Shi
Research Center of Fluid Machinery Engineering and Technology, Jiangsu University, Zhenjiang 212013, China
Email: zds@ujs.edu.cn

Abstract: Partial cavitation and cloud cavitation on NACA66(MOD) was studied based on PANS turbulence model combined with the Zwart cavitation model. The results agree well with the data from the experimental results. The existence of a kind of unsteady characteristics of cloud cavitation is analysed in details. The reasons for the inhibition mechanism of the sheet cavitation are discussed. Pressure shock wave occurs near the tailing edge of the hydrofoil when the collapse of cloud is so large leading to great influence on the cavity closed to the leading edge. The evolution of the pressure peak and its propagation toward the leading edge were investigated through the study of the time domain signal of the monitor points along the streamline of flow. Finally, the propagation of the pressure wave on the suction side was further investigated through analysing the spatial-time history of wall pressure.

1. Introduction
In the past years, many research about cavitation were carried out using experiment and simulation [1-4], which show that the turbulence model is crucial determining the accuracy of simulation prediction. The traditional RANS model over predicts the turbulence viscosity of the cavity rear part. In order to solve the problem, Coutier D et al [5] proposed an eddy viscosity correction for RNG $k$-$\varepsilon$ model, and obtained good simulation predicting results. With the development of the computer technology, large eddy simulation (LES) began to applied in the unsteady cavitation simulation, but it is difficult to obtain a grid-independent solution [6]. Girimaji et al [7] proposed a partially-averaged Navier-Stokes(PANS) model, to achieve a seamlessly transition from RANS to direct numerical simulation(DNS) of the Navier-Stokes equations. Ji et al. [8-9] used the PANS model in the cavitation simulation for the twist hydrofoil and obtained an agreement results.

In the past years, the question in all cavitation researches concerns the origin of the flow instability. Leroux et al. [10] found the relation of the length of the cavity and the foil chord exists in the experiment. Later, Leroux [1] found the inhibition phenomenon of residual cavity. Jung H S et al. [11] employed the Shin cavitation model [12] to further investigate the problem.

In the present paper, the cavitation field around the two-dimensional NACA66(MOD) hydrofoil were investigated, trying to further explore the interactions of sheet cavitation and cloud cavitation.

1 Corresponding author.
2. Governing equations and numerical method

2.1. Turbulence model
The PANS model can transit smoothly from RANS to DNS through changing the two control parameters \( f_k \) and \( f_\varepsilon \), which are defined as follow:

\[
f_k = k_u / k, \quad f_\varepsilon = \varepsilon_u / \varepsilon
\]

(1)

Where \( k \) and \( \varepsilon \) are total turbulence kinetic energy and turbulence eddy dissipation respectively. The subscript \( u \) refers to the unresolved quantities. Based on the standard \( k-\varepsilon \) model, Girimaji [7] shows the governing equations of the PANS model as follow:

\[
\frac{\partial (\rho k_u)}{\partial t} + \frac{\partial (\rho u_j k_u)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_{ku}} \right) \frac{\partial k_u}{\partial x_j} \right] + P_{kw} - \rho \varepsilon_u
\]

(2)

\[
\frac{\partial (\rho \varepsilon_u)}{\partial t} + \frac{\partial (\rho u_j \varepsilon_u)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_{eu}} \right) \frac{\partial \varepsilon_u}{\partial x_j} \right] + C_{\varepsilon 1} \frac{\rho \varepsilon_u}{k_u} - C_{\varepsilon 2} \frac{\varepsilon_u^2}{k_u}
\]

(3)

Compared to the standard \( k-\varepsilon \) model, PANS model modified some coefficients as follow:

\[
\sigma_{ku} = \sigma_k f_k^2 / f_\varepsilon, \quad \sigma_{eu} = \sigma_\varepsilon f_k^2 / f_\varepsilon
\]

(4)

\[
C_{\varepsilon 2} = C_{\varepsilon 1} + f_k / f_\varepsilon (C_{\varepsilon 2} - C_{\varepsilon 1})
\]

(5)

Where \( \sigma_k = 1.0, \quad \sigma_\varepsilon = 1.3, \quad C_{\varepsilon 1} = 1.44 \) and \( C_{\varepsilon 2} = 1.92 \).

2.2. Cavitation model
Cavitation model is a mathematical model using describing the mass transfer between the liquid and vapor, which is defined as

\[
\frac{\partial \rho \alpha_v}{\partial t} + \frac{\partial (\rho \alpha_v u_i)}{\partial x_j} = m
\]

(6)

The source terms for the mass transfer rate are given by

\[
m = \begin{cases} 
-F_v^3 r_{\text{num}} (1-\alpha_v) \rho_v \frac{2 (p_v - p)}{3 \rho_i} & (p < p_v) \\
F_v^3 \frac{3 \alpha_v \rho_v}{R_B} \frac{2 (p - p_v)}{3 \rho_i} & (p > p_v)
\end{cases}
\]

(7)

Where \( F_v = 50, \quad F_i = 0.01, \quad r_{\text{num}} = 5 \times 10^4, \quad R_B = 10^{-6} \) based on the work by Zwart et al [13].

2.3. Boundary conditions and grid
The simulation used the NACA66(MOD) hydrofoil with a chord length \( c = 150 \) mm. Figure 1 shows the classical boundary conditions for hydrofoil: imposed inlet velocity and fixed outlet pressure. A no-slip wall is used for upper and lower walls and hydrofoil surface. The total grid elements are 89172 and the \( y+ \) values are within 0–10.

According to the experiment [12], the angle of attack and the cavitation number are 6°, 0.99 respectively. Velocity \( U_{\text{in}} = 5.33 \) m/s is fixed at the inlet, corresponding to Reynolds number \( 8 \times 10^5 \). The outlet pressure is determined by the cavitation number. In the unsteady simulation, the time step \( \Delta t \) was set to 10e-3s.
3. Results and discussion

3.1. One-cycle cavitation oscillation

According to the arrangement of the Leroux’s experiment, the cavitation field around the NACA66(MOD) hydrofoil has been investigated. Employed of the homogeneous flow model, the volume fraction contour in the picture represents the distribution of the bubbles in the flow field. The two extremes of the rainbow chart, the red and blue correspond to pure vapor and water. The comparison between the experiment and simulation of the cavitation cavity, named dynamics 1 by Leroux, shows in the follow pictures.

![Figure 3. The comparison of the time evolution of void fraction.](image)

Based on the above results, cloud cavitation of dynamics 1 within each cycle can be divided into the following steps: (a) cavitation first appeared at the leading edge in the form of the attached cavity and gradually expand along the hydrofoil surface toward the tail edge; (b) With the attached cavity...
continued to advance toward the tail edge direction, the length of the cavity unceasing increases and the adverse pressure gradient at the rear part of the cavity gradually increases. (c) The shear layer closed to rear part of the cavity clusters began to become unstable and thus the re-entrant jet initial forms under the adverse pressure gradient; (d) With the re-entrant jet becoming stronger, the cavity were broken up to two parts and the front part of the broken cavity fast shrink toward the leading edge and the rear part formed the cloud bubble clusters entrained by vortex; (e) Along with the cloud cavity moving to the downstream, the front portion of the residual attach cavity continues to grow and begin a new cycle.

As seen in the figure i and j, the residual sheet cavity closed to the leading edge is suppressed and then disappears after the cloud cavity forms at the trailing edge of the hydrofoil. What does suppress the development of the sheet cavity at the front part of the hydrofoil?

3.2. The interactions between the sheet cavitation and cloud cavitation

In order to eliminate the effects of the model size, a dimensionless parameter \( \frac{(p - \bar{p})}{\rho_0 u_0^2} \) was selected. Where \( \bar{p} \) is the averaged pressure. Figure 4 shows some pressure contour near the instant of the figure i, j within the figure 3. From the current numerical calculation, it can be seen that the local pressure peak is appeared during the process of the formation and shedding of the cloud cavity. These local pressure peaks gradually spread toward the surroundings through the pressure wave. Figure 4 shows the local pressure peaks and the spreading pressure wave induced by it suppress the development of the sheet cavity at the front part of the hydrofoil. The possible reason is that the area of the passage is changed due to the formation of the cloud cavity. And the larger cloud cavity has a blocking effect on the flow field. Then the pressure peaks form at both sides of the cloud cavity. And the pressure peak at the front part of the cloud cavity and the enough stronger of the pressure wave induced by it suppress the development of the sheet cavity at the front part of the hydrofoil. The inhibition on the sheet cavity become weaken with the decreasing of the stronger of the pressure peak and its moving to downstream due to the cloud cavity shedding toward the downstream. Subsequently, the sheet cavity regenerates.

Figure 4. Local pressure peaks and its propagating to surroundings.

Figure 5 shows the simulation results of the pressure pulse within the two cycles at the three monitor points, namely \( x/c=0.3, 0.5, 0.7 \). As seen in the figure, the pressure fluctuates in the range from the high value (corresponds to the pure liquid) to low value (corresponds to the appearance of the bubbles). Sequence 1 and 2 respectively correspond to the development of the cavity and the shedding of the cloud cavity. The pressure drop indicated by the black arrows indicates the appearance of the bubbles here. It can clearly see that after the collapse of the cloud cavity, the pressure wave propagate from the trailing edge to leading edge, followed by the pressure increases progressively, and then pressure spikes occur. With the receding of the stronger pressure wave induced by the collapse of the cloud cavity, the pressure on the suction side is repeatedly fluctuate, as seen in the region 2 of the picture. Subsequently, the pressure falls down again and the cavitation reappears, and the pressure on the suction side reduces from the leading edge to trailing edge, and the attached sheet cavity also gradually extends as the order. With the attached sheet cavity along the surface expands toward the trailing edge, the pressure gradient at the trailing part of the cavity gradually increases and the stronger of the re-entrant jet gradually becomes larger. The cavity breaks up after the re-entrant jet becoming
enough stronger. Figure 5 the region marked A circled by green dash lines indicates the influences of the breakup of the cavity on the pressure monitor curves. It can conclude that the more close to the leading edge, the bigger influences the breakup of the cavity impacts on and the longer the duration time.

![Figure 5. Instantaneous pressure signals during cavity growth/destabilization.](image)

In order to further investigate the propagation of the pressure wave on the suction side, the spatial-time history of wall pressure is made, about the instant i, j of the figure 3. As seen in the picture, the wall pressure is divided into two distinct parts. At the front part the pressure is maintained at a high level, thereby suppress the development of the sheet cavity near the front part of the hydrofoil. However, as seen in the figure, the first part of the distinct parts has been divided into two half part by pressure dimples. Overall, the pressure of the suction side gradually decreases over time. Judging from the fragments of the local time, the pressure of the first part repeatedly changes in the cycle of the fall-rise-fall. The pressure dimples dividing the first part into two parts become more clearly. The latter part, where the cloud cavity occupies, a small pressure peak rises first over time and then spreads toward the downstream, and then rises again and pulls the pressure of the zone originally occupied by the cloud cavity and propagates toward upstream.

4. Conclusions

The present work used simulation to reproduce the inhibition phenomenon of sheet cavity near the front part of the hydrofoil found and named dynamics 1 in the Leroux’s experiments. Through analysing the pressure contour of the inhibition moment of the development of the sheet cavity, the local pressure peaks induced during the process of the formation of the cloud cavity and its shedding is found. The local pressure peak and the pressure wave induced by it play an important role in suppressing the development of the sheet cavity near the front part of the hydrofoil. Then the time domain signal of the pressure recorded by some monitor points at the suction side of the hydrofoil indicates the collapse of the cloud cavity results in the emergence of the pressure spike. The breakup of the cavity induced by the re-entrant jet has a different influence on the various parts of the hydrofoil. And the more close to the leading edge of the hydrofoil, the bigger of the influences, and the longer of the duration time. Through the spatial-time history of wall pressure in the suction side, the propagation of the pressure wave on the suction side was further investigated and the phenomenon of the pressure fluctuation distinctly divided into two parts is found. The higher pressure level of the first part of the pressure curves suppresses the development of the sheet cavity near the front part of the hydrofoil.
Figure 6. Spatial-time history of wall pressure in the suction side of the hydrofoil during the formation of the cloud cavity and its shedding.

Acknowledgments
This work was supported by National Natural Science Foundation of China (Grant No.51109093), Graduate Students research and Innovation Program of Jiangsu Province (CXLX13_659), Natural Science Foundation of the Higher Education Institutions of Jiangsu Province (13KJD470001) and PAPD.

References
[1] Leroux J B, Coutier-Delgosha O and Astolfi J A 2005 Physics of Fluids 17(5) 052101
[2] Wang G Y, Senocak I, Shyy W, Ikohagi T and Cao S 2001 Progress in Aerospace Sci. 37 551-81
[3] Coutier-Delgosha O, Reboud J L and Delannoy Y 2003 Int. J. for numerical methods in fluids 42(5) 527-48
[4] Ducoin A, Huang B and Yong Y L 2012 Int. J. of Rotating Machinery 2012 Article ID 215678
[5] Coutier-Delgosha O, Fortes-Patella R and Reboud J L 2003 J. Fluid Eng. 125 38-45
[6] Moin P 2002 *Int. J. of Heat and Fluid Flow* **23** 710-20

[7] Girimaji S S 2006 *J. of Applied Mechanics* **73** 413-21

[8] Ji B, Luo X W, Arndt R E A and Wu Y L 2014 *Ocean Engineering* **87** 64-77

[9] Ji B, Luo X W, Wu Y L, Peng X X and Duan Y L 2013 *Int. J. of Multiphase Flow* **51** 33-43

[10] Leroux J B, Astolfi J A and Billard J Y 2004 *J. Fluid Eng.* **126**(1): 94-101

[11] Seo J and Lele S 2009 Numerical investigation of cloud cavitation and cavitation noise on a hydrofoil section *Proc. of the 7th Int. Symp. On Cavitation (Ann Arbor, Michigan, USA, 17-22 Aug 2009)*

[12] Shin B R, Iwata Y and Ikohagi T 2003 Computational Mechanics **30**(5-6) 388-95

[13] Zwart P J, Gerber A G and Belamri T 2004 A Two-Phase Flow Model for Predicting Cavitation Dynamics *5th International Conference on Multiphase Flow (Yokohama, Japan, 30 May-4 June 2004)*