Modern Physics Letters B, Vol. 24, No. 13 (2010) 1441–1444

DOI: 10.1142/S0217984910023827

NUMERICAL AND EXPERIMENTAL STUDY ON UNSTEADY SHEDDING OF PARTIAL CAVITATION

BIN JI*, XIANWU LUO† and YULIN WU‡
State Key Laboratory of Hydroscience & Engineering, Tsinghua University, Beijing, 100084, China
[*]jib08@mails.tsinghua.edu.cn
[†]luoxw@tsinghua.edu.cn
[‡]wyl-dhh@tsinghua.edu.cn

XIAOXING PENG§ and HONGYUAN XU¶
[§]*China Ship Scientific Research Center, Wuxi, 214082*, China
henrypxx@hotmail.com
[¶]*State Key Laboratory of Hydroscience & Engineering,Tsinghua University,Beijing, 100084, China*
xhy@tsinghua.edu.cn

Received 1 June 2009

Periodically unsteady shedding of partial cavity and forming of cavitation cloud have a great influence on hydraulic performances and cavitation erosion for ship propellers and hydro machines. In the present study, the unsteady cavitating flow around a hydrofoil has been calculated by using the single fluid approach with a developed cavitation mass transfer expression based on the vaporization and condensation of the fluid. The numerical simulation depicted the unsteady shedding of partial cavity, such as the process of cavity developing, breaking off and collapsing in the downstream under the steady incoming flow condition. It is noted that good agreement between the numerical results and that of experiment conducted at a cavitation tunnel is achieved. The cavitating flow field indicates that the cavity shedding was mainly caused by the re-entrant jet near cavity trailing edge, which was also clearly recorded by high-speed photographing.

Keywords: Unsteady cavitating flow, cavity shedding, cavitation model, numerical simulation.

1. Introduction

Unsteady cavitation is responsible for undesirable effects or even damage in hydraulic installations.[1] Especially, the cavity shedding generates both large pressure fluctuations and vibrations, and also acoustic emission due to the bubble collapse close to the solid walls. All these effects are usually very unfavorable for the operation stability of hydro machines, so it is of vital importance to discern the physical mechanisms involved in such cavitation phenomena. Up to now, though cloud cavitation has been studied by many authors,[2–5] the shedding mechanism of partial cavity has not been made clear perfectly yet due to the limitation of cavitation model.[6]

The present work is devoted to the study of the cavitating flow on the suction side around a 2D hydrofoil by experimental measurements (see Ref. 7 in detail) and numerical

---

[‡] Corresponding author.

simulation. In order to improve the accuracy of numerical modeling, a single fluid approach with modified cavitation mass transfer expression has also been proposed.

2. Numerical Model

2.1. *Physical model of cavitation*

The continuum equation, Navier-Stokes equation combined with transport equation for the vapor mass fraction shown in Eq. (1) are applied to solve the cavitating flow field, which is assumed to be locally homogeneous flow where there is no slip between liquid phase and vapor phase.

$$\frac{\partial}{\partial t}(\rho f_v)+\frac{\partial}{\partial x_j}\left(\rho u_j f_v\right)=m^+-m^- . \tag{1}$$

where f_{v} is vapor mass fraction. m^+, m^- are source terms for vaporization and condensation rate. In the proposed cavitation model, the phase change rate m^+ and m^- can be written as

$$m^+ = C_e \frac{\rho_l}{\rho_v}\frac{Max\left(p_v-p,0\right)}{0.5\rho_l V_\infty^2 t_\infty \sqrt{T}}\left(1-f_v\right) . \tag{2}$$

$$m^- = C_c \frac{\rho_l}{\rho_v}\frac{Max\left(p-p_v,0\right)}{0.5\rho_l V_\infty^2 t_\infty \sqrt{T}} f_v \cdot \tag{3}$$

where C_e, C_c are empirical parameters for vaporization and condensation, calibrated as 50 and 5000 respectively. ρ_l, ρ_v are density of liquid and vapor phase. p, p_{v}, T are static pressure, saturated vapor pressure and temperature of local flow field. V_∞ is the velocity at reference point such as the domain inlet. t_∞ is the reference time defined as c/ V_∞ (c is the foil chord length, which is regarded as the characteristic length for the calculation case). In Eqs. (2) and (3), it is noted that the phase change rate is determined by the local difference between static pressure and saturated vapor pressure, the local values of phase content, density ratio and temperature.

2.2. *Numerical methods*

The calculation was conducted by applying the solver of ANSYS-Fluent combined with the proposed cavitation model. A modified k-ε RNG turbulence model was chosen for the cavitating flow simulation (see Refs. 8, 9). For the momentum equation discretization, upwind scheme for convection term, central difference for diffusion term, and PRESTO method for pressure term were used.

3. Results and Discussion

The cavitating flow around a NACA0012 hydrofoil at the attack angle α of 8°, and cavitation number σ of 1.2 is shown in Fig. 1, where the calculated cavity distribution and the picture taken by high-speed photographing at six time intervals are presented.

The color labels indicate the vapor volume fraction of cavity. The period of cavity shedding, i.e. T, is 44.22 ms. Note that the numerical simulation reproduces the periodic evolution of cavity shedding, such as cavity development, break-off and collapse at the downstream very well if compared with the experiment. Further, there is good agreement between the numerical and experimental results concerning both the self-oscillation frequency and the maximum attached cavity length.

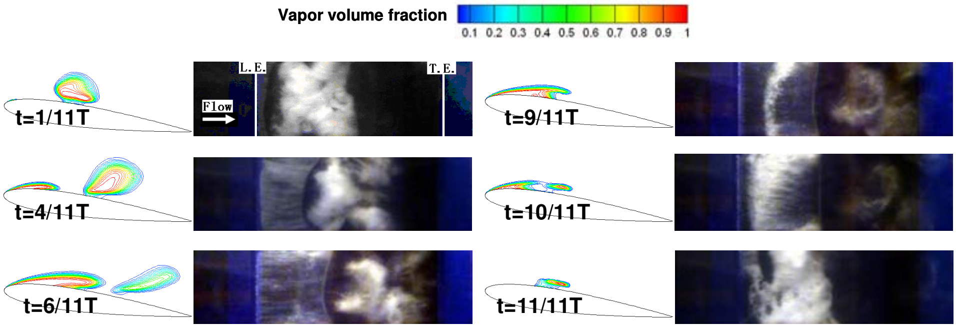
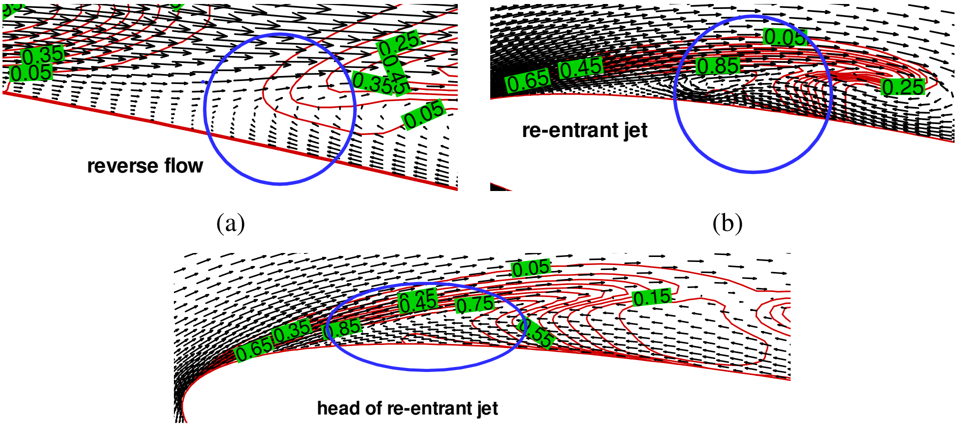


Fig. 1. Cavity shedding in a period (Left: computations. Side view; Right: Experiments. Top view).



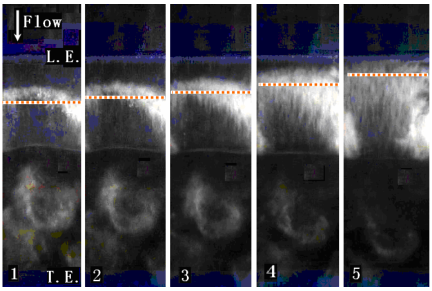
(a)

(b)

(c)

Fig. 2. (Color online) Velocity vectors at three instants: (a) $t = 6/11$ T, (b) $t = 9/11$ T, (c) $t = 10/11$ T.

Figure 2 shows velocity vectors and contour lines of vapor volume fraction (indicated by digital number) at three instants. In Fig. 2(a), there is a reverse flow just behind the attached cavity at t = 6/11 T. The reverse flow also called as re-entrant jet is close to the wall, and pushes upstream till its head arrives at the rear of the cavitation sheet near the leading edge of the foil as shown in Figs. 2(b) and 2(c). In Fig. 2(c), the re-entrant jet causes the cavity to break off. As shown in Fig. 3 where the cavity evolution from t = 9/11 T to t = 10/11 T was recorded by high-speed photographing, the experimental results confirm that the re-entrant jet is responsible for the cavity break-off and the main cloud detachment.



The head of re-entrant jet is marked by dashed lines.

The time interval between two consecutive images is 1 ms

Fig. 3. Experiment record of the cavity evolution.

4. Conclusion

The partial cavitating flow and resulting cloud cavitation around a NACA0012 hydrofoil has been investigated in this paper both numerically and experimentally. The following conclusions can be drawn:

(1) A modified transport equation governing the mass transfer for liquid/vapor mixture was developed by considering the local difference between static pressure and saturated vapor pressure, and the local values of phase content, density ratio and temperature.
(2) The numerical model based on a single fluid approach with the developed transport equation reproduced effectively the process of cavity development, break-off and final collapse at high pressure downstream.
(3) Both numerical and experimental results revealed that the unsteady cavity shedding was directly induced by the re-entrant jet along the wall.

Acknowledgment

The authors would like to thank the National Natural Science Foundation of China and Beijing Natural Science Foundation for financial supports through Project No. 50676044, 10532010 and Project No. 3072008.

References

1. R. Arndt, *Annual Review of Fluid Mechanics* **34** (**2002**) 143**.**
2. T. M. Pham, F. Larrarte and D. H. Fruman, *J. Fluid Eng*. **121** (1999) 289.
3. W. Gu, Y. S. He and T. Q. Hu, *J. Fluid Eng*. **123** (2001) 850.
4. O. Coutier-Delgosha, B. Stutz, A. Vabre and S. Legoupil, *J. Fluid Mech.* **578** (2007) 171.
5. J. Choi, C. T. Hsiao, G. Chahine and S. Ceccio, *J. Fluid Mech.* **624** (2009) 255.
6. X. Luo, Y. Zhang, J. Peng *et al.*, *J. Mech. Sci. Tech.* 22(10) (2008) 1971.
7. B. Ji, MS Thesis, China Ship Scientific Research Center, Wuxi, 2008.
8. B. Ji, F. W. Hong and X. X. Peng, *Journal of Hydrodynamics, Ser. A* **23(04)** (2008) 412.
9. O. Coutier-Delgosha, R. Fortes-Patella and J. L. Reboud, *J. Fluid Eng*. **125** (2003) 38.