**Proceedings of the ASME 2010 3rd Joint US-European Fluids Engineering Summer Meeting and
8th International Conference on Nanochannels, Microchannels, and Minichannels
FEDSM-ICNMM2010
August 1-5, 2010, Montreal, Canada**

FEDSM-ICNMM2010-30126

NUMERICAL SIMULATION OF CAVITATION CHARACTERISTIC AROUND HYDROFOIL NACA0015

Demin Liu
Tel: +86-10-62794735, Fax: +86-10-62794735, E-mail: liudm08@mails.tsinghua.edu.cn
State Key Laboratory of Hydro science & Hydraulic Engineering, Department of Thermal Engineering
Tsinghua University, Beijing 100084, China

Shuhong Liu, Yulin Wu, Hongyuan Xu
State Key Laboratory of Hydro science & Hydraulic Engineering, Department of Thermal Engineering,
Tsinghua University, Beijing 100084, China

ABSTRACT

The applicability of numerical prediction method for cavitation around hydrofoil NACA0015 was studied in this paper. For the present study, the mixture fluid methods was adopted for mass transfer between phases. For validation of this approach, simulations for the following problems were carried out: (1) leading edge cavitation on a hydrofoil; and (2)Cavitation performance and flow field analysis for a hydrofoil NACA0015. A full discussion of the results is presented below.

This paper modified cavitation mass transfer equation based on the Rayleigh-Plesset equation. The pressure difference, surface tension and the turbulence effects were considered in the new mass transfer equation on the basic of the evaporation and condensation mechanics in the micro-kinetic theory. According to the governing equations for mass, momentum, volume conservation, the hydrofoil NACA0015's cavitation characteristic was calculated by the new model. The cell-central difference finite element method was employed to discretize the governing equations. The pressure coefficient was contrasted with experiment data to validate the model. The calculation data is identical to the experiment data. As the result, it’s shown that this method can be used for the prediction of the behavior of sheet cavitation of the hydrofoil NACA0015.

Keywords: cavitation, NACA0015

INTRODUCTION

Cavitation has been an important phenomenon in technology and a challenging topic of research in engineering science for well over 100 years. Any device handling liquids is subject to cavitation, and, as is well known, it can affect the performance of turbo-machinery, resulting in a drop in head and efficiency of pumps, thereby decreasing the power output and efficiency of hydro-turbines. In general, it is a very complex vapor−liquid two-phase flow including phase changes and viscous effects cavitation.

A cavitating flow is a special two-phase flow, a turbulent, highly dynamic and highly unstable two-phase (cavity/liquid) flow in which there is not only momentum transfer between the liquid phase and cavity phase, but also mass transfer, that is, the vaporizing process and the liquidizing process. Because of this, numerical simulation of a cavitating flow has its own peculiarities and difficulties apart from the normal silt-liquid and particulate-gas two-phase flow simulation.

In recent years, there has been much progress in cavitating flow simulation. Simulation methods have developed from inviscid flow calculation to viscous flow calculation, from two-dimensional computation to three-dimensional computation, and from single-phase flow simulation to two-phase flow simulation.

1.1 Inviscid cavitating flow simulation

Beginning in the 1960s and 1970s, many cavitating flow models have been established based on the ideal fluid assumption and the singularity method. Yamaguchi and Kato (1983) proposed a cavitating flow model, which was used widely in calculation. Brewer and Kinnas (1995) used this model to calculate the flow around 2-D hydrofoils and 3-D hydrofoils, and Pellone and Peallat (1995) used it to predict the local bubbles near the hydrofoil surface. De Lange et al (1998) numerically simulated the periodic variation of bubbles in potential flows.

1.2 Single-phase flow simulation of cavitating flow

As turbulent flow simulation has developed, it has been used for cavitating flow analysis. Up to now, the most widely used method for this analysis is the single-phase flow model, although a cavitating flow is a two-phase flow consisting of a cavity phase and a liquid phase. This is called the single-phase cavitating flow model, which numerically models the flow through direct computation of the single-phase Navier-Stokes equations. A possible simplification of this type of complex flow is to assume the gas-liquid flow is a virtual single-phase, with a sharp density change as soon as the pressure drops below some critical pressure (Kubota et al 1992; Song et al 1997).

The single-phase cavitating flow model is mainly used in fixed bubble flow calculation because the position of a fixed bubble is rather stable from the point view of direct observation. Actually, it is stable in time average results. The bubbles in the flow have variations in their shape, size and length over time. The liquid flow around bubbles is the main flow area, with much greater velocity than that of vapor in bubbles. So in the model, the surfaces of bubbles can be assumed to be solid walls, on which the pressure is equal to the vaporizing pressure at a certain temperature.

In the single-phase simulation, the algorithm first simulates the whole flow field without bubbles; and then judges some areas with pressure less than the vaporizing pressure; third, treats these areas with some bubbles; and finally, recalculates the whole flow field again. This procedure is repeated to the iteration to be converged.

The single-phase simulation for cavitating flow is simple and easy, because the single turbulent simulation model and numerical method have been developed. But its application is limited to fixed-bubble cavitating flow. For other types of cavitating flow, for example, dissociative bubble flow and bubble cloud, it may be difficult to achieve an accurate simulation, because the single-phase calculation ignores the momentum and mass transfer between bubbles and the liquid.

Recently some promising results were obtained in single-phase calculation (Song et al, 1997; Arndt et al 2000; Qin et al 2003a), and models were developed to capture the main physics of complex cavitating flows.

1.3 Two-phase flow simulation of cavitating flow

The cavitating flow is actually two-phase (cavity-liquid) flow, in which there exists a mass and momentum transfer between the liquid phase and cavity phase. In contrast with the single-phase flow, in a cavitating flow there is a continuing phase (liquid phase) as well as a dispersed phase (cavity phase). The cavities are distributed in the liquid flow in the form of dispersion.

The two-phase flow calculation requires simulation of both the continuing phase and the dispersed phase. According to the different simulating models of each phase, two-phase simulation models have different schemes with different combinations of each phase model. Unlike the single-phase model, the two-phase simulation should consider interaction and the mass and momentum transfers between the continuing and the dispersed phases, as well the mass and momentum properties' jumps on the interfaces between two-phases. In two-phase simulation, the physical variables for describing two-phase flow are double those for single-phase flow. For these reasons, two-phase flow simulation is very complex (Chen Y. and Heister S D, 1994, Deshpande M. et al, 1997).

For the two-phase simulation, two models can be chosen: the mixture model and the two-fluid model.

Chen and Heister (1994) simulated the cavitating flow around an axis-symmetrical body by using the Marker and Cell method. Ventikos and Tzabiras (2000) calculated the cavitation of flow around a hydrofoil and considered the temperature variation in the flow by the pressure correction method. But these two calculations did not include turbulence.

Liu S H. et al (2005) used the $\boldsymbol{k}-\varepsilon-\boldsymbol{A}_c$ turbulence model (for liquid phase, the $\boldsymbol{k}-\varepsilon$ model; for cavity phase, the $\boldsymbol{A}_c$ algebraic model) to calculate the cavitating flow in a Francis turbine. But this model takes more time to compute and has bad iteration performance.

In this paper, the cavitation mass transfer model is modified to investigate this phenomenon, with which cavitation around hydrofoil NACA0015 is computed. The simulation results are compared with experiment results to verify the model.

NOMENCLATURE

C = hydrofoil chord length
k= turbulence kinetics coefficient
$\dot{m}$ = mass transfer rate $\quad p$ = local static pressure
$\boldsymbol{p}_v$ = cavitation pressure $\quad u$ = velocity $\quad \boldsymbol{u}_\infty$ = velocity at inlet

Greek symbols

α = volume fraction of vapor $\quad \rho$ = fluid density
k = turbulent Kinetic energy $\quad \mu$ = viscosity
σ = cavitation number $\quad \beta$ **=**attack angle

Subscripts

i, j, k = indices $\quad m$ = mixture $\quad v$ = vapor $\quad l$ = liquid

NUMERICAL METHOD FOR CAVITATING FLOW

Governing equations

In homogeneous multiphase model, a common flow field is shared by all fluids, as well as other relevant fields such as temperature and turbulence. Thus, the governing equations of homogeneous model for mass, momentum, volume conservation equation can be written as

$$\frac{\partial \rho_m}{\partial t}+\frac{\partial\left(\rho_m \bar{u}_i\right)}{\partial x_i}=0 \tag{1}$$

$$\frac{\partial\left(\rho_m \bar{u}_i\right)}{\partial t}+\frac{\partial\left(\rho_m \bar{u}_i \bar{u}_j\right)}{\partial x_j}=-\frac{\partial \bar{p}}{\partial x_j}+\frac{\partial}{\partial x_j}\left(\bar{\tau}_{ij}^{*}-\rho_m \overline{u_i^{'} u_j^{'}}\right) \tag{2}$$

$$\frac{\partial\left(\rho_l \alpha_l\right)}{\partial t}+\frac{\partial\left(\rho_l \alpha_l u_j\right)}{\partial x_j}=\rho_l \dot{m} \tag{3}$$

where,

$$\bar{\tau}^*_{ij} = \begin{cases} \mu_m \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) & i \neq j \\ 2\mu_m \frac{\partial \bar{u}_i}{\partial x_j} - \frac{2}{3}\mu_m \nabla.\vec{V} & i = j \end{cases}$$

$$\overline{\Phi} = 2\mu_m \overline{\left(\frac{\partial u_j}{\partial x_j} \right)^2} + \mu_m \overline{\left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right)^2} \quad i \neq j$$

$$-\rho_m \overline{\boldsymbol{u}_i^{'} \boldsymbol{u}_j^{'}} = \mu_t \left(\frac{\partial \bar{\boldsymbol{u}}_i}{\partial \boldsymbol{x}_j} + \frac{\partial \bar{\boldsymbol{u}}_j}{\partial \boldsymbol{x}_i} \right) - \frac{2}{3}\rho_m \boldsymbol{k}\delta_{ij} \cdot \qquad .$$

$$\rho_m = \alpha_v \rho_v + (1-\alpha_v)\rho_l \qquad \mu_m = \alpha_v \mu_v + (1-\alpha_v)\mu_l \, .$$

Where the term $\dot{\boldsymbol{m}}$ represents interphase mass transfer, i, j=1,2,3.

Cavitation mass transfer model

In most cases, the characteristic time of bubbles (growth and collapse, etc.) is less than 10^{-5}s subject to cavity size. While the time scale of turbulent fluctuation is generally far greater. Therefore, the bubble related behavior of cavities is affected by turbulence, such as inception, growth and collapse, etc. Singhal et al. (1997) has assumed as additional negative pressure drop which equals the half of the turbulence fluctuation $\boldsymbol{p}^{'}_{turb}$.

$$\boldsymbol{p}^{'}_{turb} = 0.39\rho\boldsymbol{k} \tag{4}$$

This effects is equivalent to raise the vapor pressure $\boldsymbol{p}^{'}_{v}$ by $\boldsymbol{p}^{'}_{turb}/2$ to an equivalent vapor pressure $\boldsymbol{p}^{*}_{v}$.

$$\boldsymbol{p}^{*}_{v} = \boldsymbol{p}_v + \frac{\boldsymbol{p}^{'}_{turb}}{2} = \boldsymbol{p}_v + 0.195\rho\boldsymbol{k} \tag{5}$$

The critical pressure is smaller than the vapor pressure and the difference is due to surface tension. It is negligible for large nuclei but can become important for small ones. A tension is then necessary to activate this nucleus and make it grow indefinitely.

Mass transfer model for vapor–liquid mixture based on the theory of evaporation/condensation on a plane surface, and the Kinetic theory of mass transfer are used to get the source terms. To impart the above source terms a consistent appearance between the completely mushy and completely vaporous formulations, the source term of Eq. (6) is applied only if $\boldsymbol{p} < \boldsymbol{p}^{*}_{v}$; the source term of the Sharp Interfacial Dynamics Model Eq. (7) is employed, or vice versa. Thus, the source terms $\dot{m}$ can be expressed as

If $\boldsymbol{p} < \boldsymbol{p}^{*}_{v}$:

$$\dot{m} = C_1 \frac{3(1-\alpha_v-\alpha_u)}{r} \frac{2\sigma}{2-\sigma} \left(\frac{M}{2\pi R}\right)^{1/2} (p^{*}_{v} - p) \tag{6}$$

If $\boldsymbol{p} > \boldsymbol{p}^{*}_{v}$:

$$\dot{m} = C_2 \frac{3\alpha_v}{r} \frac{2\sigma}{2-\sigma} \left(\frac{M}{2\pi R}\right)^{1/2} (p - p^{*}_{v}) \tag{7}$$

where C_1=0.13, C_2=0.01, r =0.0001, α_u is the non-dissolved gas void fraction, which takes value of $5*10^{-4}$.

SIMULATION RESULTS FOR HYDROFOIL

Model parameters for hydrofoil

For validity the cavitation model In this paper, the simulation model is a 3D NACA0015 hydrofoil in Fig.1.

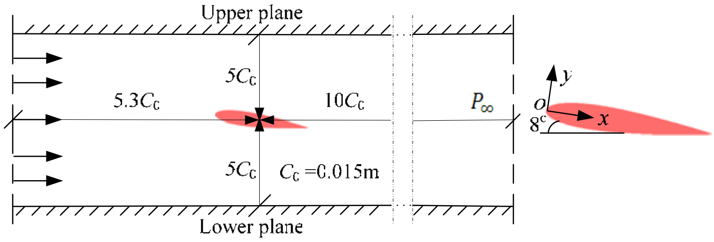


Fig.1 Simulation model and flow conditions

The leading edge and mid-chord cavitation on a hydrofoil is of particular interest for propeller cavitation studies, as it represents the two-dimensional characteristics of propeller blade cavitation. For the validation of leading edge and mid-chord cavitation on a hydrofoil, a NACA0015 hydrofoil section was selected. The span of the hydrofoil is 0.05m.The cavitation number is 1.2 and 1. The attack angle is 8° and 20°. The inlet velocity is 5m/s.

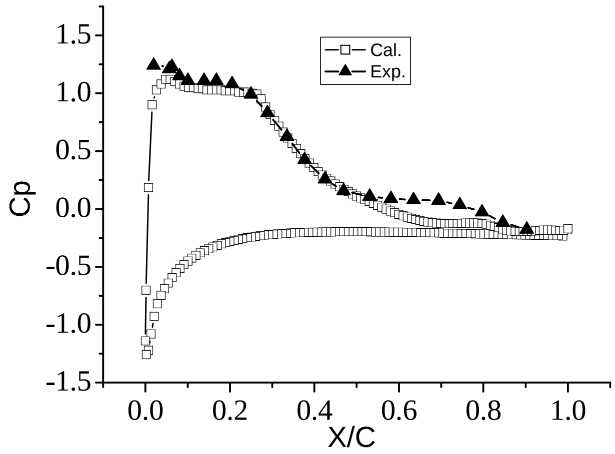
The equation definition for pressure coefficient can be written as:

$$\boldsymbol{C}_p = 2(\boldsymbol{p}_\infty - \boldsymbol{p})/\rho_l \boldsymbol{u}^2_\infty \tag{8}$$

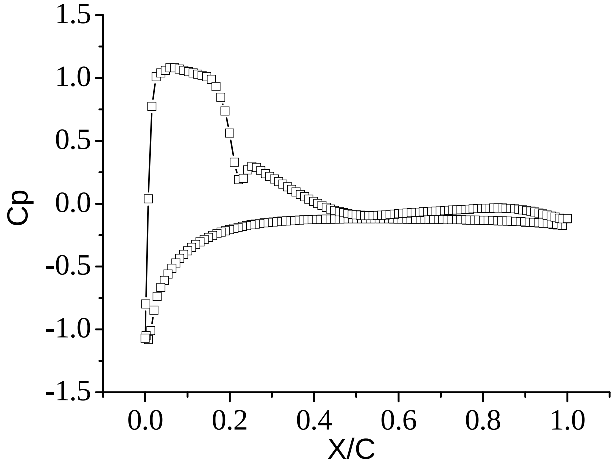
where $\boldsymbol{p}_\infty$ and $\boldsymbol{u}_\infty$ are reference pressure and velocity. In this paper, they are outlet pressure and inlet velocity of the flow channel respectively. Along the upstream and side boundaries, the undisturbed tunnel flow speed, $\boldsymbol{u}_\infty$, was imposed with a turbulence intensity of 1%. The pressure on these boundaries was obtained by a second-order accurate extrapolation. On the hydrofoil surface, a no-slip condition was imposed, i.e., the pressure obtained by the second-order accurate extrapolation. On the downstream boundary, the constant exit pressure, $\boldsymbol{p}_\infty$, which is set to match the given σ, was imposed and all other variables were extrapolated with second order accuracy.

Figure 2 shows the pressure coefficient $\boldsymbol{C}_p$ distribution along nondimensional hydrofoil chord at 3 cavitation number, 1.0, 1.2, 2.0 and two attack angle 8^0 and 20^0. However, a small negative pressure peak exists at the front of the cavity.

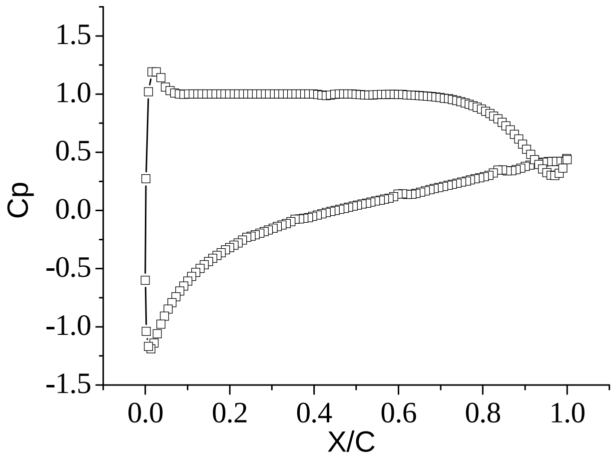
Furthermore, the cavity shape is similar to the experiment observation of attached cavity.



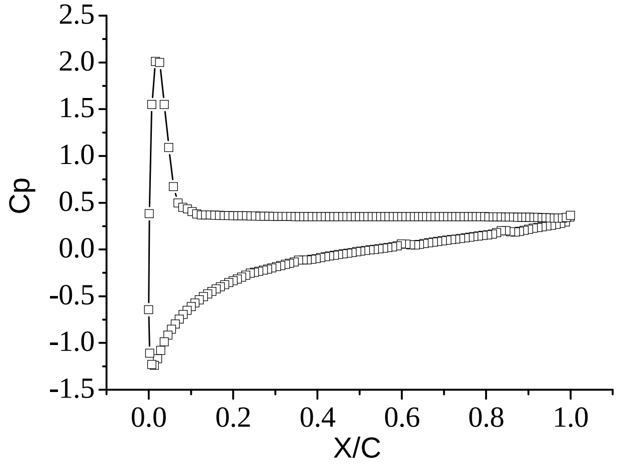
(a) $\beta = 8^0 \quad \sigma = 1.2$



(b) $\beta = 8^0 \quad \sigma = 1.0$



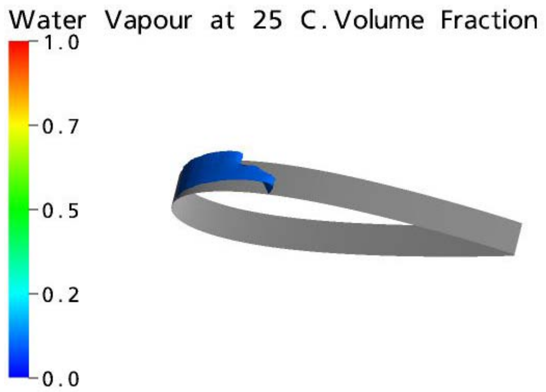
(c) $\beta = 20^0 \quad \sigma = 1.0$



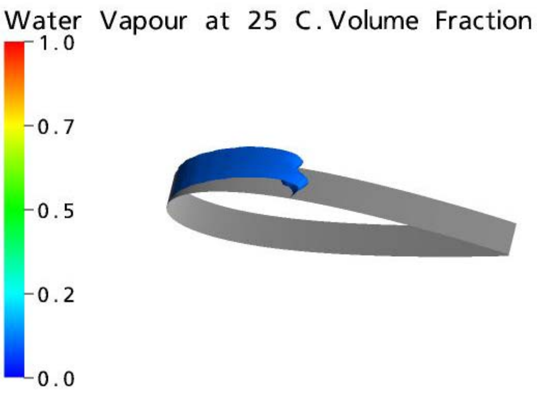
(d) $\beta = 20^0 \quad \sigma = 2.0$

Fig.2 Pressure coefficient distribution at different attack angle and cavitation number

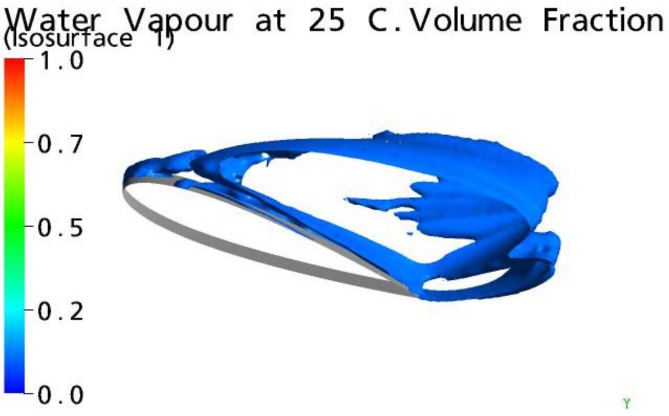
Cavitation also occurs at the trailing edge, even in the wake region (Figs. 3(c). In this case, the cavitation may occupy all the foil's back (Figs. 3(c)), which is coincident with Kubota's observation and calculation result shown as Fig. 4.



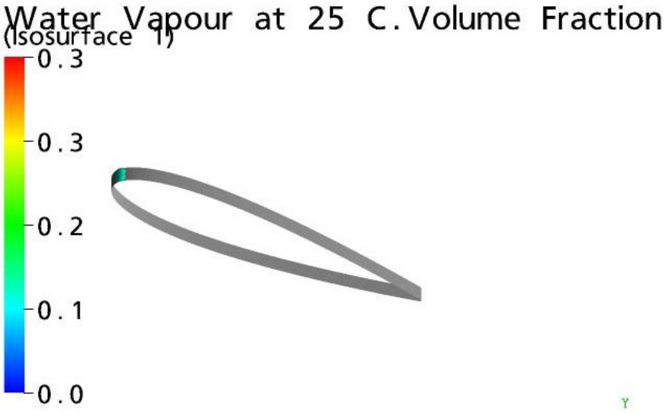
(a) $\beta = 8^0 \quad \sigma = 1.2$



(b) $\beta = 8^0 \quad \sigma = 1.0$

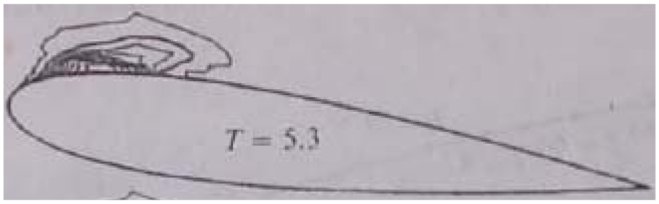


(c) $\beta = 20^0 \quad \sigma = 1.0$

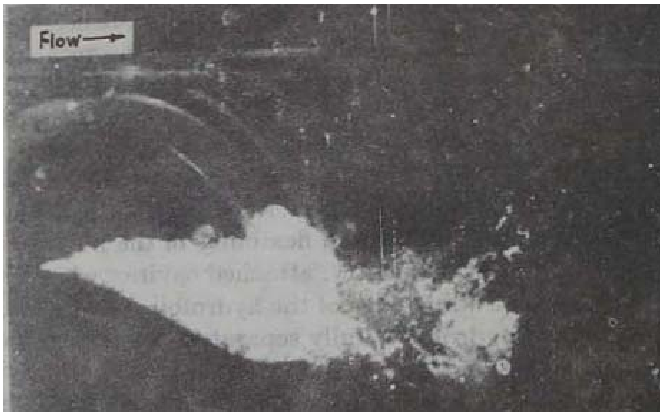


(d) $\beta = 20^0 \quad \sigma = 2.0$

Fig.3 Cavitation bubble at different attack angle and cavitation number



(a) $\beta = 8^0 \quad \sigma = 1.0$



(b) $\beta = 20^0 \quad \sigma = 1.0$

Fig.4 Kubota's cavitation bubble at different attack angle

CONCLUSIONS

1) In this paper, the improved mass transfer terms are deduced based on evaporation and condensation mechanics in the micro kinetic theory.
2) The improved model is used to simulate the cavitation flow along NACA0015 hydrofoil at different attack angle and cavitation number. The results are compared with experiment data. From the pressure coefficient and cavitation bubble shape, it can be seen that the simulation results are same trends with the experiment.

ACKNOWLEDGMENTS

The research is supported by the Chinese National Foundation of Natural Science (No. 10532010) and by the National Key Technology R&D Program in China. Authors also would like to express their thanks to the support by the Project 2010-ZY-5.

REFERENCES

Arndt R E A. Kjeldsen M. Song C C S. Keller A. 2002. Analysis of Cavitation Wake Flows, *Proceedings of the 21st IAHR Symposium on Hydraulic Machinery and Systems*, September 9-12, Lausanne.
Brewer W H, Kinnas S A. 1995. Experimental and Computational Investigation of Sheet Cavitation on a Hydrofoil. In: FED Cavitation and Multiphase Flow. *Proceedings of the 1995 ASME/JSME Fluids Engineering and Laser Anemometry Conference and Exhibition*, 210: 1-15.
Chen Y, Heister S D. 1994. A Numerical Treatment for Attached Cavitation. *J. Fluids Eng.*, 116: 613-618.
Deshpande M, Feng J, Merkle C L. 1997. Numerical Modeling of the Thermodynamic Effects of Cavitation. *J. Fluids Eng.*,

119: 420-7.

De Lange D F. De Bruin G J. 1998. Sheet Cavitation and Cloud Cavitation Re-entrant Jet and Three-dimensionality. *Applied Scic Research*, 58: 91-114.

Kubota A. Kato H. and Yamaguch H. 1992. A New Modeling of Cavitating Flows: A Numerical Study of Unsteady Cavitation on a Hydrofoil Section. *J. Fluid Mech*. 240: 59-96.

Pellone C. Peallat J M. 1995. Non-linear Analysis of Three-dimensional Partially Cavitating Hydrofoil. *Proc. of the International Symposium on Cavitation*, Deauville, France, 63-67.

Liu S H. Wu Y L. Luo X W. 2005. Numerical simulation of 3d cavitating turbulent flow in Francis turbine, *Proceedings of FEDSM2005, 2005 ASME Fluids Engineering Division Summer Meeting and Exhibition,* June 19-23, Houston, TX, USA, FEDSM2005- 77017

Qiao Qin, Charles C.S. Song.,2003, A Virtual Single-Phase Natural Cavitation Model and its Application to Cav2003 Hydrofoil [C]. Fifth International Symposium on Cavitation (CAV2003), Osaka, Japan, November 1-8,2003.

Song C C S. He J. Zhou F. and Wang G. 1997. Numerical simulation of cavitating and non-cavitating flows over a hydrofoil, *SAFL project report no. 402*, University of Minnesota.

Ventikos Y, Tzabiras G. 2000. A Numerical Method for the Simulation of Steady and Unsteady Cavitating Flows. *Comput Fluids*, 29: 63-88.

Wang G., Ostoja-Starzewski M. Large eddy simulation of a sheet/cloud cavitation on a NACA0015 hydrofoil [J].Applied Mathematical Modelling 31(2007): 417–447.

Reviewer Comments

I know did you use the post processor, ANSYS-CFX11.. What kind of CFD commercial code did you adopted to study for your paper?

Author answer:

Thank you for your question. I adopt the calculation code is ANSYS-CFX.