



## Numerical Simulation of Ventilated Cavitation behind an Axisymmetric Body

Maryam Moein-far<sup>1\*</sup>, Mahmood Pasandideh-fard<sup>2</sup> Rasool Elahi<sup>3</sup> and Iman rashidi<sup>4</sup>,

<sup>1,2,3,4</sup> Department of Mechanical Engineering, Faculty of Engineering, Ferdowsi university of Mashhad, Mashhad, Iran 91775-1111

\* Corresponding Author: Tel: +989375371702

E-mail: [moeinfar.m@gmail.com](mailto:moeinfar.m@gmail.com)

### **Abstract**

A numerical unsteady approach is followed to simulate ventilated cavitation behind an axisymmetric body. In this study liquid-gas interface is modeled, using the volume of fluid (VOF) method based on young's algorithm. Transient Navier-Stokes equations are solved along with an equation to track the cavity interface. Turbulent fluctuations in the velocity field are modeled using the Reynolds Averaged Navier-Stokes (RANS) methodology. Turbulence is assumed to be isotropic and  $RNGk-\varepsilon$  turbulence model is used in this investigation. Different free stream velocities are considered. First, the results for variation of cavity length with cavitation number have been compared to the available experimental data for a two-dimensional wedge. The good agreement between the computational and experimental results validates the present numerical method. Then this method has been used to simulate the ventilated cavitation behind a cone with the same chord and angle as the wedge. Simulation shows that, the cavity length will increase by decreasing the cavitation number. The results show that at the same cavitation numbers, the cavity lengths of the wedge are longer than those of the cone.

**Keywords:** Ventilated cavitation, Numerical simulation, Cavitation number, Axisymmetric body.

### **1. Introduction**

Cavitation is abrupt phase change phenomenon, which often takes place when pressure drops below the saturated vapor-pressure of the liquid, consequently resulting in the formation of gas filled or gas and vapor filled bubbles [1-4]. It may occur in submerged high speed vehicles as well as propellers, pumps, nozzles, and etc.

Supercavitation can be divided into natural and artificial supercavitation, the former resulting from increasing the speed of vehicles and the

latter from increasing the pressure of cavitation.

[5]. Supercavitating vehicles need to be supplied with an artificial cavity through ventilation, until they accelerate to conditions at which a natural supercavity can be sustained [6,2].

Although cavitation has disadvantages such as erosion, vibration and noise, the drag reduction associated with a natural or ventilated supercavity, has great benefit [7-10]. The high level of skin friction reduction is achieved by enveloping the body within a gaseous cavity. Only small areas at the nose and on the after-



body remain in contact with the liquid. The nose contact region called the cavitator, produces a wake in which the gaseous envelop exists and the body travels [11].

Ventilated cavitation is categorized by a non-dimensional parameter called cavitation number defined as:

$$\sigma_c = \frac{P_\infty - P_c}{\frac{1}{2} \rho V_\infty^2} \quad (1)$$

Where  $P_c$  is the sum of the vapor and air injection pressures,  $\rho$ ,  $P_\infty$  and  $V_\infty$  are the liquid density, ambient pressure and inflow velocity, respectively.

$$P_c = P_v + P_{air} \quad (2)$$

One of the other classical non-dimensional parameters often considered, is the Froude number.

$$F_r = \frac{V}{\sqrt{gd}} \quad (3)$$

Where  $d$  is a characteristic length.

Natural and Ventilated cavitation has been extensively investigated all over the world, and most of investigations focused on the shape of cavitation, the velocity and pressure distributions of flows, the hydrodynamics of models with cavitation, and so on.

Early research on supercavitating flows was performed by Efros, who employed conformal mapping techniques [12].

In 1946 Reichardt showed that it was possible to create supercavitation artificially by ventilating the flow around a body [6]. Savchenko and Semenenko [13] studied the effect of gravity and leakage rate on the shape of ventilated supercavitation experimentally. Tani et al. [14] studied ventilated airfoils both experimentally

and numerically they concluded that the ventilation improves the lift to drag ratio. Kunz et al [9] developed a multi-phase CFD method to model the flow about submerged bodies subject to natural and ventilated cavitation. Weaknesses in this modeling were observed in the cone and blunt fore-body analyses, due to single phase turbulence modeling and the inability to capture more complex off-body cavitation. The ventilated cavities were observed to yield similar solutions to natural cavities. Kuklinski et al. [11] found several different mechanisms that could cause cavity interface instabilities. Three of those mechanisms are ventilation forced instabilities, free shear instabilities and bubble oscillation through towing tank tests. Michel [15] researched the self-induced oscillation of ventilated supercavitation, and theoretically explained it. Wosnik et al. [6] measured the amount of ventilation gas required for sustaining an artificial cavity at different velocities and velocity distribution in cavitation wake. Kinnas and Young [3] used Boundary element method (BEM) techniques to model cavitating or ventilated flows. They distributed sources and dipoles along the body-cavity surface. The unknown values of these sources and dipoles were determined by imposing the dynamic condition on an assumed cavity boundary. The kinematic boundary condition was then used to update the cavity shape. Kopriva et al. [16] studied the characteristics of a low-drag partially cavitating hydrofoil (denoted as OK-2003) with ventilated cavities. They concluded that ventilated cavitation on this hydrofoil was effective in reducing drag and increasing the lift to drag ratio. Li-ping et al. [17] simulated artificial

ventilated cavity numerically and investigated the shape of ventilated cavity and the drag of underwater body. Bin et al. [18] used a three-component model based on mass transfer equation to simulate both the natural and ventilated cavitations. The effect of air flux from ventilated partial cavities on drag of bodies was studied by Amromin [19].

In this study ventilated cavitation behind an axisymmetric body (chord 60.5 mm, angle 16 deg) for different free stream velocities is simulated numerically. The VOF technique based on young's algorithm and  $RNG k - \varepsilon$  turbulent model have been used. To validate the numerical method, first the results of cavity lengths versus the cavitation numbers for a wedge (chord 60.5 mm, angle 16 deg) are compared to the results of an empirical formula obtained using the available experimental data.

This formula is:

$$\frac{l}{c} = A \sigma_c^{-n} \quad (4)$$

Where  $l$  is the cavity length and  $c$  is the chord length. The values of  $A$  and  $n$  depend on the submersion depth. In the present work  $A$  is equal to 0.34,  $n$  is equal to 1.47 and  $n$  approaches unity for small values of the submersion depths. Then the method is used to simulate the ventilated cavitation behind a cone with the same chord and angle as the wedge. In the present work the gravity effect is neglected so the Froude Number is not considered.

## 2. Governing Equations

The governing equations for the unsteady incompressible fluid flow are conservation of mass:

$$\vec{\nabla} \cdot \vec{V} = 0 \quad (5)$$

And conservation of momentum:

$$\frac{\partial \vec{V}}{\partial t} + \vec{\nabla} \cdot (\vec{V}\vec{V}) = -\frac{1}{\rho} \vec{\nabla} P + \frac{1}{\rho} \vec{\nabla} \cdot \vec{\tau} + \frac{1}{\rho} \vec{F}_b + \vec{g} \quad (6)$$

Where  $\vec{V}$  is the velocity vector,  $P$  indicates the pressure,  $\vec{F}_b$  is body force acting on fluid,  $\vec{g}$  is the acceleration due to gravity and  $\vec{\tau}$  represents Newtonian viscous stress tensor. In VOF method, the phase change boundary is simulated by a scalar field  $f$  whose value is equal to zero in the air phase and one in the liquid. When a cell is partially filled with liquid,  $f$  has a value between zero and one. The discontinuity in  $f$  is propagating through the computational domain according to:

$$\frac{df}{dt} = \frac{\partial f}{\partial t} + \vec{V} \cdot \vec{\nabla} f = S \quad (7)$$

Where  $S$  is the cavitation mass transfer source term. This equation with different mass transfer models can be used to simulate many physical phenomena such as cavitation, vaporization, and condensation. In the present work the source term is neglected.

## 3. Numerical Method

Computational domain is made up of approximately 24400 and 12476 quadrilateral cells for 2D and 2D/axisymmetric cases respectively. These grids are sufficient to ensure that all steep gradients in the flow fields are well resolved. The computational grid is shown in figure 1. The two-equation  $RNG k - \varepsilon$  turbulent model with the standard wall function is used and the finite volume method is employed to discretize the governing equations. The PISO

algorithm is considered to couple the pressure and velocity. The second-order upwind scheme is applied to discretize the momentum and the turbulent transportation equations. Unsteady calculations have been carried out and variable time step is used.

#### 4. Boundary Condition

According to computational domain shown in Fig. 1 the conditions of left, up and down boundaries are considered as velocity inlet

and the right boundary condition is the pressure outlet. A turbulent intensity of 1% and a turbulent length scale equal to one are also used at both inlet and outlet boundaries. The base of the body which is air injection port has also the velocity inlet condition and the front of the body is considered as no slip wall.

#### 5. Results and Discussion

Fig. 2 shows the comparison of numerical simulation with experimental data

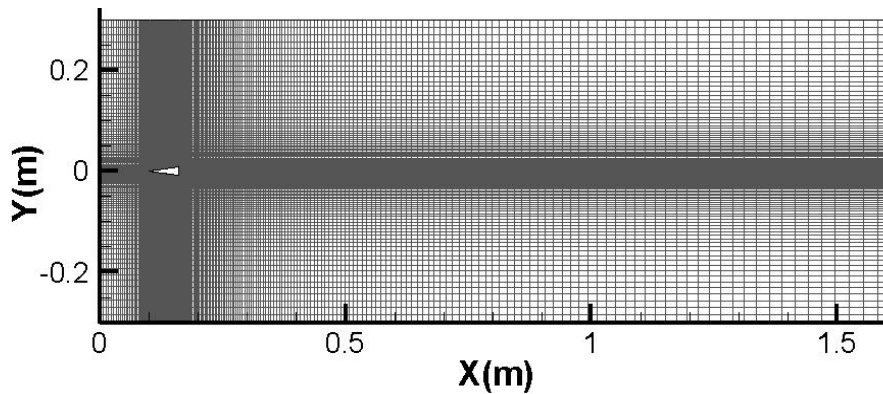


Fig. 1 Computational grid

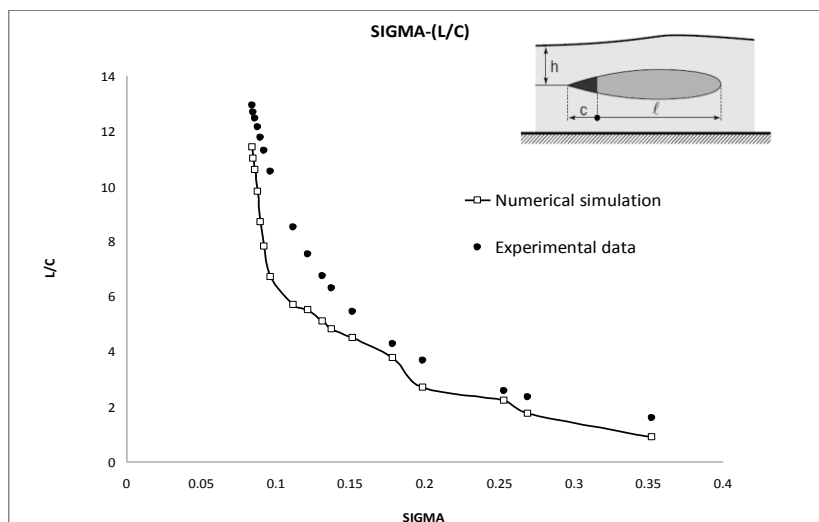


Fig. 2 Variation of the non-dimensional mean cavity length versus the cavitation number for ventilated cavities behind a two-dimensional wedge (chord=60.5 mm, angle=16 deg)

of the cavity length for a two dimensional wedge. A good agreement between the numerical and experimental results can be seen; although there are little differences between these results at low cavitation numbers. The fact that the experimental data have been measured in the three dimensional condition while the two dimensional numerical simulation is performed, may lead to these differences. Fig 3 shows the comparison of the cavity lengths of the wedge and cone (axisymmetric body) with the same chord and angle at different cavitation numbers. As it can be seen at the same cavitation

numbers the cavity lengths of the wedge are longer than those of the cone. The contours for the air volume fraction distribution with iso-pressure and stream lines at the different cavitation numbers and free stream velocities are shown in Figs 4 - 8. According to these figures the cavity length increases by decreasing the cavitation number. The same result is obvious in Figs 2 and 3. The results also show that the cavity length is independent of the free stream velocity. Therefore the most important factor that affects the cavity length is the cavitation number.

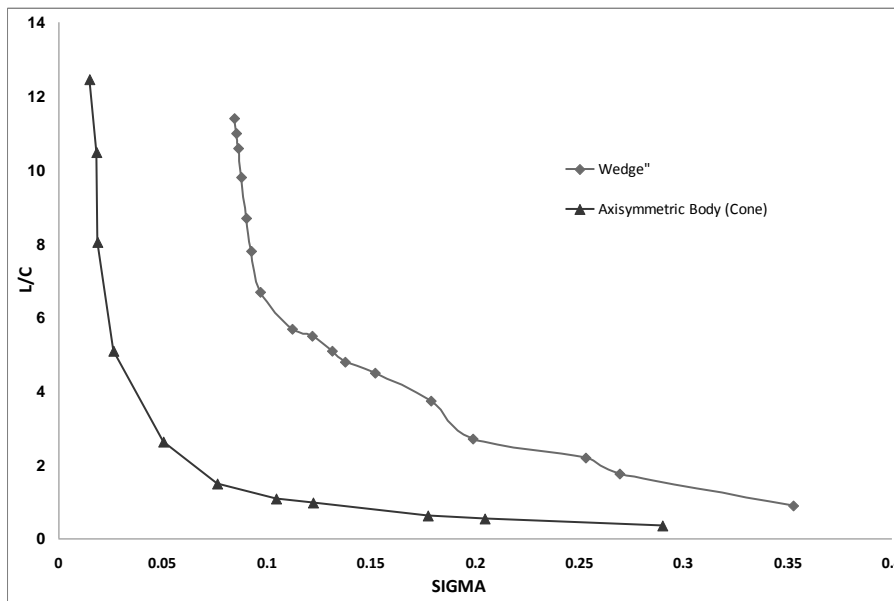


Fig. 3 Variation of the non-dimensional mean cavity length versus the cavitation number for ventilated cavities behind a wedge and a cone with the same chord and angle(chord=60.5 mm, angle=16 deg)

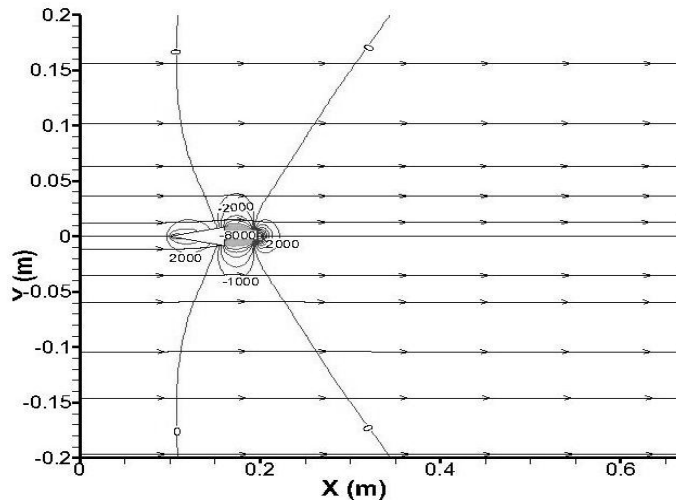


Fig 4: Air volume fraction contour, iso- Static-pressure(Pa) and stream lines( $\sigma_c = 0.20$  ,  $V_\infty = 10$  m/s)

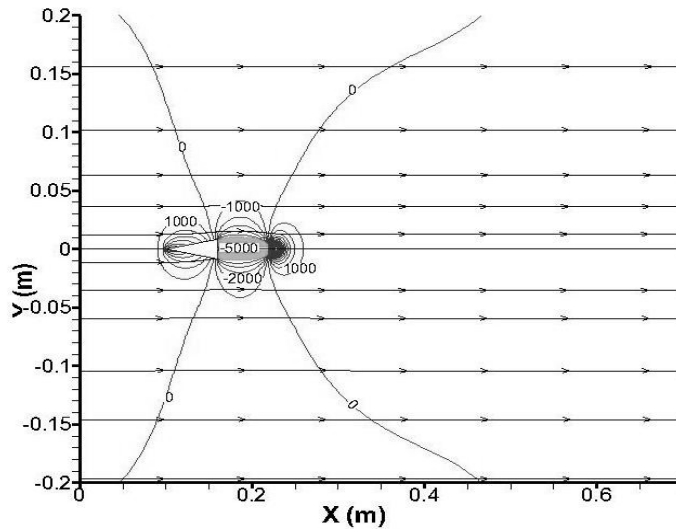


Fig 5: Air volume fraction contour, iso- Static-pressure(Pa) and stream lines( $\sigma_c = 0.12$  ,  $V_\infty = 10$  m/s)

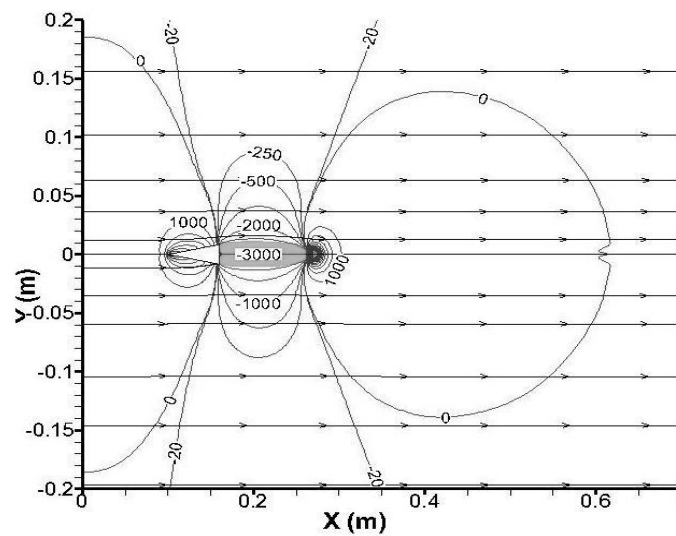


Fig 6: Air volume fraction contour, iso- Static-pressure(Pa) and stream lines( $\sigma_c = 0.076$  ,  $V_\infty = 10$  m/s)

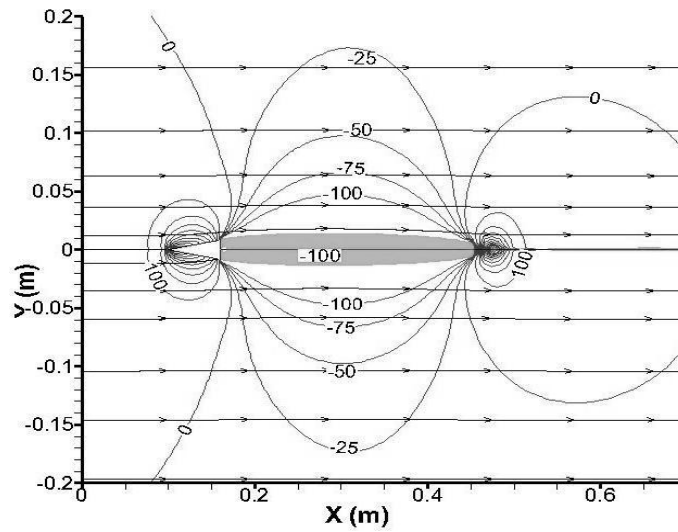


Fig 7: Air volume fraction contour, iso- Static-pressure(Pa) and stream lines( $\sigma_c = 0.026$  ,  $V_\infty = 4$  m/s)

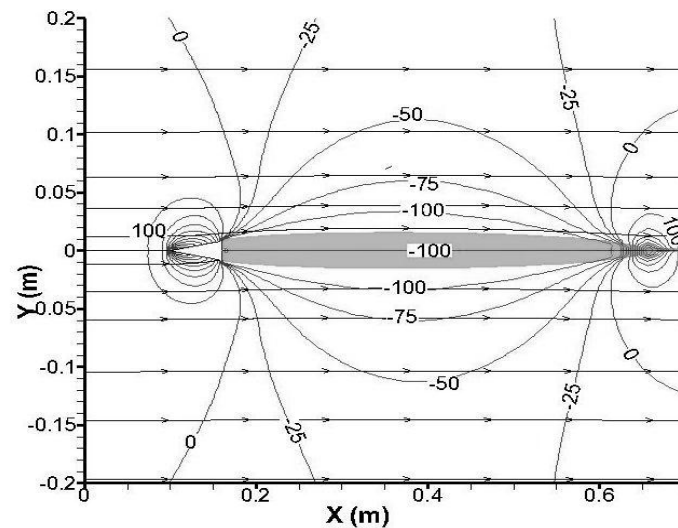


Fig 8: Air volume fraction contour, iso-Static-pressure(Pa) and stream lines( $\sigma_c = 0.019$  ,  $V_\infty = 10$  m/s)

## 6. Conclusion

In this study the unsteady ventilated cavitation behind an axisymmetric body has been investigated numerically. The deformation of the liquid-gas interface is modeled using the Volume-of-Fluid (VOF) method based on young's algorithm. RANS method and  $RNG k - \epsilon$  turbulence model, have also been used. The numerical method is successfully validated with the available experimental data of the cavity lengths for a two dimensional wedge.

The differences at the low cavitation numbers are as a result of 2D simulation instead of 3D real nature of this phenomenon. The results show the increase of the cavity length by decreasing of cavitation number. Using this numerical method, the ventilated cavitation behind a cone with the same chord and angle as the wedge, has been simulated. The main results are that the cavity lengths of the wedge are longer than those of the cone at the same cavitation numbers and the cavity length is



independent of the free stream velocity so the most important parameter affects the cavity length is the cavitation number.

### 7. References

- [1] Breslin, J.P. and Andersen, P. (1992). Hydrodynamics of ship propellers, Cambridge University Press.
- [2] Franc, J.P. and Michel, J.M. (2004). Fundamentals of cavitation, Kluwer academic publishers, New York, Boston, Dordrecht, London, Moscow.
- [3] Kinnas, S.A. and Young, Y.L. (2003). Modeling of cavitating or ventilated flows using BEM, *International Journal of Numerical Methods for Heat & Fluid Flow*, vol.13 (6), 2003, pp. 672 – 697.
- [4] Saito, Y., Takami, R., Nakamori, I. and Ikohagi, T. (2007). Numerical analysis of unsteady behavior of cloud cavitation around a NACA0015 foil, *Comput Mech*, vol.40, 2007, pp. 85 – 96.
- [5] Xue-wei, Z., Ying-jie, W., Jia-zhong, Z., Cong, W. and Kai-ping, Y. (2007). Experimental research on the shape characters of natural and ventilated supercavitation, *Journal of Hydrodynamics*, vol.19(5), 2007, pp. 564 – 571.
- [6] Wosnik, M., Schauer, T.J. and Arndt, R.E.A. (2003). Experimental Study of a Ventilated Supercavitating Vehicle, paper presented in *the 5<sup>th</sup> International Symposium on Cavitation (CAV2003)*, Osaka, Japan, Nov. 1-4.
- [7] Serebryakov, V. (2003). Supercavitation Flows with Gas Injection Prediction and Drag Reduction Problems, paper presented in *the 5<sup>th</sup> International Symposium on Cavitation (CAV2003)*, Osaka, Japan, Nov. 1-4.
- [8] Wang, G., Senocak, I., Shyy, W., Ikohagi, T. and Cao, S. (2001). Dynamics of attached turbulent cavitating flows, *Progress in Aerospace Sciences*, vol.37, 2001, pp. 551 – 581.
- [9] Kunz, R.F., Chyczewski, T.S. Boger, D.A., Stinebring, D.R. and Gibeling, H.J. (1999). Multi-Phase CFD Analysis of Natural; and Ventilated Cavitation about Submerged Bodies, paper presented in *the Proceedings of FEDSM '99, 3rd ASME/JSME Joint Fluids Engineering Conference*, San Francisco, California.
- [10] Owis, F.M. and Nayfeh, A.H. (2004). Numerical simulation of 3-D incompressible, multi-phase flows over cavitating projectiles, *European Journal of Mechanics B/Fluids*, vol.23, 2004, pp. 339 – 351.
- [11] Kuklinski, R., Henoch, C. and Castano, J., (2001). Experimental Study of Ventilated Cavities on Dynamic Test Model, paper presented in *the 4<sup>th</sup> International Symposium on Cavitation(CAV2001)*, San Francisco, California, Pasadena, USA.
- [12] Varghese, A.N., Uhlman, J.S. and Kirschner, I.N. (2003). High-Speed Bodies in Partially Cavitating Axisymmetric Flow, paper presented in *the 5<sup>th</sup> International Symposium on Cavitation (CAV2003)*, Osaka, Japan, Nov. 1-4.
- [13] Savchenko Y.N. and Semenenko V.N. (1998). Gas Absorption into Supercavity from Liquid-Gas Bubble Mixture, paper presented in *the 3rd International Symposium on Cavitation*, Grenoble, France.
- [14] Tani Y., Tanaka, K., Koshioka, Y., Nakadate, M., Kamo K. and Amano, K. (1991). Experimental and Numerical Analysis on Ventilated Airfoils, paper presented in *the AIAA*



9<sup>th</sup> Applied Aerodynamics Conference,  
September 23-25.

[15] Michel, J.M. (2001). Oscillations of Ventilated Cavities Experimental Aspects, paper presented in *the RTO AVT Lecture Series on "Supercavitating Flows*, Brussels, Belgium. February 12-16.

[16] Kopriva, J.E., Arndt, R.E.A. and Amromin, E.L. (2006). Evaluation of a Low-Drag Ventilated Partially Cavitating Hydrofoil in Steady Flow and in Periodic Gusts, paper presented in *the 6<sup>th</sup> International Symposium on Cavitation CAV2006*, Wageningen, The Netherlands, September.

[17] Li-ping, J., Cong, W., Ying-jie, W., Hai-bin, W., Jia-zhong, Z. and Kai-ping, Y. (2006). Numerical simulation of artificial ventilated cavity, *Journal of Hydrodynamics*, vol.18(3), 2006, pp. 273 – 279.

[18] Bin, J., Xian-Wu, L., Yao, Z., Hong-Juan, R., Hong-Yuan, X. and Yu-Lin, W. (2010). A three-component model suitable for natural and ventilated cavitation, *CHIN. PHYS. LETT*, vol.27(9), 2010, pp. 1– 4.

[19] Amromin, E. (2010). Microbubble drag reduction downstream of ventilated partial cavity, *Journal of Fluids Engineering*, vol.132, 2010, pp. 1– 5.