Proceedings of the International Conference on Heat Transfer and Fluid Flow
Prague, Czech Republic, August 11-12, 2014
Paper No. 181

Cavitation Control on Hydrofoils

Mohammad Mortezazadeh, Ali Katal, Khodayar Javadi
Sharif University of Technology, Department of Aerospace Engineering
Azadi Street,Tehran, Iran
mortezazadeh.md@gmail.com; alikatall@gmail.com; kjavadi@sharif.edu

Abstract – The purpose of this study is to design a passive controller on the hydrofoil, CAV 2003, to decrease and control the size of the cavitating bubble around the hydrofoil forming of the pressure drop in a specific area near the hydrofoil surface. For the purpose, a novel idea is presented here by a numerical simulation. In this idea, an appendage is located on the hydrofoil wall to decrease the size of the bubble. The location of this appendage is important to make a good condition to decrease the bubble size. Secondly, the size and height of the appendage have a significant effect on the flow around the hydrofoil and consequently select a wrong size and height can have an adverse effect. The characteristics of the numerical method, based on the cavitating bubble behaviour, are time dependent, pressure based, and finite volume. The set of Navier Stokes equations are supposed to be incompressible. Additionally, to capture the turbulent boundary layer near the hydrofoil surface, RNG k-ε model is used. The first part of this paper is allocated to verification the accuracy of the numerical simulation. The second part is the presentation on the effects of the passive controller, including the shape of the appendage and its location.

***Keywords*:** Cavitation, appendage, passive Flow control, CAV2003 hydrofoil, mixture model.

1. Introduction

Formation of vapour bubbles within a liquid occurs in two conditions, cavitation and boiling. The process of rupturing a liquid by decrease in pressure at roughly constant liquid temperature is often called cavitation (Brenne, 1995). This type of bubble formation is very common in turbo-machinery vehicles, so many studies and investigations have been done on this issue, for example it can be mentioned to the works done by Kubota et all. (1992), Alajbegovic et all. (1999), and Achneer et all. (2001). In the past decade, many researchers have tried to simulate the physics of the cavitating bubble by numerical models. Many methods have been developed like mixture and VOF. For example Nur-E-Mostafa et all. (2012) simulated the unsteady behaviour of partial cavitation on two dimensional hydrofoil by mixture model (Mostafa et al., 2012). Most works on this physic has been done by mixture method, but Roohi et all. did it by VOF (Roohi et al., 2012)

Cavitation is known by a non-dimensional number that its name is cavitation parameter:

$$\sigma = \frac{P_{ref} - P_{vap}}{\frac{1}{2}\rho V_{ref}^2} \tag{1}$$

Where P_{vap} is the vapour pressure. If Cavitation parameter is lower, that means the amount of P_{vap} is near to P_{ref} and subsequently the chance of forming bubble is higher. In this paper σ is equal to 0.8.

Cavitating conditions have both positive and negative effects. For example, Drag reduction is one of the positive effects. But mainly, cavitation is known for its violent behaviour. The vibrations, noise and erosion are samples of violent behaviour of the cavitation. Vibrations and erosion are principal reasons of destruction and performance decline.

That is caused by the fact that vaporization of water and condensation of vapour are very fast processes, much faster than the dynamics of a vapour cavity. As a result the growth and collapse of a cavity is not slowed down by these processes. Because cavitation is part of the flow, it can move rapidly from regions of low pressure into regions of a higher pressure. This leads to a very rapid collapse.

The collapse is so rapid that the local speed of sound in the fluid is exceeded and shock waves occur. The consequence is that cavitation generates noise over a wide range of frequencies, especially higher frequencies. Also the local pressure rises very strongly at collapse, leading to damage of adjacent surface. This effect is called erosion. When larger amounts of vapour are involved the implosion of cavitation can cause pressure variations in the fluid, which lead to vibrations of the cavitating structure. Cavitation can also alter the flow. This is the case on propellers when the cavitation becomes extensive. In that case the flow over the blades and the lift of the blades is altered by cavitation and the thrust of the propeller is strongly reduced. This is called thrust breakdown. Cavitation can also block or choke the flow. The volume of vapour in cavitation is much larger than the volume of water that has evaporated. In cases of extensive cavitation this leads to large volume increases and decreases when cavitation grows and collapses. The volume variations cause pressure fluctuations in the surrounding fluid, resulting in structural vibrations.

Thus, investigation and controlling the cavitation is necessary for the turbo machinery industry. In this paper a contemporary technique is shown that could decrease the size of bubble and subsequently the erosion caused by cavitation decreases.

The idea using an appendage on the hydrofoil surface has commenced from vortex generators that are very useful to control the boundary layer around aerofoils. Fig. 1 shows the effect of vortex generators on the control of the boundary layer growth (Roohi et al., 2012)

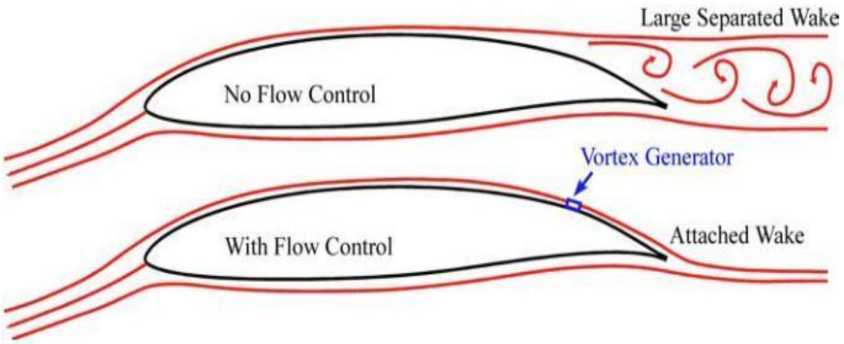

Fig. 1. Schematic showing the effect of vortex generators on the boundary layer around aerofoils. (Kerho, Kramer 2003)

2. Numerical Simulation

2.1. Governing Equations

Starting from the incompressible Navier-Stokes equations, the governing flow equations consisting of the balance equations of mass and momentum in conservative forms are:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}-((\dot{m}^{+}+\dot{m}^{-})(1-\frac{1}{\rho}))=0$$

$$\frac{\partial u}{\partial t}+\frac{\partial(\rho_m u^2+p)}{\partial x}+\frac{\partial(\rho_m uv)}{\partial y}-\frac{\partial\tau_{xx}}{\partial x}-\frac{\partial\tau_{xy}}{\partial y}=0$$

$$\frac{\partial v}{\partial t}+\frac{\partial(\rho_m uv)}{\partial x}+\frac{\partial(\rho_m v^2+p)}{\partial y}-\frac{\partial\tau_{xy}}{\partial x}-\frac{\partial\tau_{yy}}{\partial y}=0 \tag{2}$$

The equation of advection of volume fraction that indicates two phase in the domain comes in the below:

$$\frac{\partial\alpha_l}{\partial t}+\frac{\partial(\alpha_l v)}{\partial x}+\frac{\partial(\alpha_l v)}{\partial y}-(\dot{m}^{+}+\dot{m}^{-})=0 \tag{3}$$

In the above equations u and v are the components of velocity in the computational domain, and p, α_l , ρ_m are pressure, liquid volume fraction, and density of mixture, respectively. m is the amount of phase changing between to phase.

τ is the stress tensor and in the following the equations of it in each direction come:

$$\tau_{xx} = \frac{2(\mu_m + \mu_t)}{\mathrm{Re}}\left[\frac{\partial u}{\partial x} - \frac{1}{3}\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y}\right)\right]$$
$$\tau_{yy} = \frac{2(\mu_m + \mu_t)}{\mathrm{Re}}\left[\frac{\partial v}{\partial y} - \frac{1}{3}\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y}\right)\right]$$
$$\tau_{xy} = \tau_{yx} = \frac{(\mu_m + \mu_t)}{\mathrm{Re}}\left[\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right] \tag{4}$$

μ_m , μ_t is viscosity of mixture and viscosity related to the turbulence model, respectively. ρ_m , μ_m is obtained from the mixture equation related to the amount of volume fraction of each phase in each cell, i.e.:

$$\mu_m = \alpha_l(1 - \mu_v) + \mu_v \tag{5}$$

μ_v is the vapour viscosity.

2.2. Numerical Model

To calculate cavitation around the hydrofoil, an implicit finite volume method associated with multiphase and mixture model is used. As a turbulence model, we use RNG k-ε model with added wall treatment. To introduce the Reynolds number the chord length of the hydrofoil is used and in the paper the value of that is Re=5.9*105. The amount of y+ is less than 5.

A second order central scheme is used to discretize the source terms, viscosity terms, and pressure terms. The convective terms is discretized by the second order implicit scheme. For solve of incompressible equations we need a method to couple the pressure and velocity together. For the purpose, the pressure based solver SIMPLE is used.

2.3. Geometry and Computational Domain

The flow field around the hydrofoil is modelled in two dimensions. The schematic view of the CAV2003 hydrofoil geometry and the computational domain around that are presented in Fig. 2. The hydrofoil's chord is c=0.1m and is located in the middle of the domain. The hydrofoil's angle of attack is 7°. The upper surface equation of the hydrofoil is provided:

$$\frac{y}{c} = 0.11858\left(\frac{x}{c}\right)^{0.5} - 0.02972\left(\frac{x}{c}\right) + 0.00593\left(\frac{x}{c}\right)^{2.0} - 0.07272\left(\frac{x}{c}\right)^{03.0} - 0.002207\left(\frac{x}{c}\right)^{4} \tag{6}$$

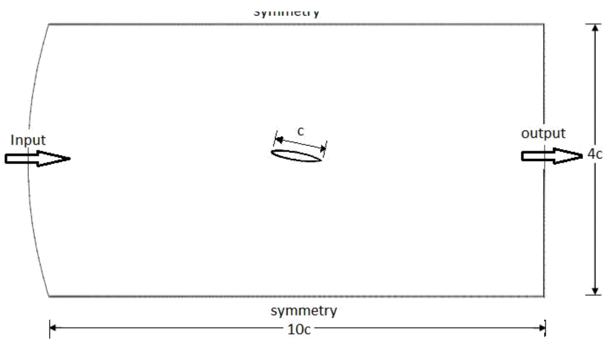

Fig. 2. Schematic diagram of the hydrofoil CAV2003, computational domain and boundary conditions

3. Results

In this section, at first CAV2003 without any passive controller is simulated and the results compared with the references to validate the numerical model. Initial conditions are in Table. 2.

Table. 1. Initial conditions

$\mathrm{Re} = 5.9 \times 10^5$	$\sigma = 0.8$
angleof attack = 7^O	$P_{ref} = 101325\ Pa$
$V_{ref} = 6.0\ m / s$	$P_v = 98929.32\ Pa$
$\rho = 998.0\ kg / \mathrm{m}^3$	$\mu = 0.001\ N / m^2 s$

The present result of the pressure distribution on CAV2003 surface in comparison to Mostafa et al., (2012) and Kawamura and Sakuda (2003) is shown in Fig. 3. The results have a good agreement.

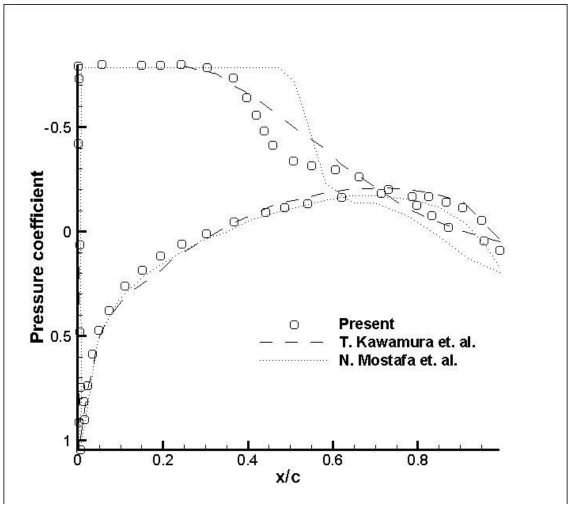

Fig. 3. Comparison of the pressure coefficient on the hydrofoil surface at $\sigma = 0.8$

The comparison between lift and drag coefficient at the time average values for the present work and others is clarify that the method is accurate and the results are dependable.

Table. 2. Comparison of the time-averaged lift and drag coefficient at cavitation number $\sigma = 0.8$

	C_L	C_D
Present	0.413	0.068
Mostafa (2012)	0.44	0.077
Pouffary (2003)	0.456	0.0783
Courier-Delgosha (2003)	0.450	0.07
Kawamura (2003)	0.399	0.047
Yoshinori (2003)	0.417	0.0638

The time history of the lift and drag coefficient at the time average values are shown in Fig. 4.:

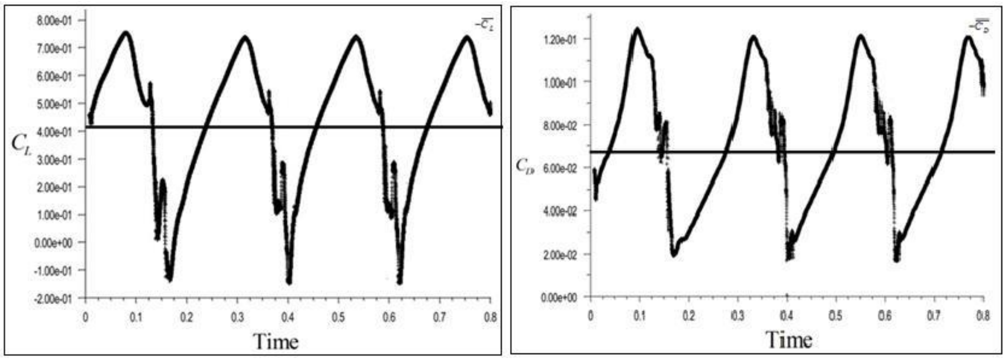

Fig. 4. Time history of drag coefficient at $\sigma = 0.8$

In the following, the contour of volume fraction at three different times is presented in Fig. 5. The bubble sizes in the present work are similar to the Mostafa et al., (2012)

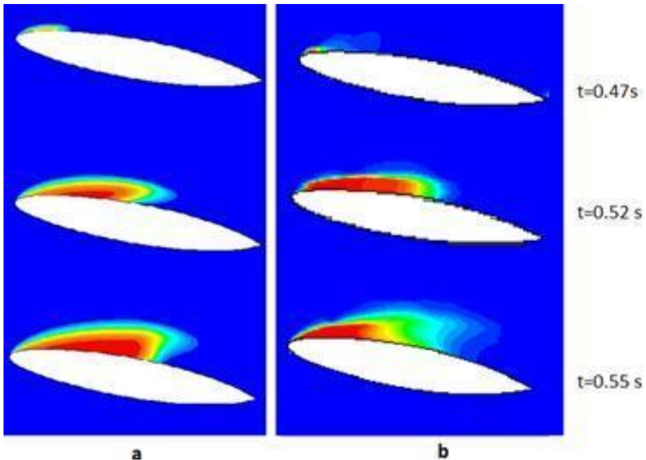

Fig. 5. Comparison of the volume fraction at different times in the case of $\sigma = 0.8$ a) Present, b) N. Mostafa et. al.

Next section in this part is allocated to the results related to the appendage's effect on control the bubble size and decrease the erosion. In this work size of the appendage is assumed corresponding to size of vortex generator on the aerofoil. Its location is near the growing of the cavitating bubble. It has been tried to use a good shape and location for this purpose. In Fig. 6. schematic of the appendage is shown near the leading edge:

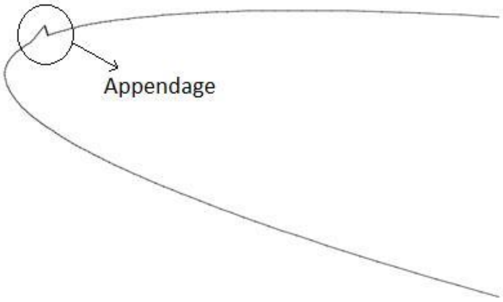

Fig. 6. Schematic diagram of the appendage located on the hydrofil CAV2003.

For initial conditions mentioned in table. 1. the results are presented here. When this idea is used to control the size of bubble, it can be seen that the periodic behaviour of the cavitation growth and collapse is changed when an appendage is used on the hydrofoil. In Fig. 7. Lift and drag coefficients are shown in time, a) CAV2003 without appendage and b) with appendage.

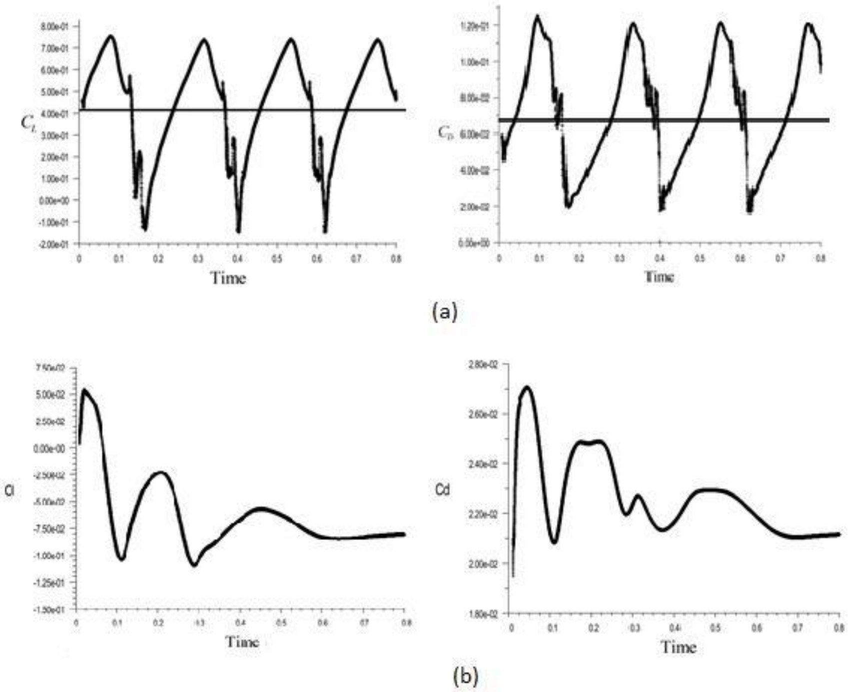

Fig. 7. Lift and drag coefficients on CAV2003 a)without appendage, b)with appendage.

Then the location of appendage is changed and this change caused the bubble size and time of its period to change. In Fig. 8. this change is presented.

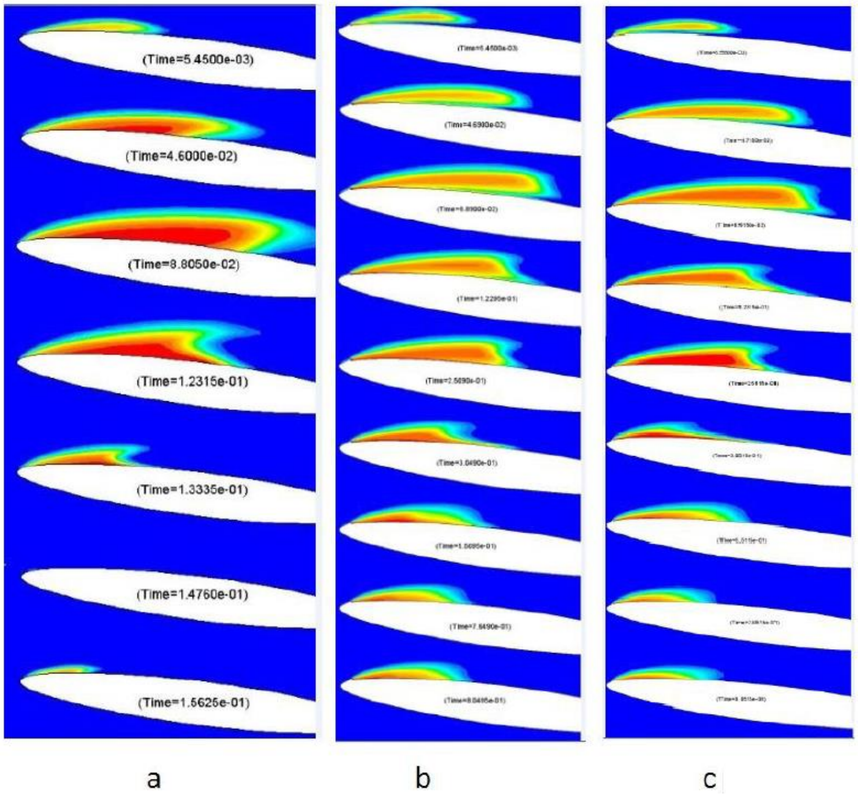

Fig. 8. Comparison between CAV2003 a)simple b)with small appendage c)with big appendage.

According to Fig. 8., it is clear that growth and collapse of the cavitating bubble arount the hydrofoil without any controller has a periodic condition, but when an appendage is used on the hydrofoil surface it caused this period to be destroyed, consequently amount of collapsing of the bubble decreases and the erosion because of the cavitation decreases.

Using an appendage around the hydrofoil has another advantage and it is decreasing the drag around the hydrofoil. It is because of the separation behind the appendage, so the area contacting with the flow decreases and the drag decreases too. In the following the velocity vectors near the appendage are shown in Fig. 9.

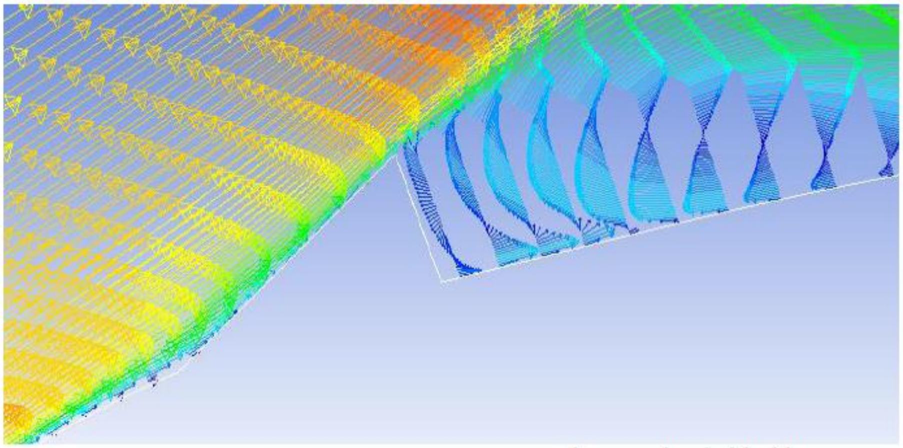

Fig. 9. Velocity vectors near the appendage and separation behind it

4. Conclusion

In some industry, such as marine and shipping, cavitation and its adverse effects are very common. So many researchers try to decrease the effects of cavitation on the instrument. A novel idea to control the growth and collapse of the bubble around the CAV2003 hydrofoil was investigated here. Enhanced an appendage near the beginning the growth of the bubble on the hydrofoil surface caused the bubble growth and collapse period decreases and consequently erosion, because of the cavitation, decreases. Separation behind the appendage causes the drag reduces and it is next advantage of this idea.

References

Brennen Ch. E. (1995).“ Cavitation and Bubble Dynamics ” Oxford University Press.

Coutier-Delgosha. O., Jacques. (2003).) A.A., Numerical reduction of cavitating flow on a two-dimensional symmetrical hydrofoil with a single fluid model, Fifth International Symposium on Cavitation (CAV2003), Osaka, Japan, Nov, 1-4.

Kawamura T., Sakuda M. (2003). Comparison of bubble and sheet cavitation models for simulation of cavitation flow over a hydrofoil. Fifth International Symposium on Cavitation (Cav2003), Osaka, Japan, Nov. 1-4, 2003.

Kerho M. F., Kramer B. R. (2003). Enhanced Airfoil Design Incorporating Boundary Layer Mixing Devices. AIAA.

Mostafa Nur. E., Karim M., Sarker M. A. (2012). Numerical Study of Unsteady Behavior of Partial Cavitation on Two Dimensional Hydrofoils. Journal of Shipping and Ocean Engineering.

Pouffary. B., Fortes-Patela. R., Reboud J. L. Numerical simulation of cavitating flow aroud a 2D hydrofoil: A barotropic approach. Fifth International Symposium on Cavitation (Cav2003), Osaka, Japan, Nov. 1-4, 2003.

Roohi E., Zahiri A.M., Passandideh-Fard M. (2012). Numerical simulation of cavitation a two-dimensional hydrofoil using VOF method and LES turbulence model. Appl. Math. Modeling.

Yoshinori. S., Inchiro. N., Tossiaki. I. (2003). Numerical analysis of unsteady vaporous cavitating flow around a hydrofoil, Fifth International Symposium on Cavitation (CAV2003), Osaka,Japan,Nov,1-4.