Numerical analysis of unsteady cavitation shedding dynamics around NACA66 hydrofoil by large-eddy simulation

B Ji1,2, K Miyagawa1, X W Luo2, Y L Wu2 and H Y Xu2

1 Applied Mechanics and Aerospace Engineering, Waseda University, Tokyo 169-8555, Japan
2 State Key Laboratory of Hydroscience & Engineering, Tsinghua University, Beijing 100084, China

E-mail: jibin@mail.tsinghua.edu.cn

Abstract. Large-Eddy simulation (LES) coupling with mass transfer cavitation model was used to resolve the turbulent flow structure with cavitation. The results will focus on the cavitation shedding dynamics around NACA66 hydrofoil. The predicted results compare well with the experimental measurements for steady/unsteady partial cavitating flows. Numerical visualizations of cloud cavity evolution and surface pressure signals show relatively good agreement with the experimental data.

1. Introduction
The unsteady behaviors of partial cavitation shedding attract much engineering attentions since they seriously affect the hydrodynamic performance of blades and propellers. Their control is expected to improve the performance and reliability of hydraulic systems. In the past decades, lots of research, including experiments and simulations, was conducted to understand the mechanism of unsteady cavity shedding.

The various limitations of measurement techniques have resulted in noticeable efforts to use numerical simulations of cavitating flows in recent years. Many cavitation models have been based on the assumption of a homogenous equilibrium medium proposed by Kubota et al. [1], where the slip between the liquid and vapor interface is neglected and the liquid-vapor mixture is treated as a single fluid that satisfies the Navier-Stokes equations. A key point in this kind of model is how to define the mixture density. One approach is based on the state equation. Delannoy and Kueny [2] used a barotropic state equation that linked the mixture density to the static pressure. Coutier-Delgosha et al. [3] used a similar barotropic state equation together with a modified turbulent viscosity to successfully simulate cloud cavity shedding in a Venturi-type duct. Another model was a multiphase cavitation mixture model based on the transport equation for the phase change. Merkle et al. [4] introduced an additional equation for the vapor (or liquid) volume fraction including source terms for evaporation and condensation (i.e. bubble growth and collapse). Kunz et al. [5], Schnerr and Sauer [6] and Singhal et al. [7] used similar techniques with different source terms.

In cavitating flow simulations, the turbulence model is crucial because the cavitation is basically unsteady in nature and there must be strong interactions between the cavity interface and the boundary layer during the cavity development. Though the current Reynolds average Navier–Stokes (RANS) equation approach has been widely used to model turbulent flows in industry, the RANS models with
eddy viscosity turbulence models have limited capability to simulate unsteady cavitating flows and need some modifications [8-12]. Thus there have been attempts to predict the flow unsteadiness during cavitation using Large Eddy Simulation. LES model for cavitating flows are expected to give better predictions of larger-scale turbulent eddies with better accuracy with some promising results already obtained [13, 14].

Inspired by their work, in present paper we treat the unsteady cavitating flow around a hydrofoil using the LES method together with a mass transfer cavitation model. Numerical simulations of the cavitation development, cavity shedding and collapse are carried out. The results are compared with experimental data to analyze cavitation shedding dynamics.

2. Description of numerical methods
The numerical model uses the LES method to solve the unsteady Navier-Stokes equations coupled with a mass transfer cavitation model. The main features of the solver are given hereafter.

2.1. Physical cavitation model
The cavitation model used in this study was developed by Schnerr and Sauer [6]. The cavitation process is governed by the following mass transfer equation:

$$\frac{\partial (\rho, \alpha_v)}{\partial t} + \frac{\partial (\rho, \alpha_v)}{\partial x_j} = \dot{m}^+ - \dot{m}^-$$  

(1)

where \(\alpha_v\) is the vapor volume fraction. The source terms \(\dot{m}^+\) and \(\dot{m}^-\) represent the effects of evaporation and condensation during the phase change and are derived from the bubble dynamics equation for the generalized Rayleigh-Plesset equation. They are defined as:

$$\dot{m}^+ = \frac{\rho_s \rho_l}{\rho} \alpha_v (1 - \alpha_v) \frac{3}{R_b} \sqrt{\frac{2 \max(p, -p, 0)}{\rho_l}}$$  

(2)

$$\dot{m}^- = \frac{\rho_s \rho_l}{\rho} \alpha_v (1 - \alpha_v) \frac{3}{R_b} \sqrt{\frac{2 \max(p-p_v, 0)}{\rho_l}}$$  

(3)

The bubble radius is related to the vapor volume fraction, \(\alpha_v\), and the bubble number density, \(N_b\), as:

$$R_b = \left(\frac{\alpha_v}{1 - \alpha_v} \frac{3}{4\pi N_b} \right)^{1/3}$$  

(4)

where \(N_b\) is the only parameter which needs to be specified and is defined as \(10^{13}\) according to Schnerr and Sauer [6]. This cavitation model has been validated for many cases, such as cavitating flow around a 2D hydrofoil and a 3D hydrofoil [15].

2.2. Governing equations and the large-eddy simulation approach
In the mixture model for vapor/liquid two-phase flows, the multiphase fluid components are assumed to share the same velocity and pressure. The basic governing equations consist of the mass and momentum conservation equations,

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_j)}{\partial x_j} = 0$$  

(5)

$$\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial u_i}{\partial x_j} \right)$$  

(6)

where \(u_i\) is the velocity in the \(i\) direction and \(p\) is the mixture pressure. The laminar viscosity, \(\mu\), and the mixture density, \(\rho\), are defined as

$$\mu = \alpha \mu_s + (1 - \alpha) \mu_l$$  

(7)

$$\rho = \alpha \rho_s + (1 - \alpha) \rho_l$$  

(8)
Applying a Favre-filtering operation to Eqs. (5) and (6) gives the LES equations:

\[
\begin{align*}
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho \bar{u}_i)}{\partial x_i} &= 0 \\
\frac{\partial (\rho \bar{u}_i)}{\partial t} + \frac{\partial (\rho \bar{u}_i \bar{u}_j)}{\partial x_j} &= -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial \bar{u}_i}{\partial x_j} \right) - \frac{\partial \tau_{ij}}{\partial x_i} \\
\end{align*}
\]

where the over-bars denote filtered quantities. Equation (10) has an extra non-linear term that does not occur in Eq. (6):

\[
\tau_{ij} = \rho \left( \overline{u_i u_j} - \overline{u_i} \overline{u_j} \right)
\]

which are called the Sub-grid Scale (SGS) stresses and need to be modeled. One commonly used SGS model is the eddy-viscosity model, which assumes that the SGS stresses are proportional to the modulus of the strain rate tensor, \( \overline{S_{ij}} \), of the filtered large-scale flow,

\[
\tau_{ij} = \frac{1}{3} \tau_{ks} \delta_{ij} = -2 \mu \bar{S}_{ij}
\]

where \( \bar{S}_{ij} \) is the rate-of-strain tensor for the resolved scale and the sub-grid scale turbulent viscosity, \( \mu_t \), is closed by the LES Wall-Adapting Local Eddy-Viscosity (WALE) model [16]. The main advantages of the LES WALE model [16] over the LES Smagorinsky model [17] are its ability to reproduce the laminar to turbulent transition with the design of the model to return the correct wall-asymptotic \( y^+ \) variation of the SGS model.

The sub-grid scale turbulent viscosity, \( \mu_t \), and the rate-of-strain tensor for the resolved scale, \( \overline{S}_{ij} \), are modeled in the LES WALE model as [16]:

\[
\mu_t = \rho L_s^2 \left( \overline{S_{ij}^d S_{ij}^d} \right)^{3/2} + \left( \overline{S_{ij}^d S_{ij}^f} \right)^{3/4}
\]

\[
\overline{S}_{ij} = \frac{1}{2} \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right)
\]

\[
S_{ij}^d = \frac{1}{2} \left( \overline{g_{ij}^d} + \overline{g_{ji}^d} \right) - \frac{1}{3} \delta_{ij} \overline{g_{kk}^d}, \quad \overline{g_{ij}} = \frac{\partial \bar{u}_i}{\partial x_j}, \quad L_s = \min \left( kd, C_s V^{1/3} \right)
\]

where \( L_s \) is the sub-grid scale mixing length, \( k \) is von Karman’s constant, \( d \) is the distance to the closest wall, \( V \) is the volume of the computational cell and \( C_s \) is the WALE constant having the value of 0.5 based on calibrations using freely decaying isotropic homogeneous turbulence [16].

2.3. Simulation setup

The time-dependent governing equations were discretized in both the space and time domains. The second order upwind scheme was used for the convective term, with the second order central difference scheme used for the diffusion term in the governing equations. The pressure staggering option (PRESTO) was selected for the pressure interpolation with the QUICK scheme used for the vapor volume fraction transport equation. The second-order implicit formulation was used for the transient term. The direct coupling method was used to solve the equations. The discrete continuity and momentum equations for the complete flow field were solved together without iterations and corrections. This solver strategy needs more computer storage, but improves the stability of the numerical procedure. The simulations were conducted using the CFD code ANSYS-Fluent.

The unsteady cavitating flow simulations were started from a steady non-cavitating flow field. Then, the cavitation model and unsteady solver were turned on for the cavitating flow simulations.
The time step was set to $1.407 \times 10^{-4}$ s ($T_{ref}/200$, where $T_{ref}=C/V_\infty$ and $V_\infty$ is the inflow velocity at the domain inlet) according to the work by Coutier-Delgosha et al. [3].

The NACA66(mod) hydrofoil was used in the present research. The hydrofoil has a relative maximum thickness of 12% at 45% chord length from the leading edge and a relative maximum camber of 2% at 50% from the leading edge. The unsteady cavitation behavior around the NACA66(mod) hydrofoil in a cavitation tunnel was extensively studied by Leroux et al. [18]. The hydrofoil chord length in the experiments was $C=0.15$ m and the foil was fixed within a 1 m long and 0.192 m wide square test section. The attack angle was 6 degrees. The inflow velocity was $V_\infty=5.33$ m/s and the static pressure was assigned according to the cavitation number, which was defined as:

$$\sigma = \left( \frac{p_{out} - p_r}{0.5 \rho V_\infty^2} \right)$$

The computational domain is shown in figure 1 as in the experiments, but simplified to a two-dimensional problem with 5 nodes in the spanwise direction. The hydrofoil was located in a channel having a height of 0.192 m. The domain inlet was 0.25 m upstream of the leading edge and the outlet was 0.75 m downstream of the leading edge. The boundary conditions had an imposed velocity at the inlet and a fixed static pressure at the outlet with free slip wall conditions at the upper and lower walls and non-slip walls on the hydrofoil. An O-H type grid was generated for the domain with sufficient refinement near the foil surface as shown in figure 1. It is noted that the values of $y+$ calculated at the first grid point away from the hydrofoil surface were within 1-2. The present grid resolution was determined based on a grid dependence study with the final mesh having about 116190 nodes.

Figure 1. Computational domain and elements near the hydrofoil.

3. Results and discussion
The experiments by Leroux et al. [18] showed that as the cavitation number decreased, the partial cavity around a 2D NACA66(mod) with a 6 degree attack angle went through various stages. For cavity lengths less than half the foil chord, the closure cavity location experienced small variations and the cavity was unsteady but was stated to be stable. In contrast, when the cavity length was longer than about half the foil chord, a cavity growth/destabilization cycle was observed with periodic shedding of vapor clouds.

The simulations in the present paper aim to analyze the unsteady cavitating flow characteristics of highly dynamic shedding behavior. The numerical results are compared with available experimental data [18] to analyze the partial cavitation transition mechanism.
Figure 2. Photographs of unsteady behavior of cavitation shedding from experiments by Leroux et al.[18]. Time between two consecutive images is 1/25 s. ($\sigma=1.25, \alpha=6$ deg).

Figure 3. Calculated vapor volume fraction contours during one cavity shedding cycle. Time between two consecutive images is 0.04 s. ($\sigma=1.2, \alpha=6$ deg).

Figure 2 shows the transient cavity patterns at seven times by experimental observation [18], while those in figure 3 are the numerical results by the LES simulation. These results show that: (i) The cavitation patterns and their evolution in one cycle predicted by the LES method agree reasonably well with the experimental observations. (ii) The evolution of the cavitating flow in each cycle is very complicated and can be divided into the following steps. The first is the development of the attached cavity from the leading edge of the hydrofoil until the cavity grows to 50% of the chord length as shown in figures 5(a)-(c). After the cavity grows more than half the chord length, the cavity shedding begins as shown in figure 5 (d). As indicated in figure 5 (e), the shedding cavity is rolled up and entrained downstream by the main flow. Finally, the shed vapor cloud collapses downstream and a
new attached sheet cavity appears at the leading edge. According to previous experimental observation,
this cavity instability is due to the interaction between the cavity interface and the re-entrant flow at
the rear part of the attached cavity.

Once the cavity shedding occurs, the wall pressure along the hydrofoil surface must vary greatly
due to the dynamic cavitation behavior. Figure 4 shows the time histories of the pressure fluctuations
at several points on the suction surface of the hydrofoil. These points are at x/c=0.3, 0.4, 0.5, 0.6 and
0.7. The experimental data is from Leroux et al. [18]. These results show that the pressures fluctuate
between a low level corresponding to the presence of vapor and a high value denoting pure liquid. For
one typical cavitation shedding cycle from instant (a) to instant (g), the cavity development can be
divided into the sheet cavity growth (instant (a)-(c)) and the cloud cavitation as well as its downstream
movement and collapse (instant (d)-(g)). The numerical and experimental results for the shedding
frequency and the magnitude of the pressure fluctuations agree fairly well. Even though the numerical
and experimental results for the shedding frequency and the magnitude of the pressure fluctuations
agree well, some discrepancy is also noticed. As shown in figure 4, the calculated pressure fluctuations
are more violent at the front surface and occur slightly earlier than those of the experimental tests,
which might be due to the no consideration of cavity structure initiated by stream-wise vortex and
beyond of present 2D calculation.

Figure 4. Comparison of pressure fluctuations at x/c=0.3-0.7 between
calculation and experiment.
4. Conclusions
In the present paper, LES simulations of unsteady cavitating flow around a NACA66 hydrofoil were performed. The numerical results were evaluated with experimental data to study the cavity evolution, shedding frequency and pressure fluctuations. Based on those results, the following conclusions can be drawn:

1. A proposed methodology including a mass transfer cavitation model and LES method is usable to simulate the unsteady cavity shedding from NACA66 hydrofoil.

2. The entire cavitating flow evolution including the cavity growth and destabilization observed experimentally is reasonably reproduced.

3. The cavity shedding generates great pressure fluctuations on the surface of the hydrofoil. Further analysis shows that there is a strong relationship between cavitation shedding dynamics and pressure fluctuations. Some discrepancy about pressure fluctuations is noticed between numerical simulation and experiments due to not resolved stream-wise vortex. And the work is in progress to consider the three dimensional effects, which might be to consider this more accuracy.

Acknowledgments
This work was financially supported by the National Natural Science Foundation of China (Project Nos. 51206087 and 51179091), the Major National Scientific Instrument and Equipment Development Project (Project No. 2011YQ07004901), and the China Postdoctoral Science Foundation funded projects (2011M500314 and 2012T50090).

References
[1] Kubota A, Kato H and Yamaguchi H 1992 J. Fluid Mech. 240 59
[2] Delannoy Y and Kueny J L 1990 Proc. Int. Conf. on ASME Fluids Engineering Division (Toronto Ontario)
[3] Coutier-Delgosha O, Reboud J L and Delannoy Y 2003 Int. J. Numer. Methods Fluids 42 527
[4] Merkle C L, Feng J Z and Buelow P E 1998 Computational modeling of the dynamics of sheet cavitation Proc. 3rd International Symposium on Cavitation (Grenoble, France, 7-10 April 1998) 2 47-54
[5] Kunz R F, Boger D A, Stinebring D R, Chyczewski T S, Lindau J W, Gibeling H J, Venkateswaran S and Govindan T R 2000 Comput. Fluids 29 849
[6] Schnerr G H and Sauer J 2001 Physical and numerical modeling of unsteady cavitation dynamics 4th Int. Conf. on Multiphase Flow (New Orleans, USA, 27 May-1 June 2001) 1
[7] Singhal A K, Athavale M M, Li H Y and Jiang Y 2002 J. Fluids Eng. 124 617
[8] Huang B and Wang G Y 2011 Chinese Phys. Lett. 28 026401
[9] Ji B, Luo X W, Wu Y L and Xu H Y 2012 Chinese Phys. Lett. 29 076401
[10] Huang B and Wang G Y 2011 J. Hydrodyn. 23 26
[11] Ji B, Luo X W, Wu Y L, Peng X X and Xu H Y 2012 Int. J. Heat Mass Tran. 55 6582
[12] Ji B, Luo X W, Wu Y L, Peng X X and Duan Y L 2013 Int. J. Multiphase Flow 51 33
[13] Wang G and Ostoja-Starzewski M 2007 Appl. Math. Model. 31 417
[14] Luo X W, Ji B, Peng X X, Xu H Y and Nishi M 2012 J. Fluids Eng. 134 041202
[15] Li D Q, Grekula M and Lindell P 2010 J. Hydrodyn. 22 741
[16] Nicoud F and Ducros F 1999 Flow Turbul. Combust. 62 183
[17] Smagorinsky J 1963 Mon. Weather Rev. 91 99
[18] Leroux J B, Astolfi J A and Billard J Y 2004 J. Fluids Eng. 126 94