Numerical evaluation of cavitation shedding structure around 3D Hydrofoil: Comparison of PANS, LES and RANS results with experiments

B Ji1,2, X X Peng3, X P Long1,2, X W Luo4, Y L Wu4
1 School of Power and Mechanical Engineering, Wuhan University, Hubei 430072 China
2 Key Lab of jet theory and new technology of Hubei Province, Hubei 430072, China
3 China Ship Scientific Research Center, Wuxi 214082, China
4 State Key Laboratory of Hydrosience & Engineering, Tsinghua University, Beijing 100084, China

E-mail: jibin@whu.edu.cn

Abstract. Results of cavitating turbulent flow simulation around a twisted hydrofoil were presented in the paper using the Partially-Averaged Navier-Stokes (PANS) method (Ji et al. 2013a), Large-Eddy Simulation (LES) (Ji et al. 2013b) and Reynolds-Averaged Navier-Stokes (RANS). The results are compared with available experimental data (Foeth 2008). The PANS and LES reasonably reproduce the cavitation shedding patterns around the twisted hydrofoil with primary and secondary shedding, while the RANS model fails to simulate the unsteady cavitation shedding phenomenon and yields an almost steady flow with a constant cavity shape and vapor volume. Besides, it is noted that the predicted shedding vapor cavity by PANS is more turbulent and the shedding vortex is stronger than that by LES, which is more consistent with experimental photos.

1. Introduction

Cavitation shedding behavior is important in engineering interest since they seriously generate the hydro-machine vibration, noise and cavitation erosion. For example, the cavitating vortex rope in the draft tube can cause lower frequency pressure fluctuation and severe cavitation surge problems. This makes us realize the significance of understanding cavitation mechanism and may help to improve the performance and reliability of hydraulic systems.

In the past, due to various limitations in measurement techniques, noticeable efforts have been made to numerically simulate cavitating flows. Many cavitation models have been based on the assumption of the homogenous equilibrium medium, 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. Coutier-Delgosha et al. [1] used a barotropic state law that linked the mixture density to the static pressure to simulate cloud cavity shedding in a Venturi-type duct. Another method has used a multiphase mixture cavitation model based on the transport equation model (TEM) for the phase change, which introduced an additional equation for the vapor (or liquid) volume fraction including source terms for evaporation and condensation processes. Comparison studies of different TEM models have been shown by Morgut et al. [2].
For simulation of cavitating flow, the turbulence model is crucial because the cavitation process is basically unsteady in nature and there must be strong interactions between the cavity interface and the boundary layer during the cavitation development. Though the current Reynolds Average Navier-Stokes (RANS) equation approach has been widely used to model turbulent flows in industry, the capability of RANS model with eddy viscosity turbulence models to simulate unsteady cavitating flows is limited and needs some modifications. There have also been attempts to predict the flow unsteadiness during cavitation using Large Eddy Simulation (LES) which are expected to more accurately predict larger-scale turbulent eddies [3-4], but most of their results are based on 2D LES solver due to very large computational resources needed. This leads to the requirement for efficient hybrid approached for application of cavitating flow, such as FBM, DES and PANS [5].

Inspired by their work, the aim of this paper is to investigate three dimensional cavitation structures and add knowledge on the impact of different turbulence models when implemented in the same code with the same mesh.

2. Description of the geometry and numerical setup

The twisted hydrofoil, studied by Foeth [6] is selected in present research. The hydrofoil is a wing of rectangular planform of a NACA0009 section with varying attack angle from 0° at the side section to 11° at the mid-section, which is symmetry with respect to its mid-span plane. This geometry can avoid the cavitation sidewall effect during the experimental test in water tunnel. So it is a very good case to resemble propeller cavitation with well defined experimental data that is easily studied.

In present paper, we compare the cavitation simulated results with a RANS approach, which is based on solving the time averaged equations, a LES approach (Ji et al. (2013)), based on the filtered equations, and a PANS approach (Ji et al. 2013), which is a bridging method from the RANS to DNS. We have previously evaluated capability of unsteady cavitation simulation with PANS model and LES model for 2D hydrofoil and 3D hydrofoil. In this paper, we briefly summarized these results and make some comparison of different models.

3. Results and discussion

The twisted design and the larger attack angle in the middle area cause the cavitation to mainly develop near the mid-span area close to the leading edge with a curved closure line. And the internal jet under below the cavitation is no longer purely with a straight upstream direction but also has a side component in the spanwise direction. The combined effect of these two components causes a very complex shedding process.

Figure 1 shows the time dependent total vapor volume $V_{cav}$ for RANS, PANS and LES. Here $V_{cav}$ was defined as:

$$V_{cav} = \sum_{i=1}^{N} \alpha_i V_i$$

(1)

where $N$ is the total number of control volumes in the computational domain, $\alpha_i$ is the vapor volume fraction in each control volume and $V_i$ is the volume of each cell.

The total vapor volume, $V_{cav}$, is a convenient parameter for understanding the transient behavior of cavitating flows. The total vapor volume calculated at each time step is shown in Fig. 1 with snapshots of three typical instants of cavitating flows with iso-surface values of $\alpha_v=0.1$ shown in Fig. 2. The choice of the vapor volume fraction value has a direct influence on the cavitation size. A higher value of the vapor volume fraction would result in less difference in terms of cavitation extent, but it tends to hide important details, i.e. interface distortion and cut-off. Based on previous work, we find the current selected vapor volume fraction is good to visualize the cavitation mechanisms.

As indicated in Fig. 1 and the left column of Fig. 2, the RANS model fails to simulate the unsteady cavitation shedding phenomenon and yields an almost steady flow with a constant cavity shape and vapor volume. As reported by many research work, this poor prediction of cavity shedding is due to the over-prediction of the turbulent viscosity in the rear part of the cavity, which suppressed the cavity shedding dynamics.
If we used PANS or LES models, the cavitation shedding behavior occurs and the vapor volume variation exhibits periodically as shown in Fig. 1 The predicted shedding frequency by PANS is 31Hz, while that for LES is 31.7 Hz, which both agree fairly well with the measured frequency (32.2 Hz). The detailed analysis of the cavitation shedding structure by PANS and LES is discussed in the
following. In instant I, the total vapor volume is a minimum after the attached cavity in the center of the hydrofoil has shed from the leading edge due to the collision between the re-entrant flow and the cavity interface. This process is called the primary shedding. Then the shedding vapor cloud becomes more turbulent and is advected downstream by the main flow, as shown in instant II. Meanwhile, the tail of the attached cavity begins to curl into a concave shape and grows quickly from the leading edge, which explains the increase in the total vapor volume from instant I to instant II in Fig. 1. After that the attached cavity grows slowly and the shedding vapor cloud quickly shrinks and finally collapses, which caused the decrease of total vapor volume in Fig. 1. It should be noted that there is a secondary shedding of both downstream lobes of the remaining attached cavity in instant III. As is shown by experimental video and our detailed analysis based upon the simulated instantaneous cavitation animation, the primary shedding event (instant I) is rather periodic while the secondary shedding of cloud cavitation (instant III) is less regular.

Comparing the flow evolutions predicted by RANS, PANS and LES with experimental results in Fig. 2 the RANS is not able to simulate cavitation shedding dynamics correctly, which is due to over-prediction of eddy viscosity in the rear part of the cavity. Although the PANS and LES reasonably reproduce the cavitation shedding patterns around the twisted hydrofoil with primary and secondary sheddings in Fig. 2 the simulated shedding vapor cloud is a little smaller than that in experiment and do not survive as long as in the experiments. Whether it is due to the insufficient grid resolution, numerical scheme, the mass transfer cavitation model behavior or the combination of all is still under investigation. Besides, as indicated by Fig. 2 it is noted that the predicted shedding vapor cavity by PANS is more turbulent than that by LES, which is more consistent with experimental photos.

4. Conclusions
We have presented the unsteady cavitating turbulent flow simulation around a twisted hydrofoil with PANS, LES and RANS. The results show that the RANS model fails to simulate the unsteady cavitation shedding phenomenon and yields an almost steady flow with a constant cavity shape and vapor volume. And the PANS and LES can reasonably predicted the 3D complicated cavitation shedding patterns including the primary shedding and secondary shedding. Besides, it is noted that the predicted shedding vapor cavity by PANS is more turbulent and the shedding vortex is stronger than that by LES, which is more consistent with experimental photos.

The main advantage of the PANS simulation over LES applied on the present grid is clear in the cloud cavitation region, where very strong vortex shedding is observed. The LES applied on present grid resulted in a weak vortex. One of the possible reasons may be due to the fact that LES is highly dependent on the mesh resolution, while the PANS is not sensitive to grid resolution. Thus, the PANS is promising since it opens a possibility for using PANS for predicting the complicated cavitation phenomena and the mechanism for the interaction between the cavitation and turbulence.

Acknowledgements
The authors would like to appreciate Dr. Foeth and Prof. & Dr. Tom van Terwisga, Delft University of Technology, for opening geometry of the Delft Twist-11 hydrofoil and the experimental data. This work was financially supported by the National Natural Science Foundation of China (Project Nos. 51206087 and 51576143).

References
[1] Coutier-Delgosha O, Reboud J and Delannoy Y 2003 Int. J. Numer. Methods Fluids 42 527-548
[2] Morgut M, Nobile E and Bilus I 2011 Int J Multiphase Flow 37(6) 620-626
[3] Roohi E, Zahiri A and Passandideh-Fard M 2013 Appl Math Model 37 6469-6488
[4] Ji B, Luo X, Peng X and Wu Y 2013 J Hydrodyn 25(4) 510-519
[5] Ji B, Luo X, Wu Y, Peng X and Duan Y 2013 Int J Multiphase Flow 51 33-43
[6] Foeth E 2008 Ph.D. Thesis, Delft University of Technology, Wageningen, the Netherlands