Ocean Engineering 99 (2015) 63–71

Contents lists available at ScienceDirect

Ocean Engineering

journal homepage: www.elsevier.com/locate/oceaneng

Numerical study on characteristic correlation between cavitating flow and skew of ship propellers

Zhifeng Zhu *

School of Electrical and Information Engineering, Anhui University of Technology, Maanshan 243032, Anhui, China

A R T I C L E I N F O

Article history:
Received 12 August 2013
Accepted 20 December 2014
Available online 30 March 2015

Keywords:
Cavitation
Propeller wake
Skew angle
Numerical analysis

A B S T R A C T

The correlation between ship propellers cavitating flow and their skew is investigated numerically using several propellers with different skew angle. With a hybrid grid strategy and sliding mesh, an RANS solver is applied to predict the vapor volume fraction and the pressure in propeller wake. The numerical predictions of the cavitation for the propeller E779A agree with the corresponding measured data in general. Under the operating condition of high blade loading, sheet cavitation becomes weak with the increase of the propeller skew angle. On the contrary, under the operating condition of low cavitation number, the sheet cavitation becomes strong with the increase of the skew. The hub cavitation is turned into strong with the increase of the skew at all the conditions. The proper increase of the propeller skew may well control propeller cavitation performance. The characteristics of the pressure signal in propeller wake coincide with the parameters of geometric model and the operating condition. Their amplitudes of the low frequency line spectra of pressure signal in propeller wake could be related to the propellers skew angle. More specifically, the amplitude of the low frequency line spectra for the highly skewed propeller seems to be linearly reduced with increased frequency.

1. Introduction

For modern surface vessels, it is difficult to avoid the cavitation due to the increase of propeller loading. The characteristics of the propeller cavitating flow are strictly related to the configuration and the operating condition of the propeller. Especially, the propeller with a high skew angle has a significantly high efficiency than the conventional propellers (Ghasseni and Ghadimi, 2011). Hence, to find out the correlation between cavitating flow and geometrical parameters of the propeller is important for blade optimization and cavitation control. It is also important for us to understand the mechanism of the cavitation. Experimental observations can show many phenomena occurring in flow field but suffer from limitations in the measurement for the internal flow of cavity (Bark et al., 2009). Numerical simulation can submit more detail information about flow field, thus it is a good complement to experiments.

At present, the numerical simulation based on a viscid-flow theory is becoming more and more attractive in the prediction of cavitating flow. Unsteady cavitating turbulent flow and shedding horse-shoe vortex structure around a twisted hydrofoil was analyzed numerically (Ji et al., 2013). It was shown that the shape of the horse-shoe vortex for the non-dimensional Q-criterion is more complicated than that of the 10% vapor fraction iso-surface and is more consistent with the experiments. The structure of the cavitating flow around a twisted hydrofoil was investigated numerically using the mass transfer cavitation model and the modified RNG k–ε model with a local density correction for turbulent eddy viscosity (Ji et al., 2014). They analyzed the flow field and revealed that cavitation promotes vortex production and increases the boundary layer thickness with local separation and the flow unsteadiness.

As for the numerical method of the prediction of propeller cavitation, the influence of grid type and turbulence model on the numerical prediction of the flow around marine propellers working in uniform inflow was analyzed by Morgut and Nobile (2012). The hybrid-unstructured meshes seem to exhibit a more diffusive character than hexa-structured meshes, and thus they are less suited for detailed investigations of the flow field. Two different turbulence models behaved similarly on both types of meshes, with the BSL-RSM turbulence model providing only slightly better predictions than the computationally more economical SST turbulence model. The cavitating flows around marine propulsors was numerically investigated by using a multi-phase RANS flow solver based on pseudo-compressibility and a homogeneous mixture model using unstructured meshes (Ahn and Kwon, 2013). To handle the relative motion between the rotating rotor and the stator, an overset mesh technique was adopted. A thorough analysis of the capability of a commercial RANS solver in conjunction with the Schnerr–Sauer cavitation model to predict the tip and the tip leak age vortex cavitation (with particular attention to the irinception) for a conventional and two ducted propellers respectively, was presented (Gaggero et al., 2014).

* Tel.: +86 13095556533; fax: +86 0555 2316596.
E-mail address: zzf@ahut.edu.cn

http://dx.doi.org/10.1016/j.oceaneng.2014.12.023
0029-8018/

To predict the pressure fluctuation induced by propeller sheet cavitation, a time domain method, which applied the acoustic theory proposed by Ffowcs Williams and Hawkings to the prediction, was focused on by Seol (2013). In some cases, this method will provide much better results than the potential-based prediction method, especially for the prediction of the location where the maximum amplitude blade rate and the pressure amplitude of higher harmonics. The pressure in propeller cavitating flow working under an uniform inflow was also calculated to investigate the effect of operating condition on features of spectra and wave shape of the pressure fluctuation (Zhu et al., 2010; Zhu, 2014). The results show that with increasing distance from propeller disk, the pressure signals at blade frequency decrease and the attenuation becomes fast with the decreased advance ratio and cavitation number. The cavitation evolution and excited pressure fluctuation around a propeller in non-uniform wake was analyzed numerically by Ji et al. (2012a). The analysis showed that the acceleration due to the cavity volume changes is the main source of the pressure fluctuations excited by the propeller cavitation.

Also under a non-uniform wake, the cavitating flow around a marine propeller was investigated numerically using Partially-Averaged Navier–Stokes method with modified k–ε model (Ji et al., 2012b). A numerical bridge from the multiphase viscous simulation of propeller cavitation hydrodynamics to its hydro-acoustics was built, and the scale effects on performances and the applicability of exist scaling law were analyzed (Yang et al., 2013).

Overall, the most present numerical researches were performed mainly to predict numerically propellers cavitation performance and pressure fluctuation in cavitating flow. Few papers focused on the influence of skew angle on the cavitating flow. The cavitating flows around a highly skewed marine model propeller in both uniform inflow and wake inflow were simulated by applying the Rayleigh–Plesset equation and the k–ω Shear Stress Transport (SST) turbulence model (Ji et al., 2011). The influence of the skew angles on the cavitation and unsteadiness performances of propellers operating in a non-uniform wake was also investigated by applying the standard k–ε turbulence and the modified Z–G–B cavitation models (Liu et al., 2012). The results demonstrate that the skewed propeller with a skew angle of 20° is the best choice for a given stern wake with a assigned thrust and the minimum force fluctuations.

In this paper, the characteristics relationship between the cavitating flow and the skew angle is investigated numerically. Four five-blade model propellers DTMB with different skew angle are used to predict the vapor volume fraction in the blade surface and the hydrodynamic performance under the different operating conditions. The pressure signals in the propeller wake and their Power Spectral Density (PSD) are also numerically predicted. With the comparison of the numerical results, the correlation between the features of caitating flow (including the cavitation, the hydrodynamic performance and the evolution of the PSD) and the skew angle is analyzed.

2. Mathematical modeling

The fluid in propeller wake is obtained by solving RANS equations with the mixture multi-phase flow model. In the mixture multi-phase flow model, it is assumed that the multi-phase flow, which contains vapor and liquid, is a mixture fluid. Therefore the mixture density ρ_m is a function of the vapor mass fraction f_v, which is given by:

$$\frac{1}{\rho_m}=\frac{f_v}{\rho_v}+\frac{1-f_v}{\rho_l} \tag{1}$$

The vapor transport equation is adopted to obtain the phase change progress induced by cavitation. It is defined as:

$$\frac{\partial}{\partial t}(\rho_m f_v)+\nabla\cdot(\rho_m \vec{v}_m f_v)=\nabla\cdot\left(\frac{\mu_t}{\sigma_v}\nabla f_v\right)+R_e-R_c \tag{2}$$

where the vapor mass fraction is expressed by $f_v=\alpha_v\rho_v/\rho_m$, $\vec{v}_m$ is the mixture velocity, σ_v is the Prandtl number of the vapor turbulence, R_e and R_c are the rates of the vapor generation and condensation, which are expressed by the Full Cavitation Model (Singhal et al., 2002). The phase change rate R_e and R_c are modified, which were presented by Zhu et al. (2010). R_e and R_c are defined as:

$$R_e=Ce\frac{k}{\gamma}\rho_l\rho_v\sqrt{\frac{2}{3}\frac{p_v-p}{\rho_l}}(1-f_v) \tag{3}$$

$$R_c=Cc\frac{k}{\gamma}\rho_l\rho_v\sqrt{\frac{2}{3}\frac{p-p_v}{\rho_l}}f_v \tag{4}$$

where $Ce=0.02$, $Cc=0.01$ are two empirical constants (Singhal et al., 2002). From the numerical results by Bardina et al. (1997) and this paper, it is concluded that the flow of the viscous sublayer

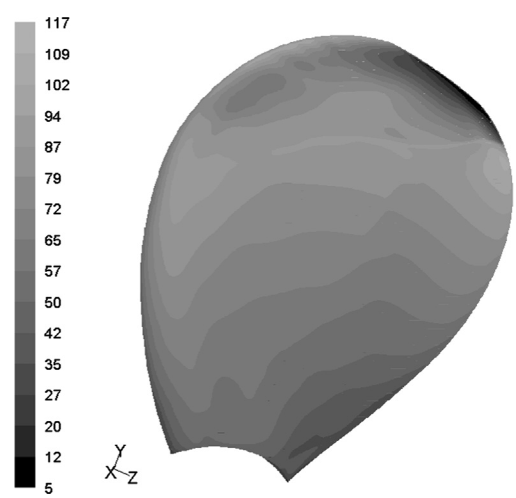

Fig. 2. Distribution of y^+ at the blade surface.

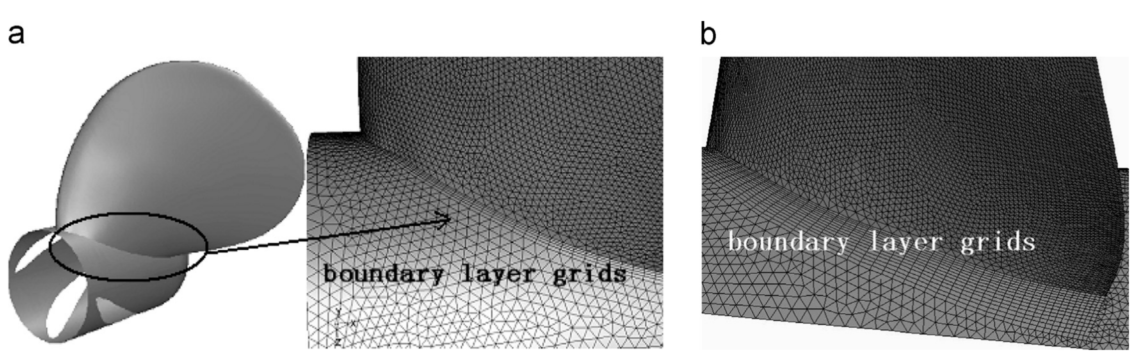

Fig. 1. Boundary layer grid. (a) Four-layer grids. (b) Fourteen-layer grids.