Ocean Engineering 160 (2018) 143–155

Contents lists available at ScienceDirect

Ocean Engineering

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

Numerical modelling of unsteady cavitation and induced noise around a marine propeller

Qin Wu [a,b], Biao Huang [b,*], Guoyu Wang [b], Shuliang Cao [a], Mingming Zhu [b]

[a] Department of Thermal Engineering, Tsinghua University, Beijing, 100084, China
[b] School of Mechanical Engineering, Beijing Institute of Technology, Beijing, 100081, China

ARTICLE INFO

Keywords:
Cavitation
Computational fluid dynamics (CFD)
Noise
Marine propeller

ABSTRACT

The objective of this paper is to numerically investigate the cavitating flow around a marine propeller and to explore the intrinsic relationships between the sheet cavitation and its radiation noise. The *k-ω* SST turbulence model with the turbulence viscosity correction and the Zwart cavitation model are introduced to the simulation of cavitating flow around a propeller in a non-uniform wake. The loading noise and cavitation noise have been predicted based on the theory for acoustic fan source and the sound radiation theory for spherical bubble respectively. The periodic cavitation development has been captured, and the periodic large pressure fluctuation around the blade has been analyzed with the dominant frequencies in accordance to the first order of the blade passing frequency. For the non-cavitation case, the high sound pressure levels mainly concentrate in the low frequency stage and decrease from the low-order to the high-order blade passing frequency. While for the sheet cavitation case, the sound pressure level at the high-order of blade passing frequencies are enhanced. The sound pressure induced by the cavitation development is periodically varied accompanied with the periodic pulsating cavity evolution, and the acoustic energy mainly focus on the low order of blade passing frequencies.

1. Introduction

As the energy shortage problem has been increasingly serious all over the world, the ship and marine engineering, like many other engineering fields, is under great pressure to decrease its environmental impact, such as reducing the engine exhaust emissions and improving the energy efficiency (Nguyen et al., 2016; Geertsma et al., 2017). Advances in power and propulsion systems bring up higher requirement of the marine propeller performance and efficiency. The propeller blades experience significant fluctuations of inflow velocity and hydraulic pressure when they rotate behind the ship in a non-uniform wake, resulting in periodic occurrence of cavitation. The propeller cavitation together with the vibratory excitation forces caused by the pressure fluctuation are considered as the primary factor of energy and efficiency losses, as well as the propeller noise (Weitendorf, 1982; Wu et al., 2015, 2017, 2018; Liu, 2017; Long et al., 2017; Wang et al., 2017a,b, 2018). Hence, it is necessary to prevent or control such undesired effects while maintaining the energy efficiency at the expected level. The propeller noise radiation has attracted more attention recently, since it is closely related to the submarine concealment in nature. The noise originates from various sources, among which the propeller cavitation plays a major role.

Much work has been conducted to the cavitation measurement and prediction of marine propellers. Pereira et al. (2004) presented an experimental investigation on a cavitating propeller by using an advanced imaging technique and established a refined map of the cavitating behavior. Stella et al. (2000) experimentally investigated the propeller wake evolution by means of the flow visualization and LDV. They performed the hydrodynamical characteristics around the propeller and the evolution features downstream. Due to the limitations in experimental measurements, many efforts have been devoted into the numerical modeling and predictions. Kinnas et al. (2003) applied the boundary element method and the vortex-lattice method to simulate the sheet cavitation around a propeller. They presented different types of the cavity patterns and the transient cavity evolutions. Gaggero et al. (2014) numerically predicted the tip leakage vortex cavitation for propellers and demonstrated the capability of the RANS solver and the Schnerr-Sauer cavitation model. Ji et al. (2011, 2012) simulated the cavitating flow around a highly skewed marine propeller in the wake flow and well predicted the pressure fluctuation caused by cavitation and blade rotation. To improve the accuracy of the simulation results, Yu et al. (2017) conducted numerical simulation of an unsteady cavitating flow around a highly-skewed propeller in a non-uniform wake based on explicit large

* Corresponding author.
E-mail address: huangbiao@bit.edu.cn (B. Huang).

https://doi.org/10.1016/j.oceaneng.2018.04.028
Received 16 August 2017; Received in revised form 30 November 2017; Accepted 8 April 2018

eddy simulations (LES), Kunz cavitation model, volume of fluid (VOF) method and a moving mesh scheme. They obtained the factors affecting the cavitation and the interaction between the cavitation structures and vortex structures.

Based on the understanding of the propeller cavitation performance, various studies have tried to explore the typical sources of propeller noise and the related mechanism. Wittekind and Schuster (2015) conducted the visual observation and pressure pulsation measurement of a full scale ship with the sheet cavitation occurred around the propeller. But they didn't estimate the radiated noise straightforwardly. Atlar et al. (2001) presented the cavitation tunnel test of a model propeller and the noise measurement of the full-scale propeller. The results showed the low-noise performance of the propeller. Kowalczyk and Felicjancik (2015) observed the sheet cavitation and tip vortex cavitation of a model scale propeller under different loading conditions, and carried out the noise measurement to investigate the hydroacoustic characteristics. They showed that the source of broadband pressure fluctuations and the related sound source. Since the hydroacoustic measurements are always difficult to perform in traditional experimental facilities, the numerical prediction of the acoustic performance has attracted more and more attention. Salvatore et al. (2006) developed an integrated hydrodynamics/hydroacoustics approach for marine propeller cavitation and analyzed the propeller-induced noise emission. Bensow and Liefvendahl (2016) applied a scale resolved Large Eddy Simulation together with an acoustic analogy to predict the propeller radiated noise. The numerical results agree well with the measurements. But the validation for radiated noise in cavitating flow is not considered. Lloyd et al. (2015) evaluated the predictive capabilities of the Ffowcs Williams-Hawkings acoustic analogy method and compared it with the Navier-Stokes solutions. They found that the FW-H method compares well with the direct RANS pressure in the propeller plane and it is sensitive to the accuracy of the input data. Pan and Zhang (2013) predicted the marine propeller noise with the formation developed by Farassat with non-uniform inflow considered. They discussed the directivity feature of the sound pressure and found that the axial forces are responsible for noise emission. A similar approach was also used by other researchers (Seol et al., 2005; Ianniello and De Bernardis, 2015).

The objective of this paper is to numerically investigate the cavitating flow around a marine propeller and to explore the intrinsic relationships between the turbulent non-cavitating/sheet cavitating flow and its radiation noise. The numerical methods are introduced in Section 2, and the detailed analysis of the propeller performance and the noise estimation are presented in Section 3, including the loading noise and the cavitation noise. Finally, the conclusions are summarized.

2. Numerical methods

2.1. Governing equations & turbulence model

Based on the assumption of the homogeneous fluid with water and vapor mixture, the incompressible and unsteady Reynolds Average Navier-Stokes (URANS) equations are used due to its balance between the accuracy and the computational cost.

$$\frac{\partial(\rho_m u_j)}{\partial x_j} = 0 \tag{1}$$

$$\rho_m \frac{\partial(u_i)}{\partial t} + \frac{\partial(\rho_m u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[(\mu_m + \mu_t)\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] + f_i \tag{2}$$

$$\rho_m = \rho_l \alpha_l + \rho_v \alpha_v \tag{3}$$

$$\mu_m = \mu_l \alpha_l + \mu_v \alpha_v \tag{4}$$

where ρ_m is the mixture fluid density, u is the velocity, with the subscripts i, j denoting the directions of the Cartesian coordinates, p is the pressure, μ_m is the mixture laminar viscosity, μ_t is the turbulent viscosity, ρ_l and ρ_v are the liquid and vapor densities respectively, μ_l and μ_v are the liquid and vapor dynamic viscosity.

The current simulation solves the URANS equations using the k-ω SST turbulence model (Menter, 1992), which applies the k-ε model away from the wall and the k-ω model near the wall. In order to improve the numerical simulations by considering the local compressibility effect of multiphase mixtures on the turbulence model, the turbulent viscosity is reduced by replacing μ_t with μ_{t_mod} (Coutier-Delgosha et al., 2003).

$$\frac{\partial(\rho_m k)}{\partial t} + \frac{\partial(\rho_m U_j k)}{\partial x_j} = P_k - D_k + \frac{\partial}{\partial x_i}\left[\left(\mu_m + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right] \tag{5}$$

$$\frac{\partial(\rho_m \omega)}{\partial t} + \frac{\partial(\rho_m U_j \omega)}{\partial x_j} = C_\omega P_\omega - \beta_\omega \rho_m \omega^2 + \frac{\partial}{\partial x_i}\left[\left(\mu_m + \frac{\mu_t}{\sigma_k}\right)\frac{\partial \omega}{\partial x_i}\right] + 2\rho_m(1-F_1)\sigma_{\omega 2}\frac{1}{\omega}\frac{\partial k}{\partial x_i}\frac{\partial \omega}{\partial x_i} \tag{6}$$

$$\mu_t = \frac{\rho a_1 k}{\max(a_1\omega; SF_2)}, \quad \mu_{t\_mod} = \mu_t f(n), \quad f(n) = \frac{\rho_v + (1-\alpha_v)^n(\rho_l - \rho_v)}{\rho_v + (1-\alpha_v)(\rho_l - \rho_v)} \tag{7}$$

where P_k and P_ω are production terms, D_k is the destruction term, F_1 is blending functions. In Eqn. (7), $n = 3$ is chosen and the validation studies can be referred to Huang et al. (2012).

2.2. Cavitation model

The cavitation process is governed by the mass transfer equation for the conservation of the liquid volume fraction, which can be defined as:

$$\frac{\partial(\rho_l \alpha_l)}{\partial t} + \frac{\partial(\rho_l \alpha_l u_j)}{\partial x_j} = \dot{m}^+ + \dot{m}^- \tag{8}$$

where $\dot{m}^+$ and $\dot{m}^-$ represent the condensation and evaporation rate for the phase change.

In this work, the cavitation model proposed by Kubota et al. (1992) is used, which is derived from the Rayleigh-Plesset equation. The growth and collapse of the bubble are governed as:

$$\frac{dR_B}{dt} = \sqrt{\frac{2(p_v - p)}{3\rho_l}} \tag{9}$$

where R_B is the radius of the spherical bubble.

Then the source and sink terms for this model are defined as:

$$\dot{m}^- = -C_{dest}\frac{3\alpha_{nuc}(1-\alpha_v)\rho_v}{R_B}\left(\frac{2}{3}\frac{p_v - p}{\rho_l}\right)^{1/2}, \quad p < p_v \tag{10}$$

$$\dot{m}^+ = C_{prod}\frac{3\alpha_v\rho_v}{R_B}\left(\frac{2}{3}\frac{p - p_v}{\rho_l}\right)^{1/2}, \quad p > p_v \tag{11}$$

where p_v is the saturated vapor pressure, α_{nuc} is the nuclei volume fraction, and C_{dest} is the constant generation rate of vapor in the region where the local pressure is less than the vapor pressure, C_{prod} is the constant rate for re-conversion of vapor back to liquid in a region where the local pressure exceeds the vapor pressure. According to Zwart et al. (2004), the model constants are: $\alpha_{nuc} = 5 \times 10^{-4}$, $R_B = 1 \times 10^{-6}$, $C_{dest} = 50$, $C_{prod} = 0.01$. Validation of the cavitation model with the assumed constants has been conducted by Huang et al. (2013, 2014).

2.3. Hydrodynamic setup

The highly-skewed propeller used in 'Seiun-Maru' ship is studied in