International Journal of Heat and Mass Transfer 55 (2012) 6582–6588

Contents lists available at SciVerse ScienceDirect

International Journal of Heat and Mass Transfer

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

Partially-Averaged Navier–Stokes method with modified k–ε model for cavitating flow around a marine propeller in a non-uniform wake

Bin Ji [a], Xianwu Luo [a,*], Yulin Wu [a], Xiaoxing Peng [b], Hongyuan Xu [a]

[a] State Key Laboratory of Hydroscience & Engineering, Tsinghua University, Beijing 100084, China
[b] China Ship Scientific Research Center, Wuxi 214082, China

ARTICLE INFO

Article history:
Received 15 November 2011
Accepted 21 June 2012
Available online 18 July 2012

Keywords:
Cavitating flow
PANS
Marine propeller
Non-uniform wake
Pressure fluctuation

ABSTRACT

Unsteady cavitating turbulent flow simulations need to be responsible for both cavitation and turbulence modeling issues. The Partially-Averaged Navier–Stokes (PANS) computational model developed from the RANS method and the k–ε turbulence model are used to model turbulent cavitating flow with a mass transfer cavitation model in the present paper. An objective of this study is to pursue more accurate estimates of unsteady cavitating flows with large-scale fluctuations at a reasonable cost. Firstly, the unsteady cavitating flow simulations over a NACA66-mod hydrofoil are performed using the PANS method with various values of the resolution control parameters ($f_k = 1 \sim 0.2, f_\varepsilon = 1$) to evaluate the numerical methods based on experimental data. The comparison with the experiments show that the numerical analysis with a $f_k = 0.2$ can predict the cavity evolution and shedding frequency fairly well. Then, cavitating flow around a marine propeller in non-uniform wake was simulated by PANS method. The calculations show that large cavity volume pulsation as the blade passes through the wake region is resolved better by the PANS method with $f_k = 0.2$ than by the RANS method with the k–ε or k–ω SST turbulence models. This can be contributed to the fact that a smaller f_k give larger cavity volume pulsations leading to increased cavity volume accelerations and larger pressure fluctuations above the propeller, while a larger f_k overestimates the turbulent viscosity along the rear part of the cavity. Finally, it is confirmed from the simulation by the PANS method with $f_k = 0.2$ that the whole process of cavitating flow evolution around the propeller in non-uniform wake can be very well reproduced including cavitation inception, sheet cavitation and tip vortex cavitation observed experimentally.

1. Introduction

Cavitation on ship propellers can cause thrust breakdown, noise, vibration and serious erosion. The reports of the Propulsion Committee [1–5] and the Cavitation Specialist Committee [6–9] of the International Towing Tank Conference (ITTC) contained state-of-the-art reviews on cavitating propellers, unsteady excitation forces induced by cavitating propellers, and propeller design in cavitating environments. The performance specifications for a modern propeller normally include a limit on the maximum levels of hull excitation pressures and/or forces. These levels should not be exceeded to achieve acceptable levels of vibration. According to the ITTC report, the principal source of excessive vibration in modern ships is partially cavitating propellers characterized by sheet cavitation on the upper half of the propeller disk and strong, developed tip vortex cavitation. Thus, accurate measurements and predictions are of importance.

Due to the various limitations of measurement techniques [10], cavitation researchers need to model large scale cavitation evolution. Potential flow methods have been used for decades to model large scale cavitation around propellers [11–14]. These methods treated the fluid outside the bubble as potential flow, while the shape and size of the bubble itself were determined from dynamic equilibrium assumptions across the bubble–liquid interface and closure conditions. Such methods are widely used today due to their inherent computational efficiency and their proven effectiveness in predicting the first order dynamics of sheet cavitating flows [15], but they have the limitations of the potential flow model applied to a flow with complex bubble geometries and inherent vortical structures. Recently, improved computational fluid dynamics (CFD) methods have demonstrated the ability to effectively analyze these flows. For marine propellers, Lindau et al. [16] used a dual-time, preconditioned, implicit artificial compressibility algorithm combined with the homogeneous multiphase model proposed by Kunz et al. [17] to model cavitating flow around a propeller. They predicted the cavity size and shape as well as the cavitation breakdown behavior. Watanabe et al. [18] used the commercial software FLUENT to numerically simulate the flow around two conventional

* Corresponding author. Tel./fax: +86 10 62789853.
E-mail address: luoxw@mail.tsinghua.edu.cn (X.W. Luo).

0017-9310/$ - see front matter © 2012 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.ijheatmasstransfer.2012.06.065

propellers for non-cavitating and cavitating operating conditions based on the full cavitation model proposed by Singhal et al. [19]. Their predicted cavity shape and pressure fluctuations on the blade surfaces were in good agreement with the measurements. Rhee et al. [20] analyzed cavitating flow around a marine propeller with the predicted cavity shapes on the propeller blade and the thrust breakdown which agreed well with experimental results. However, the authors pointed out that the tip vortex cavitation was missing due to insufficient grid resolution. Kawamura et al. [21] presented unsteady cavitating simulations of a marine propeller operating in a non-uniform wake. The time dependent growth and collapse of the sheet cavity in the wake was well reproduced, but the tip vortex cavity was not captured. Salvatore et al. [22] compared several results using multiphase flow models from various laboratories predicting the propeller performance and cavitating phenomena in uniform flows and wake flows. The numerical results showed good agreement for non-cavitating steady flows and insufficient agreement in the cavitation behavior for cavitating flows especially for propellers operating in non-uniform flows.

For cavitating flow simulations, 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 RANS models with eddy viscosity turbulence models can not accurately simulate unsteady cavitating flows so they need some modifications [23]. There have also been attempts to predict flow unsteadiness during cavitation using Large Eddy Simulations (LES) which are expected to more accurately predict larger-scale turbulent eddies (such as Wang and Ostoja-Starzewski [24]), but quite significant computational resources are needed. Some hybrid models have been proposed to strike a compromise between RANS and LES [25,26]. The Partially-Averaged Navier–Stokes (PANS) method is a recently proposed bridging model proposed by Girimaji [27]. Recently, some trials, such as cavitating turbulent flows past a Clark-Y hydrofoil, were conducted [28].

Inspired by their work, the present paper treats the unsteady cavitating flow using the PANS method together with a mass transfer cavitation model. The applicability of the present method was evaluated for unsteady cavitating flow around a NACA66-mod hydrofoil and unsteady cavitating flow around a highly skewed marine propeller in a non-uniform wake.

2. Numerical method and physical model

2.1. Governing equations for the PANS model

The PANS model described by Girimaji [27] is used with a mixture model to simulate the unsteady cavitating flows.

The vapor/liquid two-phase mixture model assumes the fluid to be homogeneous, so the multiphase fluid components are assumed to share the same velocity and pressure. The continuity and momentum equations for the mixture flow are:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_j)}{\partial x_j} = 0 \tag{1}$$

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x_j} = \rho f_i - \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[(\mu+\mu_t)\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij}\right)\right] \tag{2}$$

where u_i and f_i are the velocity and body force in the i direction, p is the mixture pressure, μ is the laminar viscosity and μ_t is the turbulent viscosity which is closed by the PANS turbulence model. The mixture density, ρ, is defined as:

$$\rho = \alpha_v \rho_v + (1-\alpha_v)\rho_l \tag{3}$$

where α_v is the volume fraction of the vapor component. The subscript v and l refer to the vapor and liquid components.

The modeling challenge in PANS is to determine the closure model as a function of the ratio of the unresolved-to-total kinetic energy, f_k, and the ratio of the unresolved-to-total dissipation, f_e, which are defined as:

$$f_k = \frac{k_u}{k}, \quad f_\varepsilon = \frac{\varepsilon_u}{\varepsilon} \tag{4}$$

where k is the total turbulent kinetic energy, ε is the dissipation rate and the subscript u refers to the unresolved quantities.

The turbulent governing equations in the PANS model [27] developed from the standard k–ε model [29] are:

$$\frac{\partial(\rho k_u)}{\partial t} + \frac{\partial(\rho u_j k_u)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_{ku}}\right)\frac{\partial k_u}{\partial x_j}\right] + P_{ku} - \rho\varepsilon_u \tag{5}$$

$$\frac{\partial(\rho \varepsilon_u)}{\partial t} + \frac{\partial(\rho u_j \varepsilon_u)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_{\varepsilon u}}\right)\frac{\partial \varepsilon_u}{\partial x_j}\right] + C_{\varepsilon 1}P_{ku}\frac{\varepsilon_u}{k_u} - C^*_{\varepsilon 2}\rho\frac{\varepsilon_u^2}{k_u} \tag{6}$$

where P_{ku} in Eqs. 5 and 6 is the unresolved scale production term. The unresolved kinetic energy, the dissipation Prandtl numbers and $C^*_{\varepsilon 2}$ are given by:

$$\sigma_{ku} = \sigma_k \frac{f_k^2}{f_\varepsilon}, \quad \sigma_{\varepsilon u} = \sigma_\varepsilon \frac{f_k^2}{f_\varepsilon} \tag{7}$$

$$C^*_{\varepsilon 2} = C_{\varepsilon 1} + \frac{f_k}{f_\varepsilon}(C_{\varepsilon 2} - C_{\varepsilon 1}) \tag{8}$$

where $C_{\varepsilon 1} = 1.44$, $C_{\varepsilon 2} = 1.92$, $\sigma_k = 1.0$ and $\sigma_\varepsilon = 1.3$.

A smaller f_k gives a finer filter. Girimaji [27] suggested that the PANS equations are identical to the RANS equations, but with different model coefficients, which enables the PANS model to be easily implemented into computational fluid dynamics (CFD) codes without any significant changes. Only the model coefficients in Eqs. 7 and 8 need to be modified to implement the PANS turbulence model.

2.2. Cavitation model

The cavitation model is based on the assumption that the water and vapor mixture in the cavitating flow is a homogeneous fluid. The cavitation process is governed by the mass transfer equation for the conservation of the vapor volume fraction, α_v:

$$\frac{\partial(\rho_v \alpha_v)}{\partial t} + \frac{\partial(\rho_v \alpha_v u_j)}{\partial x_j} = \dot{m}^+ - \dot{m}^- \tag{9}$$

where ρ is the fluid density and u is the flow velocity. The source terms $\dot{m}^+$ and $\dot{m}^-$ in Eq. (9) represent the evaporation and condensation for the phase change during cavitation and the subscript v denotes the vapor.

According to the Rayleigh–Plesset equation, the growth of a single vapor bubble depends on the pressure difference between the local static pressure, p, and the saturated vapor pressure, p_v. By neglecting the second-order derivative of the bubble radius, which is dominant only during rapid changes in the bubble size, the Rayleigh–Plesset equation can be written as:

$$\frac{dR}{dt} = \sqrt{\frac{2}{3}\frac{|p_v - p|}{\rho_l}} \tag{10}$$

where R is the spherical bubble radius and p is the static pressure. Subscript l denotes the liquid.