Ocean Engineering 168 (2018) 41–59

Contents lists available at ScienceDirect

Ocean Engineering

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

Prediction of near field propeller cavitation noise by viscous CFD with semi-empirical approach and its validation in model and full scale

Nobuaki Sakamoto*, Hikaru Kamiirisa

National Maritime Research Institute (NMRI), 6-38-1, Shinkawa Mitakashi, Tokyo 181-0004, Japan

A R T I C L E I N F O

Keywords:
Marine propeller
Cavitation
Underwater ship radiated noise
Environmental protection
Viscous CFD

A B S T R A C T

Propeller cavitation noise has been a large interest among maritime industries in terms of mariners' comfort and marine environmental protection. The objective of this study is to propose and validate practical computational methods to predict near field propeller cavitation noise by complementary use of viscous CFD and semi-empirical formula. Acoustic analogical method represented by Ffowcs Williams-Hawkings equation is not utilized at all in the present study.

Four different propellers equipped on actual ships are of the interest. Cavitation phenomena around these propellers are reproduced by viscous CFD simulations behind full-scale ship wake. Time series of pressure fluctuations as well as the area of cavitation extent on the blades are logged. The former is directly utilized to quantify the level of tonal noise at near field while the latter is utilized to estimate upper bound of broadband noise together with the Brown's semi-empirical formula.

The proposed methods accurately estimate the tonal noise up to 3rd blade frequency, in the meantime, upper limit of broadband noise is well predicted in comparison to the measurement results. The present method is very practical and thus it can be one of the criteria to estimate propeller cavitation noise in the near field.

1. Introduction

Underwater ship radiated noise (USRN) has been a large interest among maritime industries in terms of mariners' comfort as well as marine environmental protection. Two major components of underwater ship radiated noise are 1) low frequency and periodic machinery noise which is originated from main and auxiliary engines, and 2) tonal and broadband noise which is originated from marine propulsors. Although the former is important, the latter is the most dominant noise source under the normal navigation speed and thus is of interest in the present study. In 1995, International Council for the Exploration of the Sea released the guidelines of underwater ship radiated noise for fishery research vessels (Mitson, 1995). In 2014, International Maritime Organization has released the guidelines for the reduction of underwater noise from commercial shipping to address adverse impacts on marine life (International Maritime Organization (IMO), 2014). The former directly regulates the upper limit of radiated sound pressure level (SPL) while the latter regulates the 1st and 2nd blade rate harmonic amplitudes of stern pressure fluctuation depending on the block coefficient C_B of the ship. European Union countries organized several research projects, e.g. “SILENV” (Badino et al., 2013), “AQUO” (Audoly et al., 2014) and “SONIC” (Bureau VeritasDNV, 2015), to investigate experimental and computational methods for assessing environmental impact of underwater ship radiated noise. More recently, the recommendation has been made by Convention of Biological Diversity Subsidiary Body on Scientific, Technical and Technological Advice (CBD-SBSTTA) on 2016 that the ratifiers should prepare toolkits for assessment of USRN (The Subsidiary Body on Scientific, Technical And Technological Advice, 2016). Since computational methods will be one of the essential factors for such toolkits, the present study focuses on their development.

Numerical estimation of propeller cavitation and resultant noise are usually challenging since the flow field around cavitating propeller contains multiscale flow physics, i.e. phase change of the fluid, turbulence, unsteady loading on the propeller blade and so on. Therefore, the propeller cavitation noise has both tonal and broadband frequencies. To capture such noise characteristics, local pressure fluctuation around noise source needs to be predicted accurately. Modeling dynamics of cavitation bubble around rotating propeller by nonlinear Rayleigh-Plesset equation with thermodynamic interaction (Kamiirisa and Goto, 2005) contributes to estimate pressure fluctuation with high frequency and resultant broadband noise. Yet this method requires assumptions in volumetric change of cavitation bubble in time as well as distribution of bubble radius. The semi-empirical formulae based on model and full

---

* Corresponding author.
E-mail addresses: sakamoto@nmri.go.jp (N. Sakamoto), kamiirisa@nmri.go.jp (H. Kamiirisa).

https://doi.org/10.1016/j.oceaneng.2018.08.061
Received 9 April 2018; Received in revised form 29 August 2018; Accepted 30 August 2018

scale measurements (Brown, 1976, Wittekind, 2014, Wittekind and Schuster, 2016, Bureau VeritasDNV, 2015) are able to provide rough estimation in the upper bound of SPL. This method is practical and suitable at initial design stage of hull and propeller, yet sufficient number of measurement campaigns in model and full scale are necessary to construct database for these formulae which are usually expensive in cost and time. Viscous computational fluid dynamics (CFD) can be one of the alternatives to compensate the disadvantages of former two methods. Unsteady propeller cavitation is solved by Reynolds-averaged Navier Stokes (RaNS) equation together with appropriate cavitation and turbulence models. Far field noise can be calculated by complementary use of acoustic analogical method represented by Ffowcs Williams-Hawkings (FW—H) wave propagation equation and viscous CFD. The FW-H method has widely been accepted in the field of aeroacoustics, see for instance, (Farassat, 2007). According to International Towing Tank Conference (ITTC) Specialist Committee on Hydrodynamic Noise (Specialist Committee on Hydrodynamic Noise, 2017), rapid development is on-going in the application of FW-H method together with viscous CFD to hydroacoustic problem specifically for propeller radiated noise. Recent studies for such investigation can be found in Kellet et al. (2013), Ianiello et al. (2014a), Ianiello et al. (2013), Ianiello et al. (2014b), Lloyd et al. (2015), Li et al. (2015), Bensow and Liefvendahl (2016), Lidtke et al. (2016), and Ianiello (2016). These studies solve flows around non-cavitating and cavitating propellers via viscous CFD simulations. The permeable surface (termed “FW-H surface”) in the propeller vicinity is prepared as a part of computational domain in such a way that the local flows around a propeller are included inside the FW-H surface. Acoustic sources originated from flow physics inside the FW-H surface are modeled as monopole, dipoles and quadrupoles. These acoustic sources are then mapped onto the FW-H surface, and the FW-H wave propagation equation is solved to evaluate pressure fluctuation emitted from the FW-H surface. The pressure waves are evaluated at arbitrary receivers located inside or outside of the computational domain. Received pressure fluctuations in time are subjected to frequency analysis and are converted to SPL. This method is able to carry out local flow analysis around a propeller and investigation of its hydroacoustic wave propagation to the far field at the same time. Yet the FW-H equation cannot take the effect of marine environmental conditions into consideration. Vertical and lateral distribution of profiles of acoustic speed under the water strongly affect to the propagation path of the acoustic wave (Urick, 1982), in the meantime, reflections from free surface and seabed can sometimes not be ignored (Holford, 1981). As the ITTC (Specialist Committee on Hydrodynamic Noise, 2017) stated, numerical predictions for acoustic wave propagation under the water are well established such as normal mode method and parabolic equation method for which marine environmental condition can rigorously be considered as input parameters for simulation (Tappert et al., 1977, Urick, 1983). Another computational method is to separate estimation of SPL and acoustic wave propagation under the water. Dominant frequencies and their SPL originated from a propeller vicinity are estimated by viscous CFD and/or “CFD + semi-empirical” (Specialist Committee on Hydrodynamic Noise, 2017) methods, and these results become inputs for simulations of underwater acoustic wave propagation taking marine environmental effects into consideration. In the meantime, some input parameters to the semi-empirical formulae (Wittekind, 2014) can be extracted from solutions of viscous CFD simulations.

According to these backgrounds, the present study aims to investigate the validity of viscous CFD and CFD + semi-empirical methods to estimate propeller cavitation noise in the near field in model and full scale. The definition of “near field” in this manuscript means that 1) hydrophone is flush mounted on ships' bottom in their stern vicinity, and/or 2) distance of closest point of approach between the propeller equipped on the target hull and the measurement ship with drooping hydrophone is approximately 100 m. Four cases of cavitating propellers are of the interest in the present study. As step-by-step investigation, initial validations are carried out to understand the capability of viscous CFD for non-cavitating propellers operating in uniform flow. Then the propeller cavitation simulations behind the ship wake are carried out to validate cavitation pattern and extent of propeller cavitation in time. Since some measurement data are quite limited, necessary conditions to simulate propeller cavitation are also estimated by viscous CFD. After these validations and preparations, propeller cavitation noise in the near field is directly estimated from viscous CFD; acoustic analogical approach is not utilized at all. Maximum extent of propeller cavitation on the blade surface is extracted from viscous CFD solution and becomes the input for the Brown's semi-empirical formula (Brown, 1976). Estimated frequencies and SPL are validated with the available experimental data in model and full scale. Followed by this introduction, Chapter 2 presents the computational methods of viscous CFD and propeller cavitation noise in the near field. Chapter 3 describes the test cases and simulation design utilized in the present study. Brief descriptions of model and full scale experiments are also explained in this chapter. Chapter 4 provides validation results between computational and experimental results. Chapter 5 summarizes the conclusion and future works.

2. Computational method

2.1. Flow field

The computational results are obtained using general-purpose commercial CFD package STAR-CCM + ver 11.04 (double precision version) (CD-adapco®, 2016). The governing equations are the continuity and unsteady RaNS equations with dimensional forms, and are solved by finite volume method. The effects of gravity and surface tension are not taken into consideration.

Among several turbulence models on the solver, k-omega Shear Stress Transport (k-ω SST) based Detached Eddy Simulation (DES) is selected. The reason to choose DES instead of unsteady RaNS (URaNS) is its better capability to capture detail vortical structure than URaNS at blade tip (Riera et al., 2016). Capturing low pressure region at the core of tip vortex will yield better resolution in tip vortex cavitation (TVC). The DES model utilized in the present study switches RaNS region and Large Eddy Simulation (LES) region according to the flag φ defined as

$$\phi \begin{cases} = 1 \ if \ l_t < C_{DES}\Delta: \ RaNS \ region \\ > 1 \ if \ l_t > C_{DES}\Delta: LES \ region \end{cases} \tag{1}$$

$$l_t = \frac{\sqrt{k}}{\beta^* \omega} \tag{2}$$

where l_t is the turbulent length scale, C_{DES} and β are the model constants (=0.78 and 0.09, respectively), Δ is the largest distance between a target cell center and cell center at neighbor cells (e.g. detached eddy length scale), k is the turbulent kinetic energy, and ω is the turbulent specific dissipation rate. According to Eq. (1) the activation of RaNS region and LES region is strongly affected to the cell arrangement. Although figures are not shown in this manuscript, visualization of ϕ in a computational domain confirms that LES region is activated slightly away from the blades and behind the hub.

To model rotation of a propeller, two methods are utilized. For steady simulations, e.g. uniform flow computations to estimate propeller open water characteristics (POC), moving reference frame (MRF) approach is adopted. For MRF approach, computational domain is separated into MRF region and the outside of MRF region. The governing equations solved inside the MRF region take the effect of centrifugal and Coriolis' force due to angular velocity into consideration as body force terms in the momentum equation, while they are not in the outside of MRF region. The fluxes between MRF region and the outside region is weakly exchanged through cell faces with appropriate coordinate transformations. In the present study, the MRF region is