Ocean Engineering 60 (2013) 68–80

Contents lists available at SciVerse ScienceDirect

Ocean Engineering

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

Submarine propeller computations and application to self-propulsion of DARPA Suboff

Nathan Chase, Pablo M. Carrica *

IIHR-Hydroscience and Engineering, The University of Iowa, Iowa City, IA 52242, USA

ARTICLE INFO

Article history:
Received 8 March 2012
Accepted 9 December 2012
Available online 19 January 2013

Keywords:
Submarine propeller
Vortex pairing
Self-propulsion
Overset grids

ABSTRACT

Simulations of the submarine propeller E1619 using the overset flow solver CFDShip-Iowa V4.5 are presented. Propeller open water curves were obtained for two grids for a wide range of advance coefficients covering high to moderately low loads, and results compared with available experimental data. A verification study was performed for one advance coefficient ($J=0.71$) on four grids and three time step sizes. The study shows that grid refinement has a weak effect on thrust and torque but very strongly affects the wake. The effect of the turbulence model on the wake was evaluated at $J=0.4$ comparing results with RANS, DES, DDES and with no turbulence model, showing that RANS overly dissipates the wake and that in the solution with no turbulence model the tip vortices quickly become unphysically unstable. Tip vortex pairing is observed and described for $J \leq 0.71$, showing multiple vortices merging for higher loads. The wake velocities are compared against experimental data for $J=0.74$, showing good agreement. Self-propulsion computations of the DARPA Suboff generic submarine hull fitted with sail, rudders, stern planes and the E1619 propeller were performed in model scale and the resulting propeller performance analyzed.

1. Introduction

Propeller performance and efficiency are important parameters on any marine vehicle, but especially so on a submarine. A submarine propeller is optimized for noise reduction and is therefore larger and has more blades than a ship propeller. For the same torque, the additional blades may reduce the overall thrust and thus reduce both the performance and efficiency (Felli et al. 2008). Computational fluid dynamics (CFD) provides a cost effective method of testing new propeller designs compared to experimental fluid dynamics (EFD) as experimental prototypes are expensive and time consuming to create. However, CFD is also costly compared to other numerical approaches and must be validated before extensive use for design/test purposes.

Propeller forces have been studied and experimentally tested for over 70 years. Denny (1968) describes the process of obtaining propeller open water performance as it was done in the 60's.Current experimental procedures and corresponding uncertainty analysis are periodically updated by the International Towing Tank Conference (2002). Though submarine data is mostly unavailable, experimental results for the generic submarine propeller INSEAN E1619 were reported by Di Felice et al. (2009), who studied open water performance and the wake of the propeller under various loads. To measure the wake of propellers Pitot tubes, hot-wire, Laser Doppler Velocimetry (LDV) and Particle Image Velocimetry (PIV) techniques have been used. Inoue and Kuroumaru (1984) first investigated the structure and decay of vorticity of an impeller flow field utilizing a slanted single hot-wire. As technology advanced, LDV and PIV became the preferred methods for wake analysis as they allow more efficient data acquisition and easy reconstruction of a flow field cross section(Di Felice et al. 2009; Felli et al. 2011).

Numerical methodologies for studying propeller flows are classified between potential flow and CFD approaches. Potential flow models are easier to use, cheaper, and reliable if only minor viscous effects are expected. CFD is more accurate and can capture the wake and highly transient effects, but it is considerably more expensive. A potential flow low-order 3-D boundary element method was used by Young and Kinnas (2003) to evaluate supercavitating and surface-piercing propellers. The simulation's predicted results compare well with experimental measurements for steady inflow. Fuhs (2005) evaluated PUF-2, PUF-14, MPUF-3A, and PROPCAV solvers with DTMB propellers 4119, 4661, 4990, and 5168 under different conditions. He found that none of these potential flow solvers performed significantly better than the others under noncavitating conditions.

Gatchell et al. (2011) evaluated open water propeller performance using the potential flow Panel code PPB and the Reynolds

* Corresponding author. Tel.: +1 319 335 6381; fax: +1 319 335 5238.
E-mail address: pablo-carrica@uiowa.edu (P.M. Carrica).

0029-8018/$ - see front matter © 2012 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.oceaneng.2012.12.029

averaged Navier–Stokes (RANS) solver FreSCo+. The codes gave similar results at advance coefficient (J) values higher than 1.3 but dramatically under predicted the torque and thrust at lower J's. Watanabe et al. (2003) showed that the RANS solver Fluent version 6.1 predicted thrust and torque coefficients in agreement with the measurements taken from uniform flow. Califano and Steen (2011) found that a RANS simulation on a generic propeller provides satisfactory values when compared to experimental results at low J's but became inaccurate at higher J's. They concluded that the inability of the RANS model to resolve tip vortices was the cause of the discrepancies. Rhee and Joshi (2005)simulated the P5168 propeller utilizing a QUOTE turbulence modelinstead of the Baldwin–Barth model used by Hsiao and Pauley (1999). While Hsiao and Pauley (1999) found strong diffusion in the tip vortices, the work by Rhee and Joshi (2005) found that the thrust and torque values are in good agreement with measured values, but turbulence quantities in the tip vortex region were under-predicted.

Yu-cun and Huai-xin (2010) used Detached–Eddy Simulation (DES) modeling to resolve the unsteady flow field behind a DTMB 4118 propeller.DES allows capture of the tip vortices better than RANS because the turbulent viscosity is reduced where the grid is fine enough to capture large vortices. The DES calculated open water curve (OWC) and experimental OWC showed great similarity with the trends being virtually identical. Yu-cun and Huai-xin (2010) were also able to calculate vortex structures in the flow and accurately predict the pressure distribution at different blade sections. Castro et al. (2011) simulated the KP 505 propeller using DES to obtain the OWC and determine the self-propulsion point for the KRISO container ship KCS. In that case, the OWC was measured at model-scale but was simulated at full-scale. Despite the differences in propeller size, the OWCs indicated very good agreement between the simulated and experimental results.

Self-propulsion simulations usually utilize a body force model of the propeller rather than the direct modeling method used by Carrica et al. (2010, 2011). Direct modeling is less commonly used because the propeller rotates much faster than the ship advances, necessitating a small time step to provide sufficient resolution of the propeller flow. While a body force model provides acceptable results for hull analysis, a discretized propeller is needed to fully investigate the interaction of wake and appendages with the propeller. Liefvendahl et al. (2010) and Liefvendahl and Tröeng (2011) used a Large–Eddy Simulation (LES) method to simulate the E1619 propeller wake in both free stream and under moving conditions using the DARPA Suboff submarine hull near self-propulsion, and reported OWC results as well as transient loads on the propeller and individual blades. A discretized propeller was also used by Carrica et al. (2010) for a self-propelled ship, and this technique is applied in this paper to the DARPA Suboff submarine propelled by the E1619 propeller.

In this work the code CFDShip-Iowa V4.5 (Carrica et al. 2007a, 2007b) was used to study the performance of different CFD approaches to obtain the OWC and the vortical structure of the generic submarine propeller INSEAN E1619. Simulations with RANS, DES, Delayed Detached–Eddy Simulation (DDES), and with no turbulence model (NTM) simulations were performed at six different advance coefficients. A validation and verification study with four different grids and three time steps was performed for one advance coefficient (J=0.71) using experimental data from INSEAN. Uncertainty analysis was performed utilizing the procedures described in Stern et al. (2001a, 2001b) to estimate the grid, time step, and total errors. The wake profile was further analyzed utilizing a grid with an extra refinement block behind the propeller. Vorticity and isosurface results from this wake grid were compared to the fine grid results, and tip vortex resolution and vortex pairing were evaluated. The propeller was then attached to the DARPA Suboff geometry and the self-propulsion point was found, along with the propeller performance at the aforementioned point.

2. Mathematical and numerical modeling

The simulations were performed with CFDShip-Iowa v4.5 (Carrica et al. 2007a, 2007b), a CFD code with RANS, DES, and DDES capabilities. The RANS and DES approaches are based on a blended $k-\omega/k-\epsilon$ SST turbulence model (Menter 1994). The incompressible Navier–Stokes equations are nondimensionalized using the reference velocity U_0 and a characteristic length L, in this case the boat velocity and length. The mass and momentum conservation equations are:

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

$$\frac{\partial u_i}{\partial t}+\frac{\partial u_i u_j}{\partial x_j}=-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\frac{1}{Re_{\text{eff}}}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\right]+s_i \tag{2}$$

where the dimensionless piezometric pressure is $p=p_{\text{abs}}/\rho U_0^2+2k/3$ and p_{abs} is the absolute pressure. The effective Reynolds number is $1/Re_{\text{eff}}=1/Re+\nu_t$, with the turbulent viscosity ν_t obtained from the turbulence model. The turbulent kinetic energy k and the specific dissipation rate ω are computed from:

$$\frac{\partial k}{\partial t}+\left(u_j-\sigma_k\frac{\partial \nu_t}{\partial x_j}\right)\frac{\partial k}{\partial x_j}-\frac{1}{P_k}\frac{\partial^2 k}{\partial x_j^2}+s_k=0 \tag{3}$$

$$\frac{\partial \omega}{\partial t}+\left(u_j-\sigma_\omega\frac{\partial \nu_t}{\partial x_j}\right)\frac{\partial \omega}{\partial x_j}-\frac{1}{P_\omega}\frac{\partial^2 \omega}{\partial x_j^2}+s_\omega=0 \tag{4}$$

The turbulent viscosity is $\nu_t=k/\omega$ and the Peclet numbers are defined as:

$$P_k=\frac{1}{1/Re+\sigma_k\nu_t},\ P_\omega=\frac{1}{1/Re+\sigma_\omega\nu_t} \tag{5}$$

with the source for K and ω being $s_k=-G+\beta^*\omega k$ and $s_\omega=\omega(\beta^*\omega-\gamma G/k)-2(1-F_1)\sigma_{w2}(\partial k/\partial x_j)(\partial\omega/\partial x_j)/\omega$, respectively.

The regions of massively separated flows can be modeled using $k-\omega/k-\epsilon$ based DES or DDES models. The dissipative term of the k-transport equation is revised as:

$$D^k_{\text{RANS}}=\rho\beta^* k\omega=\frac{\rho k^{3/2}}{l_{k-w}} \tag{6}$$

$$D^k_{\text{DES}}=\rho\frac{k^{3/2}}{\tilde{l}} \tag{7}$$

The length scales are:

$$l_{k-\omega}=k^{1/2}/(\beta^*\omega) \tag{8}$$

$$\tilde{l}=\min(l_{k-\omega},\ C_{\text{DES}}\Delta) \tag{9}$$

where C_{DES}=0.65 and Δ is the local grid spacing. This formulation determines where the LES or RANS models will be applied. A detailed description of the DES and DDES implementation into CFDShip-Iowa is found in Xing et al. (2007) and Xing et al. (2010), respectively.

The self-propelled simulation used a proportional-integral speed controller to alter the propeller RPS to achieve the target speed. The instantaneous RPS is computed as

$$n=Pe_U+I\int_0^t e_U dt \tag{10}$$

where P and I are the proportional and integral constants of the control and the velocity error is defined as $e_U=U_{\text{target}}-U_{\text{boat}}$.