ARTICLE IN PRESS

Ocean Engineering xxx (xxxx) xxx–xxx

Contents lists available at ScienceDirect

Ocean Engineering

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

Numerical investigation of the effects of camber ratio on the hydrodynamic performance of a marine propeller

Nowrouz Mohammad Nouri*, Saber Mohammadi

School of Mechanical Engineering, Iran University of Science and Technology, Narmak, Tehran, Iran

A R T I C L E I N F O

Keywords:
Hydrodynamics
Marine propeller
Camber ratio
Open water

A B S T R A C T

In this research, effects of the camber ratio distribution over the blades of a NACA marine propeller were numerically investigated on its open water hydrodynamic performance characteristic curves. To investigate this problem, Reynolds averaged Navier-Stokes (RANS) equations were solved using computational fluid dynamics (CFD). Results are presented in terms of vorticity, pressure coefficient, hydrodynamic efficiency, thrust and torque coefficients, and hydrodynamic performance characteristic curves. Results show that the location of the maximum efficiency of the propeller is unchanged versus advance coefficient by changing the camber ratio over the blades. It is shown that magnitudes of the efficiency, torque, thrust, vorticity, pressure coefficient, and the width of the efficiency curve are changed non-linearly by changing the camber ratio.

1. Introduction

In the last decade, several researchers have investigated the propellers hydrodynamics using numerical approaches. Watanbe et al. (2003) modeled a propeller in both steady and unsteady conditions using standard k-ω turbulence model. In this research, the average amount of error was about 15% in predicting the produced thrust. Shams et al. (Hayati et al., 2012) investigated effects of the Rake on the hydrodynamic performance of a propeller. They applied a CFD tool based on RANS equations using k-ω turbulence model. This study contained a 15% computational error in computing the axial torque of propellers. Podded propellers were studied by Arikan et al. using SST k-ω turbulence model (Arikan et al., 2012). Peng et al. (2013) investigated effects of turbulence models on predicting hydrodynamic coefficients of a specific marine propeller numerically. They showed that eddy viscosity turbulence models were more accurate than Reynolds-stress models in predicting hydrodynamic coefficients of the propeller. Furthermore, they found that eddy viscosity turbulence models predicted the propellers performance similarly. Shamsi et al. (Shamsi and Ghassemi, 2015) performed some time-accurate analyses over the flow around a podded driver using sliding mesh method. They used a RANS solver for predicting unsteady hydrodynamic forces acting on the propulsion system. Other efforts were also carried out on predicting the performance of propellers using CFD methods. In spite of numerous researches done on predicting the performance of specific propellers or the physics of the surrounding flow, effects of geometrical parameters on the performance of propellers are rarely discussed by researchers. For example, Rake, Skew and number of blades are among geometrical parameters which are discussed by some researchers and by the authors in recent years (Hayati et al., 2012; Abdel-Maksoud et al., 1998; Ghasemi, 2009; Nouri and Mohammadi, 2016).

A propeller is made of several different sections distributed in the radial direction over some blades. Since the magnitude of an airfoil camber ratio is directly related to its effective angle of attack (Blevins, 1984), this parameter has a close relationship with the propellers hydrodynamic performance. This research numerically discusses the relationship between the propellers hydrodynamic performance and its radial camber ratio distribution. In this research, effects of the camber ratio on the propellers hydrodynamic efficiency, pressure distribution (cavitation behaviour), downstream vorticity distribution (noise propagation), thrust and axial torque are discussed in the open water condition. Some open water four-digit NACA-based marine propellers with similar geometrical parameters except for camber ratios were considered and their hydrodynamic characteristics were examined. Four-digit NACA airfols were selected as cross sections of propellers because:

A. They could be simply modeled through some formula.
B. Using similar airfoils in different propellers, one could reveal the effects of a specific geometrical parameter in the propellers hydrodynamic behavior.
C. Four-digit NACA airfoils were approximately similar to applied cross sections in common marine propellers.

---

* Corresponding author.
E-mail addresses: mnouri@iust.ac.ir (N. Mohammad Nouri), smohamadi@iust.ac.ir (S. Mohammadi).

http://dx.doi.org/10.1016/j.oceaneng.2017.06.026
Received 3 February 2017; Received in revised form 14 May 2017; Accepted 11 June 2017
0029-8018/

Please cite this article as: Nouri, N.M., Ocean Engineering (2017), http://dx.doi.org/10.1016/j.oceaneng.2017.06.026

The applied configurations for propellers in this research were considered similar to common un-skewed marine ones. In this way, the obtained results were reliable in the case of common marine propellers.

In this research, the applied computational tool was a CFD code based on Reynolds averaged Navier-Stokes equations (RANS) (Fluent 15.0).

2. Numerical method

In order to analyse the flow field around the propeller, continuity and momentum equations must be solved. Since the flow was assumed to be isothermal, the energy equation was neglected. Furthermore, the flow was assumed to be incompressible. In this way, steady states equations can be stated as follows by the use of Reynolds decomposition and time averaging techniques, without the effects of external forces:

$$\rho\overline{u}_j\frac{\partial\overline{u}_i}{\partial x_j}=\frac{\partial}{\partial x_j}\left[-\overline{P}\delta_{ij}+\mu\left(\frac{\partial\overline{u}_i}{\partial x_j}+\frac{\partial\overline{u}_j}{\partial x_i}\right)-\rho\overline{u_i'u_j'}\right]$$
$$\frac{\partial\overline{u}_i}{\partial x_i}=0 \tag{1}$$

where u_i, P, ρ, and x_i are velocity components, static pressure, density and coordinates, respectively. The overbar index represents time averaged values and the prime index represents fluctuating parts.

Authors proved the validity of the simulations procedure in their previous work in reference (Nouri and Mohammadi, 2016) using following assumptions:

1. The RNG-k-ε turbulence model was used.
2. The Velocity-Pressure coupling method was based on a SIMPLEC algorithm (Semi-Implicit Method for Pressure Linked Equations-Consistent).
3. The PRESTO (Pressure Staggering Option) discretization scheme was used for pressure, because it was more suitable for high-speed rotating flows.
4. The quadratic upstream interpolation for convective kinematics (QUICK) scheme was applied as the discretization scheme of convective terms.
5. The applied method for modeling rotational effects of the propeller was the moving reference frame.

The applied computational domain and its dimensions are depicted in Figs. 1 and 2.

The applied boundary conditions were:

- Fixed Inlet velocity and zero pressure gradient normal to the face of the cylindrical domain upstream the propeller (left side of Fig. 1).
- Fixed pressure and zero velocity gradients downstream the propeller and normal to the face of the cylindrical domain (right side of Fig. 1).
- Fixed velocity and zero pressure gradient on the lateral face of the cylindrical domain.
- No-Slip boundary condition on solid boundaries.

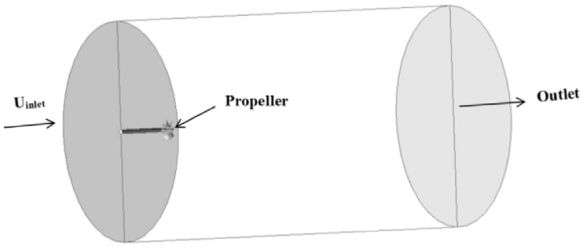

Fig. 1. The Computational domain for CFD simulations.

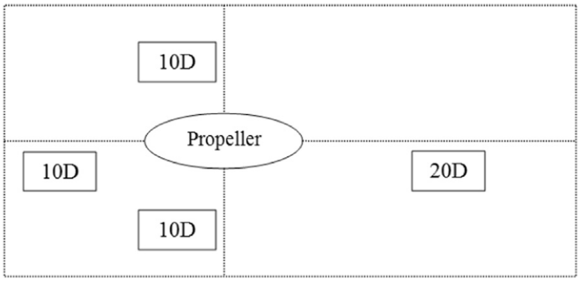

Fig. 2. Dimensions of the domain.

As investigated by the authors in reference (Nouri and Mohammadi, 2016), a mesh sizing of 1.5% of the propellers diameter (D) on the propellers faces and a bias factor of 50D for the far filed was resulted in a good prediction of the hydrodynamic efficiency. The maximum error was found about 8% in the prediction of the thrust coefficient. Therefore, the presented simulation procedure was accurate enough to predict the hydrodynamic performance coefficients of similar propellers.

3. Dimensionless parameters

The applied dimensionless parameters which are used in this research are summarized as follows (Carlton, 2011):

- Thrust Coefficient:

$$K_T=\frac{T}{\rho n^2D^4} \tag{2}$$

- Torque Coefficient:

$$K_Q=\frac{Q}{\rho n^2D^5} \tag{3}$$

- Advance Coefficient:

$$J=\frac{V_a}{nD} \tag{4}$$

- Hydrodynamic Efficiency:

$$n_p=\frac{TV_a}{2\pi nQ} \tag{5}$$

Where T is the propellers thrust in Newton (N), Q is the propellers torque in Newton-meter (N.m), n is the rotational speed of the propeller in revolutions per second (rps), D is the propellers diameter in meter (m), V_a is the advance velocity of the propeller in meters per second (m/sec), and ρ is the fluid density.

4. Case study

The main goal of this research was to study effects of the camber ratio on some marine propellers hydrodynamic behavior. Thus, eight propellers were modeled by using similar geometrical parameters except for the camber ratio. Selected camber ratios placed between 0% and 6% which was a common range for NACA airfoils. The maximum camber ratio of an airfoil is shown by f/c where f is the maximum camber of the airfoil and c is the chord length.

Following assumptions were made in order to prepare geometries of mentioned propellers: