Ocean Engineering 84 (2014) 54–60

Contents lists available at ScienceDirect

Ocean Engineering

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

High-speed planing hull drag reduction using tunnels

Reza Yousefi [a], Rouzbeh Shafaghat [a,*], Mostafa Shakeri [b]

[a] Department of Mechanical Engineering, Babol Noshiravani University of Technology, Babol, Iran
[b] Department of Mechanical Engineering, University of California, Berkeley, CA 94720, USA

ARTICLE INFO

Article history:
Received 29 May 2013
Accepted 29 March 2014

Keywords:
High-speed planning hull
Cougar
Tunnel
FLUENT
Turbulent flow
Volume of fluid (VOF)

ABSTRACT

Forward speed is perhaps the most important parameter in the design of a planing hull. The speed strongly influences the drag and thus the energy supplied by the engine of the ship. Employment of an appropriate drag reduction strategy plays an important role in the design of these hulls. The flow around a Cougar high-speed planing hull was numerically simulated and the results were compared against experiments available in the literature. To reduce the total drag, two tunnels were introduced at the bottom section of the original Cougar hull. The weight and center of gravity of both hulls remained the same. An unstructured mesh was generated in the computational domain around the hull and a Re-Normalization Group $K–\varepsilon$ formulation was used to model the turbulence. To capture the free-surface of the flow around the hull, the volume-of-fluid model was applied. The drag forces of both the original and modified Cougar hulls were obtained for various forward speeds, corresponding to the original hull length-based Froude numbers ranging from 1.00 to 5.62. The results show a 14% reduction in total drag for the modified hull at the forward speed of 60 knot.

1. Introduction

High-speed marine vessels are used for military, recreational, racing, and transportation purposes. Hydrodynamic forces on the high-speed hull, during its forward motion, support most of its weight and thus lift a large portion of the hull out of water. The main forces, supporting the weight of the hull, in the displacement and planing modes are buoyancy and hydrodynamic lift, respectively. The hull in the semi-planing mode, however, is supported by both buoyancy and lift (Faltinsen, 2006).

Clement and Blount (1963) performed extensive experimental tests to obtain the drag of the series 62 model with a range of deadrise angles. Their experimental data on this planing hull became a valuable database for future studies. Although experiments are the most reliable means for modeling the flows around these hulls, they are very costly and time consuming and data is achievable only for limited conditions. The inherent limitations of analytical and experimental techniques have motivated the researchers to use computational fluid dynamics (CFD) methods in recent years. Finite volume method (FVM) has been used to investigate the flow around planing hulls ((Brizzolara and Serra, 2011; Jahanbakhsh et al., 2009; Seif et al., 2009; Senocak and Iaccarino, 2005). The RANS (Reynolds-averaged Navier–Stokes) equations along with the $k–\varepsilon$ turbulence model were used in the aforementioned studies. Also, Ghassemi and Ghiasi (2008) presented a hybrid method to determine hydrodynamic forces for the steady state flow around a planing hull. They used the boundary element method along with the boundary layer theory. Later, Kohansal and Ghassemi presented a computational model for determining the hydrodynamic resistance of a number of planing hulls. Their results showed good agreement with the experiments.

A number of investigations have been conducted to reduce the hydrodynamic drag of the hulls. Vafaei et al. (2010) reduced the drag of a wedge-like hull by optimizing its geometry. Their optimization was based on a genetic algorithm. Another way to reduce the drag of a ship, while maintaining its deck space, is to use a multihull form. The ship′s bow wave energy in these ships is guided under the hull. Fultz (2008) has recently studied the flow around a Pentamaran hull for both single and two-phase flow models to obtain the drag and lift forces of the hull with a fixed waterline. Panahi et al. (2009) have also estimated the drag and trim angle for two hulls. The first hull was a two-dimensional wedge with two degrees-of-freedom. They then analyzed the motion of a planing Catamaran hull. They obtained the drag and trim angle and compared their findings against other existing results and found relatively good agreement.

Subramanian et al. (2007) introduced two tunnels to a planing hull for accommodating propellers and thus minimizing the shaft angle. They also investigated the effects of tunnels on the drag and lift forces of the hull. The FLUENT software was used to implement the FVM for the RANS equations. They used a single phase model and neglected the effects of the free-surface. The numerical results were compared with Savitsky's correlations. They have reported a 7% reduction in total drag due to the tunnels. Care must be used

* Corresponding author. Tel./fax: +98 111 3212 271.
E-mail address: rshafaghat@nit.ac.ir (R. Shafaghat).

http://dx.doi.org/10.1016/j.oceaneng.2014.03.033
0029-8018/

Table 1
Specifications of the Cougar hull form and its 1:10 scale model.

Parameters	Full scale	Model
Length (m)	13.187	1.3187
Beam (m)	2.9	0.29
Height (m)	1.5	0.15
Draft (m)	1.2	0.12
Length of center of gravity (m)	5.67	0.567
Full load weight (kN)	229	0.229
Engine power (Hp)	2 × 820	–

in interpreting this drag reduction as their two ship models (with and without tunnels) had different weights and centers of gravity. Therefore, the reported 7% drag reduction could mainly be due to the 5.3% lower weight of the hull with the tunnels.

In this study, the effects of introducing two tunnels to the Cougar planing hull are investigated. This idea is inspired by the Blade-Runner (ICE Marine Ltd., Surrey, UK) hull form. Geometric characteristics of the Cougar hull are reviewed in Section 2. The fluid governing equations are then presented in Section 3. Computational domain and boundary conditions and mesh generation are described in Sections 4 and 5, respectively. Results and discussions are followed in Section 6. Finally, the conclusions of the current study are presented.

2. Geometric characteristics

The Cougar high-speed planing hull was investigated in the current study. Table 1 presents the geometric characteristics, weight, and speed of the full scale Cougar hull as well as its 1:10 model scale. Two tunnels were introduced at the bottom section of the hull and were positioned symmetrically from the center plane of the hull. The introduction of the two tunnels makes the Cougar similar to a Blade-Runner hull form (ICE Marine, Surrey, UK). Numerical simulations were performed for the 1:10 scale model of the Cougar hull with and without tunnels. For the purpose of comparison, the total weight and the center of gravity were kept the same in both cases. The length of center of gravity (LCG) and center of pressure (LCP) is measured from the transom in this study. Fig. 1 shows the solid models of the original and modified hull. Also Fig. 2 shows the schematic diagram of the Cougar hull.

3. Governing equations

The fluid governing equations are described by the continuity and Navier–Stokes equations

$$\frac{\partial u_i}{\partial t}+\frac{\partial u_i u_j}{\partial x_j}=-\frac{1}{\rho}\frac{\partial p}{\partial x_i}+\nu\frac{\partial^2 u_i}{\partial x_j \partial x_j}+g_i \tag{1}$$

$$\frac{\partial u_i}{\partial x_i}=0 \tag{2}$$

where u, p, and g are the velocity, pressure and gravitational acceleration, respectively. In hydrodynamic analysis of high-speed hulls, the k–ε turbulence model is popular due to its accuracy for simple as well as complex flows with recirculation, streamline curvature and swirl. In the current study, a K–ε formulation was used to model the turbulence in the flow. In the K–ε model, the turbulence characteristics are described based on the turbulent kinetic energy (TKE) and viscous dissipation of turbulent kinetic energy. The fluid governing equations thus take the following forms:

$$\frac{\partial(\rho K)}{\partial t}+\nabla\cdot(\rho U K)=\nabla\cdot\left(\left(\mu+\frac{\mu_t}{\sigma_k}\right)\nabla K\right)+P_K-\rho\varepsilon \tag{3}$$

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\nabla\cdot(\rho U\varepsilon)=\nabla\cdot\left(\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\nabla\varepsilon\right)+\frac{\varepsilon}{K}(C_{\varepsilon 1}(P_K)-C_{\varepsilon 2}\rho\varepsilon) \tag{4}$$

The k–ε model is divided into three types: standard, RNG (Re-Normalization), and realizable. While the standard model is used in flows with a higher Reynolds number, the RNG theory uses a differential equation to account for viscous effects, which become important in flows with a lower Reynolds number. The use of this model depends on the behavior of the flow near the wall. The RNG model has a significant improvement over the standard model, especially where the streamlines are highly curved and thus vortices and circulations exist. In flows with reduced velocity and separation due to reversed pressure gradient, the RNG model performs better than the standard k–ε model. Based on the flow conditions of the current study, the Re-Normalization Group (RNG) K–ε method was used

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\nabla\cdot(\rho U\varepsilon)=\nabla\cdot\left(\left(\mu+\frac{\mu_t}{\sigma_{\varepsilon RNG}}\right)\nabla\varepsilon\right)+\frac{\varepsilon}{K}(C_{\varepsilon 1RNG}(P_K+P_{\varepsilon b})-C_{\varepsilon 2RNG}\rho\varepsilon) \tag{5}$$

where

$$C_{\varepsilon 1RNG}=1.42-f_n$$

$$f_n=\frac{\eta(1-(\eta/4.38))}{(1+\beta_{RNG}\eta^3)}$$

$$\eta=\sqrt{\frac{P_K}{\rho C_{\mu RNG}\varepsilon}}$$

The FLUENT software was used to numerically simulate the fluid flow around the Cougar planing hull. This software is based on finite volume method. The solution algorithm consists of three steps: integration of the fluid governing equations over the control volume, discretization of the equations by replacing approximations for the integral terms, and converting the equations to a set of algebraic equations.

Deformation of the hull is usually neglected and thus the hull was assumed rigid in the current study. To simulate the motion of fluid around the hull, the following steps were taken: solution of the Navier–Stokes and turbulence equations, modeling the free-surface, and investigation of the equilibrium state of the hull. To couple the velocity field and pressure, the SIMPLE model was employed. In this method, the pressure correction equation is solved in a number of iterations, and then the velocity is corrected until the continuity equation is satisfied in the computational domain.

To simulate the air–water flow through the tunnels, the volume of fluid model was applied. In this case, a transport equation is used to compute the volume ratio of the two phases at each time step. In VOF method, the Navier–Stokes and continuity equations are solved for an effective fluid (representing the two-phase flow of water and air) with variable physical properties throughout the computational domain. This fluid takes the property of the liquid water in part of the computational region and air in another region, and a combined property of air/water at the free-surface (Hirt and Nichols, 1981)

$$\rho_{eff}=\alpha\cdot\rho_1+(1-\alpha)\cdot\rho_2 \tag{6}$$

$$\nu_{eff}=\alpha\cdot\nu_1+(1-\alpha)\cdot\nu_2 \tag{7}$$

where $0<\alpha<1$ represents the percent volume occupied by each fluid in the computational domain. When $\alpha=1$, the computational cell is inside fluid 1 and when $\alpha=0$ it is in fluid 2. $0<\alpha<1$