Ocean Engineering 150 (2018) 258–271

Contents lists available at ScienceDirect

Ocean Engineering

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

Investigation of self-propulsion of DARPA Suboff by RANS method

Savas Sezen [a], Ali Dogrul [a,*], Cihad Delen [b], Sakir Bal [b]

[a] Yildiz Technical University, Turkey
[b] Istanbul Technical University, Turkey

ARTICLE INFO

Keywords:
CFD
DARPA Suboff
E1619 propeller
Propeller-hull interaction
RANS
Self-propulsion

ABSTRACT

In this study, the resistance and self-propulsion analyses of the benchmark DARPA Suboff with E1619 propeller have been done using Computational Fluid Dynamics (CFD) method. Single phase analyses have been carried out by assuming 3-D, turbulent, incompressible and steady flow, thus the governing equations (RANS Equations) have been discretized with finite volume method (FVM). First, verification and validation studies have been carried out in order to determine the optimum grid numbers for resistance analyses of bare and appended forms, and then for open water propeller analyses. The numerical studies have been done by taking the propeller-hull interaction into consideration by two methods for different velocities. First, the propeller has been modeled as an actuator disc based on body force method coupled with the experimental open water data. The propeller itself has later been modeled behind the hull with a rotating region. Performance analyses have been carried out at the self-propulsion points determined by these two methods. Self-propulsion characteristics have been estimated using thrust identity method. Finally, a comprehensive investigation of wake has been carried out by comparing the present results with those of other studies and the applicability of CFD on self-propulsion prediction of the underwater vehicles has been discussed.

1. Introduction

One of the main goals of submarine designers and researchers is to estimate the resistance and self-propulsion characteristics for different velocities satisfactorily. For this purpose, CFD methods have been widely used for simulating the flow around the submarines including the interactions between the hull and the propeller. The complexity of the work being done, the variability and the difficulty of environmental conditions is gaining importance in the design phase of the vehicle (Vaz et al., 2010). Hydrodynamic design of submarine propellers has also been an important issue for the designers. A submarine propeller has to be designed and optimized by considering the propeller-hull interaction. Finite volume method based commercial CFD codes solving RANS equations are now widely used for the estimation of propeller performance. It is possible to simulate the propeller flow not only for open water condition but also with modelling the interaction between the propeller and the hull and even the rudder.

In the past, some studies have been made about estimating the hydrodynamic signature of the propeller performance in the presence of surface hulls using numerical and experimental methods. Stern et al. (1994). have made a study in order to validate viscous flow method solving the flow around a propeller in the presence of a hull. The numerical results have been compared with those of an inviscid method. The viscous flow method has shown that the inviscid methods will be abandoned with the advance in viscous methods. Tahara et al. (2006). have later carried out steady flow analyses for the KCS model using two different RANS solvers. The numerical results have been presented for towing and self-propulsion tests and then compared with the available experimental results. Ghassemi et al (Ghassemi and Ghadimi, 2008). have developed a computational approach based on panel method in order to predict the hydrodynamic performance of a propeller-rudder system and an azimuthing podded drive system. The calculations have been made for three propeller models with three steering systems in uniform and non-uniform conditions. The results have been compared with the experimental results with and without rudder. The results of the panel method have been in good agreement with the experiments for propeller-rudder system. Zhang (2010) has investigated the well-known benchmark case KCS model numerically. The viscous flow around the model ship has been simulated by a RANS solver for three different grid sets. Then the viscous flow around KCS model has been investigated with an operating propeller by applying body force and sliding mesh approaches. The results of these approaches have been compared with the experimental ones. It has been found that CFD method is feasible for prediction of propeller-hull interaction. Another numerical study has

* Corresponding author.
E-mail address: adogrul@yildiz.edu.tr (A. Dogrul).

https://doi.org/10.1016/j.oceaneng.2017.12.051
Received 28 July 2017; Received in revised form 21 November 2017; Accepted 24 December 2017

been conducted for resistance, open water propeller and self-propulsion performance prediction for KCS model using a RANS solver by Seo et al. (2010). Local mesh refinements have been employed in order to gain a convenient mesh structure. Sliding mesh technique has been chosen for open water propeller tests. Numerical results have then been compared with the existing available data. In the research of Choi et al. (2010), numerical analyses have been carried out to simulate the flow around eight different commercial ship types for investigation of streamlines on the hull, wave pattern around hull and wake field on the propeller plane. In order to consider the self-propulsion, a potential flow solver has been employed and the propeller characteristics have been implemented into the RANS solver. Uncertainty assessment has been used for viscous resistance with different grid numbers. The paper of Carrica et al. (2010). has presented a method for the self-propulsion calculation of surface ships. The method has been based on controlling the propeller rotation speed to find the self-propulsion point while trying to reach the target Froude number. This approach reduces the necessary computational time significantly by performing only one analysis using the controller. The method has been applied to three different forms: KVLCC1, ONR Tumblehome and KCS. Berger et al. (2011). have focused on the propeller hull interaction with a coupled method. The numerical study has been carried out for KCS. The velocity field gathered from the RANS solver has been given as an input to the potential solver. After the new calculated velocity field has been applied to the RANS solver, the flow has been solved by taking the propeller hull interaction into account. The numerical results of the model propeller have been compared with those of a fully RANS computation. By this approach, the computational time has been decreased significantly and the thrust force has been predicted quickly. A numerical investigation has been carried out using an in-house developed CFD solver based on RANS and DES in another study by Carrica et al. (2011). Self-propulsion of KCS model has been analyzed in head wave and calm water conditions by taking sinkage and trim into account. The numerical results have been compared with the experimental ones (e.g. sinkage, trim and revolution rate of the propeller). Villa et al (Villa and Brizzolara, 2012). have made a numerical simulation of the flow around a ship with self-propulsion analyses by RANS solver. The coupled method has been applied: first solve the viscous flow around KCS model and then compute the performance of KP505 propeller by an unsteady panel method. The research of Win et al. (2013). has focused on the interaction between Series 60 hull and a five-bladed fixed pitch propeller using a CFD code coupled with a quasi-steady blade element theory. The analyses have been carried out with and without propeller and both results have been compared with the available experimental data. The comparison has shown that the developed blade propeller model with blade element theory is in fair agreement with the experiments. Rijpkema et al. (2013). have studied the propeller-hull interaction by simulating the steady viscous flow around KCS model with RANS method and unsteady propeller flow with BEM (Boundary Element Method). The numerical analyses have been carried out by a hybrid method. The coupled RANS-BEM approach has given accurate results for thrust compared with the experimental data. The total resistance has been estimated by a commercial CFD program for DTC Post-Panamax Container Ship in another study of Kinaci et al. (2013). They have also carried out a self-propulsion analysis without taking free surface effect into account. Abbas et al. (2015). have offered a hybrid method in order to estimate the propeller forces behind the ship. A hybrid URANS-LES model has been applied to compute the unsteady loadings on marine propellers behind the KVLCC2 tanker model. The numerical results have been compared with different empirical estimations. It has been found that the hybrid computations overestimate the thrust loading and it can simulate transversal effects in addition to the bilge vortices in the ship stern region in bare hull condition. Ponkratov et al (Ponkratov and Zegos, 2015). have focused on the full scale CFD simulations for self-propulsion analyses. The numerical analyses have been conducted using k-ω SST turbulence model. Moving reference frame (MRF) has been employed in the first stage in order to get quick results. Later rigid body motion (RBM) has been applied with lower time step size in order to obtain the self-propulsion point. In another study of Kinaci et al. (2016), CFD analyses have been carried out for resistance prediction of KRISO Container Ship. Experimental and numerical calculations have also been carried out for a fully submerged body and a validation study has been done.

All above studies have been made for surface types of ships. On the other hand self-propulsion studies have also been carried out for underwater vehicles by considering the propeller-hull interaction. The work of Philips et al. (2008). focused on the coupling of blade element momentum theory with RANS method for an autonomous underwater vehicle. Uncertainty analysis has been carried out for the grid structure. Chase (2012) has studied the self-propulsion performance of the well-known DARPA Suboff. A custom developed CFD solver has been employed for open water analyses at various advance coefficients. The effects of the turbulence models on the results have been investigated. The wake field has been compared with the experimental data for a constant advance coefficient (Chase, 2012). A seven bladed INSEAN E1619 model propeller has been studied in the presence of DARPA Suboff submarine model by Chase et al. (Chase and Carrica, 2013). The numerical analyses have been made by employing Delayed Detached Eddy Simulation (DDES) approach. The results have been compared with different turbulence models using four grids and three time steps for one advance coefficient. The results showed that the present approach is applicable in self-propulsion performance prediction of submarines. Zhang et al (Zhang and Zhang, 2014). have carried out a study involving the interaction between propeller and a submarine hull. The analyses have been made by taking the free surface effects into account. The results showed a good agreement with the experiments. It has been highlighted that the free surface has significant effect on total resistance. Budak et al. have investigated the effect of bow and stern geometries on resistance of bared DARPA Suboff via CFD method (Budak and Beji, 2016). In another study, Delen et al. (Delen et al., 2017) have investigated the hull-propeller interaction of DARPA Suboff vehicle using body force method. Hydrodynamic performance of DARPA Suboff has been estimated both numerically and empirically. The numerical results are more accurate than those of empirical method when compared with the experimental data.

In this study, open water performance of E1619 model propeller and resistance of DARPA Suboff AFF-1 and AFF-8 forms have been predicted using Computational Fluid Dynamics (CFD) method. AFF-1 is the bare form while AFF-8 has the appendages including the sail and rudder fins. Verification and validation studies have been conducted for both open water propeller analyses and resistance analyses to determine the optimum grid number. The self-propulsion point of the submarine has been found, where the propeller thrust is equal to the submarine total resistance. Two techniques have been employed for self-propulsion performance prediction: i) the body force method and ii) the model propeller itself modeled behind the submarine hull. Moving Reference Frame (MRF) method has been used for open water and self-propulsion analyses. The MRF method for the submarine might work, as long as there are no objects in the propeller wake (Ponkratov and Zegos, 2015). The self-propulsion performance has been computed by thrust identity method. A detailed wake survey has also been done numerically and the results have been compared with available experimental and numerical data.

2. Theoretical background

2.1. Mathematical formulation

The governing equations are the continuity equation and the RANS equations for the unsteady, three-dimensional, incompressible flow. The continuity can be given as;

$$\frac{\partial U_i}{\partial x_i} = 0 \tag{1}$$