Ocean Engineering 144 (2017) 35–49

Contents lists available at ScienceDirect

Ocean Engineering

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

Numerical and experimental investigation of propulsion in waves

Sebastian Sigmund, Ould el Moctar*

University of Duisburg-Essen, Institute of Ship Technology and Ocean Engineering, Duisburg, Germany

A R T I C L E I N F O

Keywords:
Added resistance in waves
Propulsion in waves
Computational fluid dynamics
Physical tests
Propulsion characteristics

A B S T R A C T

The effects of head waves on propulsion characteristics of a single and a twin screw ship were investigated based on a Reynolds-averaged Navier-Stokes (RANS) solver and physical tests. Initially, propeller open water characteristics were determined in a homogeneous inflow. Next, computations of the towed models without propeller(s) in calm water and in waves were performed to obtain calm water resistances and waves added resistances. Finally, forces acting on the self-propelled models in calm water and in regular head waves were computed. Using obtained results propulsion characteristics in calm water and in waves were determined. Computations were performed using a RANS based flow-solver coupled with the nonlinear six-degrees-of-freedom equations of motion. When needed, the sliding interface method was used, enabling rotation of the geometrically modeled propeller. All computations were performed on the same numerical grid to keep errors originating from different spatial and temporal discretizations as small as possible. Grid studies were conducted to evaluate discretization errors. Computational results were compared to experimental results obtained from physical model tests. It was shown that the RANS solver is capable of investigating the propulsion behavior of a ship in regular head waves. Fair agreement between numerical and experimental results was obtained.

1. Introduction

Ship speed contractually agreed on by owner and ship yard needs to be verified during sea trials. The environmental conditions for these sea trials usually represent nearly ideal fair weather conditions. To ensure that a ship is able to attain its contractual design speed also under operating conditions, a general experience-based 15% of reserve power, called "sea margin," is generally added to the power required for calm water conditions (ITTC, 2008). An additional ten percent engine margin accounts for aging of the ship's hull (caused by fouling) and the installed machinery. Until now, this procedure has proven to be suitable for practical applications and was seldom questioned. This was brought about by the increased size of modern ships operating at higher service speeds and the associated increased installed power as well as the available long-term experience of these ships. These ships were thus equipped with enough reserve power for adequate maneuverability in severe seaways. However, a disadvantage of this approach is that a ship is optimized for conditions it may hardly ever experience, and the added reserve power is generally considered to be independent of ship size or type, a situation that may lead to underpowered smaller and overpowered larger ships operating in the same sea areas. Due to the economic crisis in 2009 and the introduction of the Energy Efficiency Design Index (EEDI) (MEPC I, 2011) by the International Maritime Organization (IMO) to regulate CO_2 emission related to the transport of loads at sea, fuel consumption became an important issue for ship designers and operators. Consequently, more efficient ships are requested, optimized for their individual operational conditions. Also, questions arose concerning the required power under operational conditions and the consideration of this aspect in the design process.

Added resistance and changing propulsion characteristics under operational conditions are key aspects to identify the added required power. Several authors performed systematic studies of wave added resistance (e.g., Sadat-Hosseini et al., 2013; Kashiwagi et al., 2010; Kim and Kim, 2011; Kim et al., 2012; Liu et al., 2011; Sportelli and Huijsmans, 2012; Ley et al., 2014; el Moctar et al., 2016a; Guo and Steen, 2011; Duan and Li, 2013; Lyu and el Moctar, 2017; Seo et al., 2013, 2014; Turnock et al., 2014; Kuroda et al., 2008). Till now the influence of the seaway on the propulsion characteristics has received little attention. However, experimental procedures for required power in waves have been developed at different institutions (e.g. ITTC 2012, 2014, Kitagawa et al., 2014). Presumably, due to the added resistance in waves, this influence only shifts the operation point of the propeller. However, propulsion is decisively affected by the velocity distribution in the propeller plane, and this distribution in turn is influenced by not only the seaway-induced ship motions, but also the orbital wave particle velocities. Propulsion tests in waves at the propeller's operation point are difficult to conduct because implementing a speed-dependent frictional deduction is difficult. Also, measuring the propeller's wake in

* Corresponding author.
E-mail address: ould.el-moctar@uni-due.de (O. el Moctar).

http://dx.doi.org/10.1016/j.oceaneng.2017.08.016
Received 2 May 2017; Received in revised form 11 July 2017; Accepted 13 August 2017
0029-8018/

a seaway is complex and thus difficult to perform. Techniques based on potential theory may be inappropriate owing to the associated frictionless and irrotational flow assumptions. In contrast, approaches based on solving the Navier-Stokes equations represent a good approximation of the physical flow conditions. To date, such approaches have been used to compute mainly a ship's resistance in calm water, whereby, for the case of calm water propulsion, the geometric modeling of the rotating propeller is usually dispensed with. Instead, so-called body force methods are widely used. Such methods vary from a simple preset distribution of propulsion forces in the propeller plane (e.g., Choi et al., 2010; el Moctar et al., 2001; Winden et al., 2014, Hino et al., 2015) to coupling a boundary element method with a RANS solver (e.g., Cura-Hochbaum and Vogt, 2008; Krasilnikov, 2013). Using body force methods leads to a significant saving of computer time; nevertheless, such methods impose limits in mapping the physical flow and, also, they are subject to uncertainties. Up to now, RANS computations of propulsion based on the geometric modeling of a rotating propeller (using sliding or overset grid techniques) have been performed mainly by neglecting the free surface or under calm water conditions (e.g. Lübke, 2005; Carrica et al., 2010, 2011), and this for a specified propeller rate and a given ship speed. Results from these investigations showed often good agreement to measurements. Yet, numerical studies of propulsion characteristics for ships in waves based on solving the Navier-Stokes equations are hardly found in the literature.

In this study, the propulsion of a cruise ship and an ultra large containership in calm water and in regular waves of different lengths and heights was systematically computed. First, we computed the calm water resistance for different speeds as well as the open water characteristic of the propeller. Next, we obtained the propulsion forces for the investigated ships in calm water. For this, we geometrically modeled the hull, the rotating propeller using the sliding grid technique, and the rudder and determined the propeller thrust and torque. The free surface was taken into account in all computations. Subsequently, for both ships we obtained the added resistance in regular waves of different lengths and heights and, finally, we computed the propulsion of the freely advancing ships in the same regular waves. As done under calm water we geometrically modeled the rotating propeller and the rudder. We conducted our computations at model scale and compared these to model test measurements. Model tests were performed at the Hamburg Ship Model Basin (HSVA) (Valanto and Hong, 2015) and at the Norwegian Marine Technology Research Institute (MARINTEK) (Sprenger et al., 2016; el Moctar et al., 2016b). Various aspects were discussed, such as the influence of waves on propeller forces, propeller efficiency, relative rotative efficiency, hull efficiency, thrust deduction and wake fraction as well as the propulsion efficiency.

2. Numerical methods

Only the main features of the applied numerical technique are described here. For further details, see Ferziger and Peric (2002), Muzaferija and Peric (1999), el Moctar et al. (2016b) and Oberhagemann (2016). A RANS solver based on the finite volume approach was used for numerical computations. The solution domain is subdivided into finite volumes, for which the conservation equations are solved. For an incompressible, isothermal and viscous fluid with density ρ, the governing equations for conservation of mass, $m = \int_V \rho dV$, and momentum, $\vec{I} = m\cdot\vec{v}$, are

$$\frac{\partial}{\partial t}\int_V \rho dV + \int_S \rho(\vec{v}-\vec{v}_s)\cdot\vec{n}dS = 0 \tag{1}$$

$$\frac{\partial}{\partial t}\int_V \rho\vec{v}dV + \int_S \rho\vec{v}(\vec{v}-\vec{v}_s)\cdot\vec{n}dS = \int_S T\cdot\vec{n}dS + \int_V \vec{b}dV \tag{2}$$

In the momentum Eq. (2), the surface and volume forces are included in the stress tensor, T, and in the body force, $\vec{b}$, respectively, where V is the control volume, $\vec{v}$ the fluid velocity, and $\vec{v}_S$ the velocity of the control volume surface, S. The stress tensor, T, includes viscous shear stress and a pressure term, p, multiplied by the unit tensor, I, and reads as follows:

$$T = \mu\{\nabla\vec{v}+\nabla\vec{v}^T\}-pI \tag{3}$$

here μ is the dynamic viscosity. A second order scheme discretizes time derivatives using constant time steps. Second order schemes were used also for the spatial discretization.

The Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) algorithm (Ferziger and Peric, 2002) provides an implicit coupling between pressure and velocity. At each time step, outer iterations iteratively correct pressures and velocities.

To simulate a body floating in waves, a two-phase fluid flow is defined for water and air. This solver implements the interface capturing technique of the volume-of-fluid (VOF) type, which has proven to be suitable for handling complex free surface phenomena (Muzaferija and Peric, 1999). It computes an additional transport equation to obtain the scalar volume fraction, α, between zero and one, to indicate the spatial distribution of each fluid inside each cell. The effective fluid properties are defined as follows:

$$\rho_{eff} = \alpha\rho_{water}+(1-\alpha)\rho_{air} \tag{4}$$

$$\mu_{eff} = \alpha\mu_{water}+(1-\alpha)\mu_{air} \tag{5}$$

where ρ_{water} and ρ_{air} are densities of water and air, respectively, and μ_{water} and μ_{air} are dynamic viscosities of water and air, respectively. The High-Resolution Interface-Capturing (HRIC) scheme discretizes the transport equation for α.

At inlet boundaries, the velocity (ship speed and orbital velocities) and free surface elevation according to linear wave theory are prescribed. In the fluid field, ship speed and orbital velocities and the location of the free surface are initialized. Turbulence is modeled using the k-ω-SST turbulence model (Menter, 1994) with wall functions. To compute ship motions in waves, rigid body equations of motion are implicitly coupled with the RANS equations. For details, see Brunswig and el Moctar (2004) and el Moctar et al. (2016b), (2017). Computed ship motions are realized, based on the mesh morphing method (Oberhagemann and el Moctar, 2007). The sliding interface method allows the propeller to rotate when needed.

3. Test cases and experimental setup

Two modern ships were experimentally investigated, namely, a medium sized twin screw cruise ship and a 14,000 TEU single screw post-panamax containership Duisburg Test Case (DTC) (el Moctar et al., 2010). Table 1 summarizes the ship's main particulars as well as the associated test conditions, and Fig. 1 presents hull sections of the investigated models and their idealized aft bodies equipped with propeller(s), shaft(s), strut, and rudder(s). Table 2 summarizes propeller and rudder parameters. The containership was equipped with a twisted rudder fitted with a Costa bulb.

Open water towing tests and propulsion tests in calm water were performed according to standard procedures (ITTC, 2008). A brief description of these tests and/or the associated computational procedure is given below:

The propeller open water characteristics were determined by running the propeller in undisturbed inflow conditions at different propeller advance ratios, $J = v_a/nD$. For this, propellers of diameter, D, were tested for constant propeller speed, n, whereby the propeller inflow axial velocity, v_a, was varied. Propeller thrust, T, and torque, Q_0, were measured and defined as follows:

Thrust coefficient