Ocean Engineering 136 (2017) 54–66

Contents lists available at ScienceDirect

Ocean Engineering

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

# Effects of a duct before a propeller on propulsion performance

J.S. Go, H.S. Yoon*, J.H. Jung

*Department of Naval Architecture and Ocean Engineering, Pusan National University, 2, Busandaehak-ro 63beon-gil, Geumjeong-gu, Busan 46241, Republic of Korea*

ARTICLE INFO

*Keywords:*
Duct
Propeller
Force coefficients

ABSTRACT

We numerically investigated the effect of a duct on propeller performance. Numerical simulations were performed for a wide range of diameters ($0.7D \le D_D \le 1.0D$) and angles of attack of the duct ($0° \le \theta \le 20°$), where $D$ is the propeller diameter. The duct profile is NACA0015 whose chord length is $0.3D$ and trailing edge is located $0.2D$ upstream from the center of the propeller. The propeller force coefficients presented the same increasing pattern and the rate of the propeller efficiency with the duct to the bare propeller augments with increasing $\theta$, irrelevant to $D_D$. However, the net thrust reflecting the total drag, including the duct, depends on the duct size. Hence, there is a critical value of $\theta$ such that the magnitude of the total drag becomes maximum for a relatively large duct size. A certain size of duct leads two regions of the inflow into the propeller, which is originated by the duct. One is the inner region of the duct where the high-speed flow exists because of the contraction of the duct. Another is the outer region of the duct where the flow decelerates due to the duct wake. The circumferential locality of the axial velocity becomes weaker with increasing $\theta$ due to the accelerated flow along the axial direction. Eventually, lower and higher pressure coefficients on the suction- and pressure-side surfaces, respectively, cover a much wider area than those of the case without the duct, leading to greater thrust and torque.

## 1. Introduction

The power performance of recently designed ships has been improved by using energy saving devices (ESDs). ESDs reduce energy loss during sailing, or partially recover lost energy. These devices can be installed in three zones of a ship's hull: before the propeller, at the propeller station and after the propeller (Carlton, 2012). ESDs before the propeller aim at improving the inflow on the propeller plane. Those aft of the propeller utilize the rotational flow of the propeller slip-stream. The other ESDs use combined advantages of those in front of and after the propeller.

In the present study, we focused on the effect of a duct before the propeller on hydrodynamic forces exerted on the propeller and on the flow around the propeller. Thus, the flow characteristics of the propeller are considered as the base problem of our study. The propeller operates in various environments according to the sea conditions and to environmentally-induced motions (waves, wind and currents). These conditions can trigger energy loss induced by cavitation, vibration and noise, which in turn can amplify the magnitude and the harmonic content of hub loads. (Dubbioso, 2014).

The propulsive loads need to be analyzed in detail in order to improve energy efficiency. In addition, the characteristics of the propeller wake are the crucial factors that cause energy loss. Several researchers have investigated the physical phenomenon with experiments (Di Felice et al., 2004; Lee et al., 2004; Felli et al., 2006, 2011; Paik et al., 2007) and computations (Morgut and Nobile, 2012; Ji et al., 2012; Seol, 2013; Muscari et al., 2013; Dubbioso et al., 2013). Recently, these studies been comprehensively reviewed by Baek et al. (2015) wherein an author of the present paper participated to numerically investigate the effect of the advance ratio on the wake characteristics of a marine propeller in a propeller open water test. Based on our knowledge of propellers, we extended our research to investigate the effect of a duct before the propeller on the hydrodynamic forces exerted on the propeller and on the flow around the propeller. The duct considered is similar to a pre-swirl ESD such as wake equalizing duct (WED).

Wake equalizing ducts were first introduced by Schneekluth (1986). They aim to improve the overall propulsive efficiency of a ship by reducing the amount of separation over the afterbody of the vessel. Wake equalizing ducts achieve this by helping to establish a more uniform inflow into the propeller by accelerating the flow in the upper region of the propeller disc and by attempting to minimize the tangential velocity components in the wake field (Carlton, 2012).

Friesch and Johannsen (1994) performed resistance and self-

* Corresponding author.
*E-mail address:* lesmodel@pusan.ac.kr (H.S. Yoon).

http://dx.doi.org/10.1016/j.oceaneng.2017.03.012
Received 23 June 2016; Received in revised form 1 February 2017; Accepted 10 March 2017

propulsion tests on a single screw ship with and without a WED in a large cavitation tunnel in order to investigate the scale effects of ducts. They found that a wake equalizing duct can result in energy savings at full scale, but that it was difficult to prove a similar effect by model tests at a Froude number equivalent speed.

Korkut (2006) investigated the effect of partial wake equalizing ducts on the powering characteristics of a hull form by using resistance, self-propulsion and flow visualization measurements. Two different hull forms were generated from the original hull form of a vessel in order to optimize the stern flow of the vessel. A partial wake equalizing duct with an appropriate stern form produced less resistance and effective power requirement than those of the original hull form, indicating the duct's positive effect on the powering characteristics.

Celik (2007) numerically calculated the effect of a WED on the propulsion performance of a chemical tanker. Computations were performed for several arrangements of the WED for various ship speeds. He reported that the propeller characteristics and resistance of the ship are slightly affected by the presence of a WED, but additional thrust is produced by the WED. He also found that the maximum gain obtained by using an appropriate WED design is about 10%.

The Mewis duct (Mewis, 2008, 2009) was originally developed within the context of smaller container ships and bulk carriers by a combination of a wake equalizing duct and pre-swirl fins, resulting in a energy savings of 7–9%. The Mewis duct is supported by a number of pre-swirl fins and is positioned eccentrically with respect to the shaft's centerline. The duct aims to bring uniformity to the propeller inflow and reduce the rotational losses in the slipstream of the propeller by the pre-swirl fins.

Recently, various researchers have further investigated the effects of ESDs on propulsive performance (Dang et al., 2012; Yuhai et al., 2013; Feizi et al., 2013; Martinas, 2015). Based on our literature survey and review of the references, the performance of the duct before the propeller as ESDs is naturally evaluated by incorporating ships. Hence, the incoming flow of the duct is strongly dependent on the hull form, especially at the stern. The angle and size of the duct should be tuned based on the incoming flow in order to archive the energy saving formation. Finally, it is difficult to find guidelines for the design of the angle and size of the duct independent of the incoming flow pattern.

In the present study, we imposed a uniform inflow condition without considering the ship to investigate the net effect of a duct on propeller performance. In fact, the inflow condition induced by the hull is crucial to choose the angle and size of the duct. In this regard, we wanted to provide the general information for the design of the duct, regardless of the ship type, and mainly focused on the relation between the incoming flow and the angle and size of the duct to improve our physical insight into the duct design. If knowing this relation, we can efficiently design the duct for any ship hull. Therefore, we employed the uniform flow to consider the general situation regardless of the ship hull.

We considered a wide range of duct diameters ($D_D$) and angles of the attack ($\theta$), and numerically investigated the effects of $D_D$ and $\theta$ on the hydrodynamic forces and efficiency of a marine propeller by carefully analyzing the related flow fields. In general, the ducted propeller was originally developed within the context of smaller container ships and bulk carriers which have speeds less than around 20 knots and thrust coefficients greater than unity (Carlton, 2012). However, the duct has continuously been developed to extend the concept to faster ships and various ship types. For this reason, we aimed at providing the general information for the duct design for readers, regardless of the ship type. Consequently, we believe that our results can provide guidance in establishing a reliable approach to duct modeling.

## 2. Computational details

### 2.1. Governing equations

The present three-dimensional (3-D) problem is governed by the Navier-Stokes equations and the continuity equation. Once the Reynolds-averaging approach for the turbulence modeling is applied, the Navier-Stokes equations can be written in Cartesian tensor form as

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

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] + \frac{\partial}{\partial x_j}(-\rho\overline{u'_i u'_j}) \tag{2}$$

where $x_i$ are Cartesian coordinates, $u_i$ are the corresponding velocity components and $p$, $\rho$ and $\mu$ are the pressure, density and viscosity, respectively. The $k-\omega$SST turbulence model was used to close the Reynolds stress term of $-\rho\overline{u_i' u_j'}$. This turbulence model was successfully used to solve the flow around a propeller in the author's previous study, Baek et al. (2015). The pressure-velocity coupling and the overall solution procedure are based on the semi-implicit method for pressure-linked equation-consistent (SIMPLEC) algorithm. The SIMPLEC second-order scheme was used for pressure and convection terms. A second-order central difference scheme was used for diffusion terms. A convergence criterion of $10^{-6}$ was used in our simulations. Further details of the implementation can be found in the STAR CCM+ manuals (CD-Adapco, 2014).

### 2.2. Computational conditions and boundary conditions

We used the KP505 propeller model, which was considered in Baek et al. (2015), and principal particulars of KP505 are summarized in Table 1. Thus, for the sake of brevity, we do not present the 3-D geometric model and parameters of the KP505, which can be found in Baek et al. (2015).

In the present study, we adopted the sliding mesh technique, which is accomplished by directly rotating the subdomain including the propeller as opposed to moving reference frame (MRF) technique. This sliding mesh technique involves accurate time-dependent simulations where a strong fluid-propeller interaction is considered (Mathur, 1994; Blades and Marcum, 2007). Therefore, the domain was split into a global fixed part and a moving part that was specified by a smaller cylinder enclosing the blades and hub entirely.

A rotational speed of 12 revolutions per second (rps) was considered and the corresponding free stream velocities vary with the advance ratio. However, we considered only one advance ratio (0.7), which is approximately the design advance ratio of the KP505 (Lee et al., 2002).

Fig. 1 shows the computational domain and boundary conditions.

**Table 1**
Principal particulars of KP505.

| KP505 | Principal |
|---|---|
| Scale Ratio | 31.6 |
| Diameter, D (m) | 0.250 |
| Pitch/Diameter mean | 0.950 |
| Ae/Ao | 0.800 |
| Hub Ratio | 0.180 |
| No. of blades | 5 |
| Direction of rotation | Right-hand |
| Section | NACA66 |