Ocean Engineering 36 (2009) 377–385

Contents lists available at ScienceDirect

Ocean Engineering

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

PIV measurements of hull wake behind a container ship model with varying loading condition

Jung Yeop Lee [a], Bu Geun Paik [b], Sang Joon Lee [a,*]

[a] Department of Mechanical Engineering, Pohang University of Science and Technology, San 31, Hyoja-dong, Namgu, Pohang, Kyungbook 790-784, Republic of Korea
[b] Marine Transportation Research Department, Maritime and Ocean Engineering Research Institute, KORDI, Jang-dong 171, Yuseong-gu, Daejeon 305-343, Republic of Korea

ARTICLE INFO

Article history:
Received 18 June 2008
Accepted 8 January 2009
Available online 20 January 2009

Keywords:
Two-frame PIV
Loading condition
Hull wake
Longitudinal vortex
Inflow

ABSTRACT

Flow characteristics of the hull wake behind a container ship model were investigated under different loading conditions (design and ballast loadings) by employing the particle image velocimetry (PIV) technique. Measurements were made at four transverse locations and two longitudinal planes for three Reynolds numbers (Re) ($=U_0Lpp/\nu$, where U_0 is the freestream velocity, Lpp is the length between two perpendiculars of the ship model and ν is the kinematic viscosity) of 5.08×10^5, 7.60×10^5, and 1.01×10^6. It was observed that symmetric, large-scale, longitudinal counter-rotating vortices (with respect to centerline) of nearly the same strength were formed in the near wake. For the ballast-loading condition, the vortices appear at propeller plane below the propeller-boss. The vortex center exhibits a significant upward shift near the propeller-boss as the Reynolds number increase, and as the flow moves downstream. Under the design-loading condition, the vortices first appear at a further downstream location than that for the ballast-loading condition above the propeller-boss. This difference in the flow structure can significantly change the inflow conditions to the propeller blades, such as the streamwise mean velocity profiles and turbulence intensity distributions at the propeller plane. In particular, under the ballast-loading condition, asymmetric inflow may weaken the propulsion and cavitation performance of the marine propeller.

1. Introduction

Recently, the demand for large marine vehicles to transport various products at a relatively cheap cost has greatly increased. As the marine vehicles become larger and faster, the loading on propeller blades increases and the interaction between the flow around a ship and propeller becomes more complicated. Thus, a better understanding of the flow and an accurate analysis of the complex flow around a ship are important for modern ship design. The wake formed behind a ship hull has a large influence on the marine propeller performance. The inflow condition to the marine propeller blades is important especially, since a non-uniform and unstable inflow may cause problems such as vibration and cavitation (Edward, 1988). Cavitation is a phenomenon encountered by highly loaded propellers beyond certain critical revolutions which results in a progressive breakdown of the flow and a consequent loss of thrust. However, before this stage is reached, it manifests itself through noise, vibration, and erosion of the propeller blades, strut, and rudders. Such problems are mainly attributed to the flow characteristics of ship hull wake, especially the inflow velocity profile and angle of attack into the propeller, thereby the detailed investigation on the wake without a propeller is important. Any serious attempt to reduce these problems requires reliable information on the inflow and wake analysis based on the detailed experimental measurements.

Large variations in hull design strongly influence the flow structure in the stern region of vessels and the wake behind the ship. The wake behind a bare hull without a rotating propeller has been investigated experimentally using point-wise measurements such as a hot-wire anemometer (Lee et al., 2003a), pitot tube (Kim et al., 2001), and laser Doppler velocimetry (LDV, Kakugawa et al., 1991). However, these point-wise measurement techniques provide sparse velocity vectors and require substantial time to map the entire flow field.

Particle image velocimetry (PIV) velocity field measurement techniques have been recently used to measure the flow field around a ship (Cotroni et al., 2000; Calcagno et al., 2002; Di Felice et al., 2004; Lee et al., 2004). The velocity field measurement technique has a remarkable advantage in terms of facility time. Lee et al. (2003b) and Paik et al. (2004) investigated the flow characteristics over the stern and in the near-wake regions of a modern container hull form and the complex flow characteristics of the wake behind a marine propeller, using a PIV technique. Although the overall ship hull wake was well described by Lee

---

* Corresponding author. Tel.: +82 54 279 2169; fax: +82 54 279 3199.
E-mail address: sjlee@postech.ac.kr (S.J. Lee).

0029-8018/$ - see front matter © 2009 Elsevier Ltd. All rights reserved.
doi:10.1016/j.oceaneng.2009.01.006

et al. (2003b), all their experiments were carried out only at the design-loading condition. Paik et al. (2007a) visualized the inflow ahead of a rotating propeller attached to a container ship model. In their experiments, the inflow in the upper plane, above the propeller axis, was observed to be quite different from that below the propeller axis. In addition, the spatial distribution of the axial-velocity component of the propeller inflow was asymmetric with respect to the vertical center axis, exhibiting different values of axial velocity on the port and starboard sides.

Most of the previous studies on the flow around a ship hull have been carried out for the design-loading condition. Moreover, the stern region was also optimally designed based on the design-loading condition. When the loading condition of a ship changes, the wake flow structure and its behavior around a ship will change i.e., the spatial distribution of the longitudinal vortices and inflow condition is modified. As a consequence, the performance of the propeller located in the stern region is affected significantly, downgrading the operation of the propeller (Paik et al., 2007a). It has been well known that unsteady cavitation occurs more frequently when the propeller is operated in the ballast-loading condition. Under the ballast-loading condition, the cavity thickness on a blade becomes larger and the area occupied by cavity is getting larger. The increased cavity area and instability in the ballast-loading condition increase the pressure fluctuation nearly twice, compared to the design-loading condition. However, as far as we surveyed, experimental or numerical analysis of wake flow for the case of ballast-loading condition is not readily available, because all the previous studies were carried out at the design-loading condition. Therefore, a detailed analysis of the variation of the flow structure of the ship wake under different loading conditions is essential to understand any potential of undesirable effects such as poor cavitation performance, noise, and vibration as mentioned above.

The main objective of the present study is to investigate the complex flow structure of the bare hull wake at two loading conditions namely, ballast loading and design loading by using a KRISO 3600TEU container ship (hereafter called KCS) model. This study will provide fundamental understandings on the hull wake structure at the ballast-loading condition. In addition, the comparison of the hull wake between the two loading conditions will be helpful in the evaluation of operating performance of a marine propeller according to the loading condition. The present experimental data would be useful for the design of a marine propeller, which is usually optimized for the design-loading condition. Velocity fields at four downstream cross-sections from the propeller-boss plane and two longitudinal planes were obtained using a two-frame PIV technique.

2. Experimental apparatus and method

The experiments were carried out in a circulating water channel (CWC) with a test section of $1.0^W \times 1.0^H \times 4.5^L\,\mathrm{m}^3$. The maximum speed was about 2.2 m/s and the surface-flow accelerator was operated to provide a uniform flow at the test section. The KCS model was scaled down to 1/161.74. The container ship model has a length (Lpp) between two perpendiculars (FP and AP) of 1.422 m, breadth (B) of 0.199 m, draft (T) of 0.0667 m, depth (D) of 0.117 m, and block coefficient (C_B) of 0.65. Lpp was divided into 20 sub-sections (stations) and the locations of FP and AP correspond to the station (St) 20 and 0, respectively. Fig. 1 shows the front and rear views, the body plan of the KCS prototype, and the principal dimensions of the ship model. The blockage ratio when the ship model was placed in the CWC was less than 1.47% which can be considered to be negligible and therefore, no velocity correction was made. At the freestream velocity of 0.6 m/s ($\mathrm{Re} = 7.60 \times 10^5$), the flow speed varied within 6.7% along the depthwise direction and the depth Froude number ($U_0/\sqrt{gH}$, where U_0 is the freestream velocity, g is the acceleration due to gravity, and H is the depth of the channel) varied in the range from 0.192 to 0.202 in the measurement area near the free surface.

The two-frame PIV system consists of a dual-head Nd:YAG laser, a CCD camera, a delay generator, and a frame grabber. The CCD camera has a resolution of 1024×1024 pixels and can capture pairs of particle images separated by short time interval (Δt) using the frame-straddling method. The CCD camera and the pulse laser were synchronized using the delay generator to obtain sequential pairs of particle images. From the captured particle images, velocity fields were evaluated using the cross-correlation PIV algorithm based on fast Fourier transform (FFT). The size of the interrogation window was 48×48 pixels with 50% overlapping without any window or image deformation. The error vectors (outliers) were removed based on *absolute range*, *RMS tolerance*, and *magnitude difference*. Then, the removed vectors were replaced by bi-linear and Gaussian-weighted interpolation methods. Finally, the vector fields were smoothed using a small smoothing kernel (3×3 neighborhood) to reduce the effect of artificial data introduced by smoothing process. The maximum uncertainty in the measurement of time-averaged mean velocities was estimated to be less than 1.5% of the freestream velocity (U_0). To estimate the uncertainty of the PIV measurements, particle images of a quiescent flow were evaluated using the procedure recommended by Raffel et al. (1998). The standard errors for the measured displacement vectors are summarized in Table 1. For PIV measurements, the tracer particles in the flow images must exceed a certain size to extract correct velocity vectors. In the present work, the average particle diameter in the flow images was about three pixels, and hence the measurement uncertainty

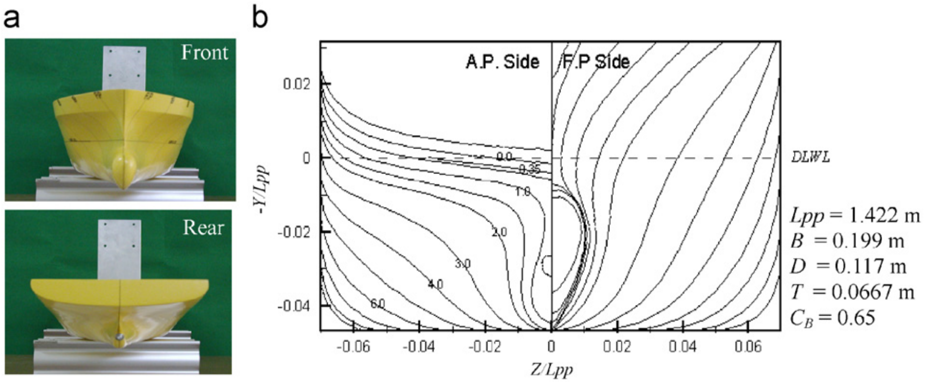

Fig. 1. Front and rear views and body plan of the KCS model and its principal dimensions: (a) front and rear view; (b) body plan.