Ocean Engineering 108 (2015) 181–190

Contents lists available at ScienceDirect

Ocean Engineering

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

An improved wake description by higher order velocity statistical moments for single screw vessel

Andrea Pecoraro[a], Fabio Di Felice[a,*], Mario Felli[a], Francesco Salvatore[a], Michele Viviani[b]

[a] Italian Maritime Research Institute (CNR-INSEAN), Rome, Italy
[b] Genoa University – DITEN – Naval Architecture Unit, Italy

ARTICLE INFO

Article history:
Received 4 April 2014
Accepted 21 July 2015

Keywords:
Ship and propeller wake
LDV
Propeller–hull interaction
Flow separation

ABSTRACT

Unfavorable wake and separated flow from the hull might cause a dramatic decay of the propeller performance in single-screw propelled vessels such as tankers, bulk carriers and containers. For these types of vessels, special attention has to be paid to the design of the stern region, the occurrence of a good flow towards the propeller and rudder being necessary to minimize fuel consumption and avoid excessive vibratory shaft loads and risk for cavitation erosion. The present work deals with the analysis of the propeller inflow in a single-screw chemical tanker vessel affected by massive flow separation in the stern region. Detailed flow measurements by Laser Doppler Velocimetry (LDV) were performed in the propeller region at model scale, in the large Circulating Water Channel of CNR-INSEAN. These tests were undertaken with and without the propeller in order to investigate its effect on the inflow characteristics and the separation mechanisms. In this context, the study also concerned a phase locked analysis of the propeller perturbation at different distances upstream of the propulsor. The study shows the effectiveness of the 3rd order statistical moment (i.e., skewness) for describing the topology of the wake and accurately identifying the portion affected by the detached flow. The skewness coefficient also suggests that a better physical representation of the propeller inflow is provided by the mode value of the velocity, which is the maximum of the probability density function, rather than the mean velocity.

1. Introduction

The hydrodynamic performance of ship is largely dominated by the interaction between the propulsor and the hull boundary layer and wake. This is particularly true for single-screw layouts in which the propeller operates in the middle of the hull wake. The incoming flow is typically characterized by a large region of axial velocity defect downstream of the stern skeg and by transversal velocity components originated by the upward flow due to the curvature of the hull stern lines and by the counter-rotating vortices shed at the two sides of the stern bulb.

The accurate knowledge of the velocity field in the propeller region is fundamental information for the design of a propeller fulfilling imposed hydrodynamic performance requirements. The standard approach is to perform model tests to determine the velocity distribution at the propeller plane by bare hull flow measurements at constant speed (*nominal wake*). During the last years, the application of Computational Fluid Dynamics (CFD) tools to predict the nominal wake of a given hull form and to study the hull–propeller interaction, has also become popular (Huang and Groves, 1980; Choi and Kinnas, 2001; Larsson et al., 2010; Villa et al., 2012; Gaggero et al., 2013). A usual assumption is that the flow in the bare hull stern region is steady in correspondence to the various positions in the propeller disc. A deeper insight into the problem reveals that in some cases the bare hull flow incoming to the propeller plane is characterized by non-negligible fluctuations in time of the intensity of velocity components. This deviation from ideally steady flow conditions is motivated by separation of the hull boundary layer flow upstream of the propeller plane. This behavior can be observed in particular on high C_B single-screw hull forms. The boundary layer separation may be unsteady and this may result in retarded flow spots that are randomly conveyed downstream and determine variations of the intensity and direction of water flowing through the propeller plane. Due to its random nature, these fluctuations in time may also determine asymmetric wake distributions with respect to the centerline that are not expected for single screw hull forms.

While time averaged steady wake flow distributions provide sufficient information for the design of a propeller, matching imposed targets in terms of delivered power and rotational speed, a more comprehensive insight into the non-homogeneous nature of the flow incoming to the propeller plane is necessary to analyze the propeller performance in terms of risk of cavitation, vibratory loads transmitted through the hub to the shaft line and of pressure

* Corresponding Author.
E-mail address: fabio.difelice@cnr.it (F. Di Felice).

http://dx.doi.org/10.1016/j.oceaneng.2015.07.038
0029-8018/

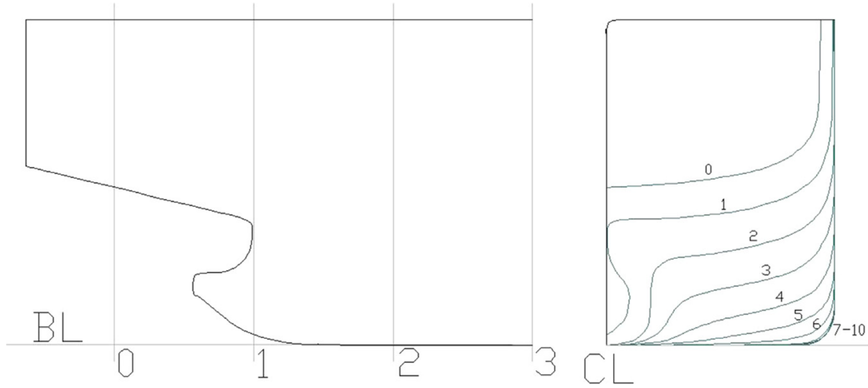

Fig. 1. Longitudinal profile and transverse sections.

pulses radiated to the hull plate and to the flow field. Although radiated noise problems are traditionally related to navy ships operating requirements, these aspects are now becoming important also for merchant ships due to the increased attention to environmental issues leading to more and more stringent requirements for navigation in protected areas (André et al., 2011).

Aim of the present paper is to describe the results of a thorough experimental analysis in which Laser-Doppler Velocimetry (LDV) tests have been performed to characterize hull–propeller hydrodynamic interaction with emphasis on transient flow features. The measurements have been carried out at the CNR-INSEAN Large Circulating Water Channel. As a test case, a 7000 DWT tanker is considered. This test case is the subject of extensive model tests as well as computational studies mostly carried out in the context of the collaborative research project STREAMLINE (2010–2014) funded under the European Seventh Framework Programme, see Lane et al. (2013). Prior to the present study, model tests performed at the Gdansk Ship Design and Research Center (CTO, Poland) revealed a large amount of retarded and reversed flow upstream of the propeller plane likely due to boundary layer separation in the hull stern region. The separated flow was the origin of transient flow incoming to the propeller plane. During the model tests carried out at CTO, flow separation was clearly detected from the results of classical paint-tests with the bare hull model towed at constant speed. This flow behavior was further confirmed by the nominal wake measurements using Pitot tubes that revealed asymmetry of velocity intensity and direction with respect to the hull centerline. Dead water regions upstream of the propeller plane and velocity fluctuations were also predicted by several computational studies, see e.g., Starke and Bosschers (2012), Queutey et al. (2013), Liefvendahl and Bensow (2014).

In order to adequately characterize the transient flow nature of the proposed case study, propeller phase locked measurements are performed. The adopted technique has been already applied to study the wake of twin screw ship propellers, see Felli and Di Felice (2005) and for the first time it is used here to analyze propeller–hull interaction for a single screw ship. The technique allows a deep insight in the analysis of high order statistical moments and a novel approach to characterize the ship wake is proposed.

A detailed description of the propeller hydrodynamic behavior helps to identify the main issues due to the propeller operation in order to seek solutions with a higher efficiency. Furthermore, a detailed representation, especially for separated flows, looking not only to the mean flow characteristics but also to the higher statistical moments, is required for enhancing the accuracy of numerical predictions. The study shows the effectiveness of the 3rd order statistical moment (i.e., skewness) for describing the topology of the wake and accurately identifying the portion affected by the detached flow. The skewness coefficient also suggests that a better physical representation of the propeller inflow is provided by the mode value of the velocity, which is the maximum of the probability density function (PDF), rather than the mean velocity. The availability of reliable and detailed velocity measurements contributes to fulfill this goal providing guidelines for numerical models improvements as well as database for their validation.

This paper is organized as follows. The case study and the experimental set-up adopted for the proposed analysis are described in Section 2, whereas results of model tests are presented and discussed in Section 3. Conclusions and suggestions for future work are summarized in Section 4.

Table 1
Main propeller parameters.

Propeller characteristics (model scale)	
Diameter, D_P (mm)	233,33
Number of blades, Z	4 (fixed pitch)
Rotation	Right handed
Nominal pitch ratio ($P/D_{0.7R}$)	1.0
Skew angle (deg)	13.0
Rake angle (deg)	3.0
Hub diameter ratio, D_H/D_P (at prop. disk)	0.168

2. Experimental setup

2.1. Ship and propeller model

The ship under investigation is a typical chemical tanker and the aft body geometry is shown in Fig. 1. The scale of the tanker ship model used in the present experiment is 1:16.5, with main dimensions (length, breadth and draft) as follows: L_M=5.696 m, B_M=0.934 m, T_M=0.363 m. The test was carried out at 1.77 m/s, (Fr=0.23, based on L_M) and at a propeller angular velocity n= 8.92 rps corresponding to a full scale ship speed of 14 kn. The four-blade, fixed-pitch, model propeller with diameter D=233.33 mm was installed on a shaft closed into a skeg nacelle. The main characteristics of the propeller are reported in Table 1 and a 3D view can be seen in Fig. 2.

2.2. Facility

Measurements were carried out at the CNR-INSEAN Circulating Water Channel, a free surface cavitation channel with a test section (10 m long by 3.6 m wide by 2.25 m deep) that allows one to perform wake surveys for ship models with maximum