Ocean Engineering 121 (2016) 98–111

Contents lists available at ScienceDirect

Ocean Engineering

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

Experimental investigation of airflow over the helicopter platform of a polar icebreaker

Mostafa Rahimpour, Peter Oshkai *

Department of Mechanical Engineering, University of Victoria, PO Box 1700 STN CSC, Victoria, BC, Canada V8W 2Y2

ARTICLE INFO

Article history:
Received 28 September 2015
Received in revised form
11 May 2016
Accepted 12 May 2016

Keywords:
Ship airwake
Atmospheric boundary layer
Particle image velocimetry
Turbulence metrics

ABSTRACT

The airwake over the helicopter platform of a Canadian Coast Guard (CCG) polar icebreaker was studied experimentally. By application of high-speed particle image velocimetry (PIV) on a 1:522 scaled model of the polar icebreaker, quantitative flow field data were obtained in several vertical and horizontal planes. The investigation compared the effects of two types of inflow conditions: (i) a uniform flow and (ii) a simulated atmospheric boundary layer (ABL) on the flow structure over the helicopter platform of the ship. The incidence angle (α) between the oncoming flow and the orientation of the ship varied between 0° and 330° with the increment of 30°. The unsteadiness of the flow and the turbulent fluctuations were quantified by calculating the components of the Reynolds stress tensor and the turbulence intensity. Higher maximum values of the turbulence intensity were observed in the case of the simulated ABL. For both inflow conditions, the incidence angle of 300° corresponded to the highest turbulence levels over the helicopter platform.

1. Introduction

A combination of ship motion and prevailing wind, interacting with vessel's superstructure, creates a region of highly disturbed flow, which is referred to as ship airwake. The airwake is produced as a result of bluff body separation. It contains unsteady turbulent flow structures and is characterized by large spatial and temporal velocity gradients. Such significant increase in the turbulence levels, as well as the escalated temporal and spatial gradients of wind velocities, compared to the undisturbed flow conditions, are known to interfere with maneuverability of helicopters, which can result in loss of control during operation from ship-based platforms (Healy, 1987). In compensating for the large-scale disturbances that are present in this complex and unsteady flow, the pilot may be forced to fly the aircraft dangerously close to the superstructure or the helideck, or unexpected gusts may carry the helicopter away from the vessel (Johns and Healy, 1989). Due to significant challenges for the pilots during the launch and the landing, an in-depth understanding of the ship airwakes, particularly in the vicinity of the helicopter platform, is crucial to minimize the risks involved in helicopter operation.

In analysis of complex flow phenomena, such as 3D boundary layer separation and wake, experimental investigations are the main approach for providing an insight into the flow physics. The full-scale field measurements are often not suitable for a comprehensive study due to the lack of repeatable flow conditions. Additionally, in the full-scale field tests, in order to sample a highly non-uniform flow field, only few strategically placed anemometers can be used, which results in low spatial resolution. In contrast, investigations on scaled models provide repeatable flow condition and yield results with significantly higher spatial resolution. In an early study, in 1987, the prospects for investigation of helicopter/ship interface were discussed by Healy (1987). The author commented that a ship airwake was highly complex, and its dynamics was virtually unknown at the time. He identified the oncoming airflow and the ship motion as the important parameters in investigation of this phenomenon.

Another important parameter that determines the structure of the ship airwake is the yaw angle of the ship with respect to the oncoming airflow. Johns and Healey (1989) performed a flow-visualization study on the 1:140 scaled model of the USN DD-963 Class Destroyer in a boundary layer wind-tunnel using a smoke generator. Through this study, the effects of yaw angles of 30°, 15°, 0°, 330° and 345° on the airwake were investigated. It was observed that, in general, the level of turbulence on the flight deck was higher when the relative wind was on the starboard side. In addition, it was shown that the yaw angle of 330° corresponded with the highest turbulence over the center portion. Rhoades (1992) investigated the ship airwake using hot-wire anemometry on the 1:165 scaled model of another vessel. The authors indicated that the turbulence levels and the size of the recirculation zones over the deck of that ship increased as the ship was yawed. The

* Corresponding author.
E-mail address: poshkai@uvic.ca (P. Oshkai).

http://dx.doi.org/10.1016/j.oceaneng.2016.05.023
0029-8018/

maximum turbulence level was found to correspond to the 50° incidence angle, which began to decrease until it was relatively small in 110°. Tai and Carico (1995) performed numerical simulation of the airwake of a USN DD-963 Class Destroyer in atmospheric winds of 10 and 30 kn at the wind angle of 30°. In this study, a finite volume algorithm was used to for discretization of Reynolds-averaged Navier–Stokes equations. Moreover, the Baldwin–Lomax model with the Degani–Schiff modification was employed for turbulence modeling. As the main feature of the flow, regions of massive flow separation were observed. There was also a good agreement between numerical results for time-averaged velocity components with those of wind tunnel studies. In order to analyze the airwake behind the superstructure of the Halifax-Class Patrol Frigate, Zan et al. (1998) performed investigations on a 1:50 scaled model of the vessel in an atmospheric boundary layer wind tunnel. By application of hot-film anemometers, the flow field in the vicinity of flight deck was studied at 0° and 12° yaw angle with respect to the oncoming airflow. This investigation was complemented with numerical simulations as well as full-scale field measurements using vane-type anemometers. The authors employed CFD-ACE, a commercially available computational fluid dynamics (CFD) flow solver, for numerical simulation. Comparison between the scaled-model tests and the full-scale measurements revealed a good agreement between the results obtained by both methods. There was a qualitative agreement between the experimental results and the results of the numerical simulation, where both predicted the same flow topology. The CFD simulations produced higher gradients in velocity fields, compared to the experimental data. The details of this numerical work and the comparison with the experimental results are reported by Syms (2004).

Through the Technical Co-operation Program (TTCP) (Wilkinson et al., 1998; Reddy et al., 2000), two generic 3D frigate models, Simple Frigate Shape (SFS1) and its successor SFS2, were chosen for wind tunnel experiments and numerical simulations to produce results representative of vessels of the same class. As example of these studies, Toffoletto et al. (2002) used computational fluid dynamics to compare the flow over the flight deck region for SFS1 and SFS2. The RNG k–ε turbulence modeling scheme was used and the results were compared with water tunnel flow visualization data. In was shown that for the case of SFS2 velocity varied more frequently, but magnitude of velocity variations were relatively smaller compared with results for SFS1. Furthermore, the results revealed that the time-averaged velocity fields over the flight deck region were not significantly different. Moreover, comparison with the water channel visualization data showed that the numerical simulation could in fact predict the general structure of the flow field. Polsky (2003) investigated the ship airwake in beam winds using a monotone integrated large eddy simulation (MILES). This numerical study was validated using the time-averaged surface pressure measurements in a wind tunnel for a scaled SFS1. It was showed that grid quality is of great significance, and that a coarse grid would result in considerably over-predicted separation region. The validated code was then employed to investigate the airwake of a US Navy vessel and the results were compared with those of full-scale field measurements. In contrast to the uniform inflow/outflow boundary condition, with the ocean surface modeled as an inviscid wall, it was shown that the application of an atmospheric boundary layer as an inflow/outflow boundary condition and considering the ocean surface as a viscous wall boundary significantly improved the results. Yesilel and Edis (2007) performed steady and unsteady numerical investigation for SFS1 and SFS2 model and presented the results for wind angles of 0°, 45°, 90° and 330°. In this investigation the Reynolds-averaged Navier–Stokes equations were solved by employing several turbulence models. The obtained time-averaged velocity fields were also validated by comparison with wind tunnel data. Syms (2008) simulated the flow around SFS1 and SFS2 in 0°, 45° yaw using the lattice-Boltzmann flow solver PowerFLOW. As the main finding, this investigation showed that mean and unsteady flow field of a frigate-like shape could be captured accurately using a lattice-Boltzmann algorithm. Forrest and Owen (2010) studied the airwake of SFS2 as well as a Royal Navy vessel, at several incidence angles, using detached eddy simulations (DES). The results of this study were compared with wind tunnel data (for SFS2) and field measurement (for Royal Navy frigate). It was concluded that using an ABL velocity profile as a boundary condition improved the agreement between the field data and the numerical results. Using implicit large eddy simulations (ILES), Thornber et al. (2010) studied the airwakes of two different Royal Navy vessels in an ABL at several incidence angles ranging from 0° to 180°. A good agreement between numerical results and those of wind tunnel experiments and full-scale measurements was observed for mean flow, fluctuating quantities and power spectral density. The authors identified the five main features in the ship airwake, corresponding to the turbulent flow structures generated by the masts, the horseshoe-like vortices located between the hull and sea surface, as well as the shear layers shed from the sides and the top of the hangar, the edge of the flight deck itself, and the longitudinal vortices that were located along the ship deck. Bardera Mora (2014a) performed wind tunnel measurements on a model with SFS1 geometry. In this investigation, the oil film technique was used for flow visualization over the surface of the flight deck. The velocity fields were obtained using PIV as well as laser Doppler anemometry (LDA). In addition to time-averaged velocity components, turbulence

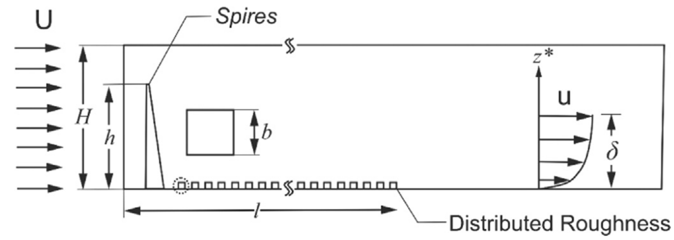

Fig. 1. Schematic of flow conditioning elements.

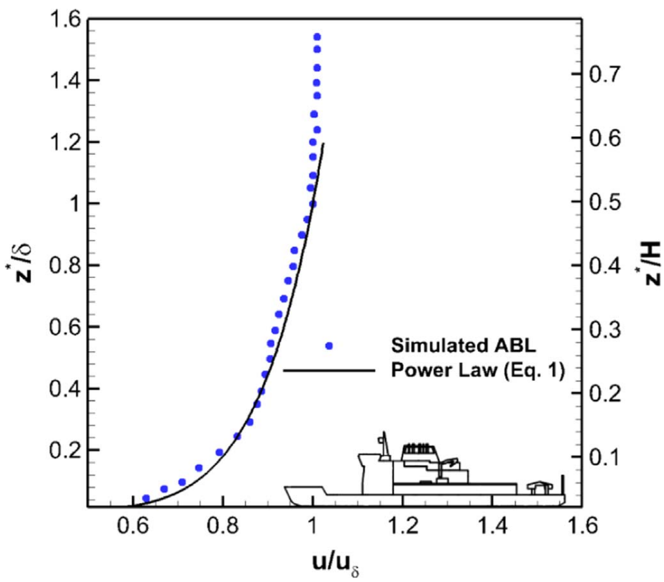

Fig. 2. Distribution of time-averaged streamwise velocity in the simulated ABL.