Ocean Engineering 110 (2015) 2–10

Contents lists available at ScienceDirect

Ocean Engineering

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

Experimental analysis of an air cavity concept applied on a ship hull to improve the hull resistance

Jamie Butterworth [a], Mehmet Atlar [b], Weichao Shi [b,*]

[a] Babcock International Group, UK
[b] School of Marine Science and Technology, Newcastle University, UK

ARTICLE INFO

Article history:
Received 15 March 2015
Accepted 16 October 2015
Available online 31 October 2015

Keywords:
Air cavity
Frictional drag reduction
Energy Efficiency Design
Model test
Stability
Non-linear motion response

ABSTRACT

At the forefront of ship design is the desire to reduce a ship's resistance, thus being the most effective way to reduce operating costs and fulfil the international criteria for reduction in CO_2 emissions. Frictional drag is always proportional to the wetted surface of the vessel and typically accounts for more than 60% of the required propulsive power to overcome; hence the desire to reduce the wetted surface area is an active research interest. An initial full-scale sea trail on a vessel by introducing air as a lubricating medium has indicated 5–20% propulsive energy savings (DK-GROUP, 2010).

Following the report of the fundamental tests with the air cavity concept applied on a flat plate, which was conducted in the Emerson Cavitation Tunnel of Newcastle University (Slyozkin et al., 2014), this paper explores the same concept only this time applied on an existing container ship model to investigate whether it benefits in frictional drag reduction, whilst producing a net energy saving. The middle section of this 2.2 m ship model was modified to accommodate a 0.43×0.09 m^2 air cavity in the bottom of the hull and then various model scale tests have been conducted in the towing tank of Newcastle University. The model experiments produced results ranging from 4% to 16% gross drag reduction. Upon applying scaling factors, it is estimated from the experimental results that around 22% gross energy could be saved in a full-scale application with just a 5% reduction in the wetted surface area.

Further complementary model tests were also conducted to explore the effect of the air cavity on the stability of the model and on the vertical motion responses in a regular head and following wave. While the cavity did not affect the vessel stability the motion response behaviour seemed to be affected non-linearly by the effect of the air cavity.

1. Introduction

Recently, there has been a growing demand due to the ever changing economic, environmental and regulatory challenges that face the shipping industry, namely the implementation of the Energy Efficiency Design Index (EEDI) that has led to an increase in innovative designs to combat some of these challenges. The EEDI *'is a non-prescriptive, performance-based mechanism that leaves the choice of technologies to be used in a specific ship design to the industry'* (IMO, 2014). In short, the EEDI is designed to promote innovative solutions towards meeting the efficiency levels, whilst maintaining the statutory build regulations of the chosen classification society.

One particular area of active research has been the introduction of air as a lubricating medium and is yet to be exploited to its full and commercial potential. There are three specific categories of air injection: Bubble Drag Reduction (BDR) (Kodama et al., 2000); Air Layer Drag Reduction (ALDR) (Elbing et al., 2013, 2008); and Air Cavity (AC) (Foeth, 2008). Each option has had varying amounts and levels of experimental research conducted producing mixed results.

The BDR technique uses the injection of small air bubbles to reduce the skin friction along the boundary layer of the vessel. This technique in its experimental phase produced very promising results, with efficiency savings of up to 80%. However, the technique is heavily reliant on the hull form of the vessel i.e. flat bottomed. It also encountered problems in respect to the volume of bubbles required to reduce the skin friction on the full scale testing. It was stated that total resistance reduction in case of ballast and full load condition was 11% and 6% respectively (Hoang et al., 2009). However, this is a gross saving with little mention as to how much

* Correspondence to: School of Marine Science & Technology, Armstrong Building, Newcastle University, United Kingdom, NE1 7RU. Tel.: +44 191 208 6726.
E-mail address: w.shi6@ncl.ac.uk (W. Shi).

http://dx.doi.org/10.1016/j.oceaneng.2015.10.022
0029-8018/

energy is required to generate sufficient volume of bubbles. Much of the continued research has been on methods for retaining the bubbles on the underside of the vessel to further optimise this technique. In more recent times this technique has been developed to reduce the energy required for bubble generation (Kumagai et al., 2015). The development of the Winged Air Induction Pipe (WAIP) allows for the water flow to over the hydrofoil surface; thus a low pressure region is produced on the upper surface of the hydrofoil. The negative pressure allows for a reduced pressure insertion of air.

The ALDR method is similar to the BDR technique, but rather than producing a bubble mix, ALDR creates a thin continuous layer of air along the length of the underside of the hull. The layer of air is created by means of a slot at the forward end on the underside of the vessel. The thin layer of air traverses the vessel before it begins to break up into bubble form. This technique has been more recently advanced by adding a 'cavitator' to help initiate the air film (Zverkhovskyi, 2014). The 'cavitator' is designed to divert the impending flow of water on the underside of the vessel with a small slotted opening behind the 'cavitator' delivering the air supply. It was found that this method was particularly successful as it significantly reduced the amount of air flux required to generate the air film compared to the standard ALDR.

The ACS technique, also known as Partial Cavity Drag Reduction (PCDR) or Air Cavity (AC) is based on a series of openings on the underside of the vessel that are purged with air. Much of the research on this particular aspect has been in an experimental capacity with a particular focus towards how the air cavity is initiated, the interaction with the hull and body of water and how the release of air at the aft end of the cavity imparts on the hull. Experimental research as conducted by Slyozkin et al. (2014) involved a plate with a backwards facing step (BFS) at the forward end of the incoming water flow and was tested in the Emerson Cavitation Tunnel. In that experiment, air was delivered just behind the BFS to generate an air cavity and was then tested at varying flow conditions. The primary aim of this research was to determine the stability of the air cavity at the varying flow speeds, but also the volume of air required to initiate and maintain the cavity, thus the net energy savings. The optimum condition produced a profound result of 26% reduction in resistance. Slyozkin was able to conclude that if the cavity adds little to form drag, requires a low maintenance gas flux, tolerates flow perturbations and is stable over a wide speed range then this technique has great full scale potential in comparison to other air lubrication techniques.

Based on the conclusions by Slyozkin, the design of the cavity was imperative towards the success on assessing whether the introduction of an air cavity has a positive effect in producing a net energy saving. The work carried out by Slyozkin and Makiharju used a flat plate with a BFS testing arrangement (Makiharju et al., 2010; Slyozkin et al., 2014). The BFS allows for the initiation of the air cavity to occur more freely requiring less air pressure and flux. In both testing arrangements the forward edge of the plate has been horizontally neutral with a reasonable length before the BFS to allow for the flow to become uniform when travelling along the plate's surface.

Based on the equations of Ceccio and Makiharju the quantity of air that is required to generate the initial cavity and to then maintain are quite different (Mäkiharju et al., 2012). The equations were derived based on the results of flat plate experimentation and as such will not have a linear correlation between the experiments carried out by Ceccio and Makiharju and the proposed experimentation for this project. It has been observed that more air flux is required to initially generate the air cavity; however, much less is required to sustain the cavity. The Eq. (1) is used to determine the air flux required to maintain the air cavity.

$$Q = W\left(0.00701\ U^2 - 0.0866\ \ U + 0.277\right) \qquad (1)$$

where Q is the air capacity required, m^3; W is the width of air cavity, m; U is the ship speed, m/s.

Within the above framework the main objective of the research study presented in this paper is to further advance the existing research on Air Cavity Drag Reduction based on the principal Author's MSc research (Butterworth, 2014), more specifically the research aims to develop a rudimental cavity form to determine whether air cavity is an efficient method of reducing frictional drag of a model container ship hull form through experimental resistance tests. Furthermore, to determine the optimum ratio of air flux to frictional drag reduction and to evaluate the change in frictional drag of the hull cavity and the hull form. Finally, to identify whether a cavity form has a significant impact on the stability of a vessel and to assess if and to what extent the cavity affects a vessel's sea keeping properties at a rudimentary level.

In order to achieve the above objectives Section 2 of the paper describes the experiments description which includes the model hull, experimental facility, set-up and test matrix. In Section 3 the results from the three sets of experiments are presented and discussed while in Section 4 the resistance test results are extrapolated to full-scale for the assessment of drag reduction benefit of the air cavity for a full-scale container vessel. Finally Section 5 presents the conclusions obtained from the study.

2. Experiments description

2.1. Experimental setup

The emphasis of this experimentation was to determine how the cavity form integrates with the hull form to determine whether there is potential for further development of this particular technique aiding for drag reduction. The chosen method of experimentation was to utilise Newcastle University's towing tank facility. The towing tank is 37 m in length, 3.7 m in width and has a maximum carriage velocity speed of 3 m/s. The towing tank also harbours the potential for wave generation to assess a vessel's seakeeping characteristics. Further recent details of this facility can be found in Atlar (2011).

The scale hull model used in the experiments was an existing 2.2 m container ship model, which had initially been manufactured for the testing of the Inclined Keel Hull concept developed in Newcastle University (Seo, 2010; Seo et al., 2012). As such, this particular model had a relatively slim bulbous bow and a flat-bottomed hull form with no inclination as shown in Fig. 1. This allowed the cavity form to be positioned horizontally with minimal inclination to prevent the flow from disturbing the effectiveness of the cavity form. The model was also appendage free. The main particulars of the vessel are shown in Table 1.

It is the intention that the size of the cavity should be of sufficient size to produce the intended result, but not such that the cavity designs produces form drag. Therefore, the design parameters for the cavity are given Table 2 and shown in Fig. 2 where the air cavity is accommodated on the bottom of the hull.

In accordance with the wetted surface area and the design water line, DWL, it was the intention that the model would be simulated in a ballasted design condition and as such required the following ballasting arrangements (Table 3). Note the Gifford dynamometer of the towing carriage to tow the hull model in the tank imposes a mass of 5.5 kg on the centre of buoyancy on the vessel.

According to the requirements of this experiment, a constant and regulated air supply was to be delivered to the model throughout each of the proposed trials. The air was to be delivered by a compressor that had been secured onto the carriage of the