Ocean Engineering 140 (2017) 233–243

Contents lists available at ScienceDirect

Ocean Engineering

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

An experimental comparison between different artificial air cavity designs for a planing hull

Filippo Cucinotta*, Eugenio Guglielmino, Felice Sfravara

Department of Engineering, University of Messina, Contrada Di Dio (S. Agata), 98166 Messina, Italy

ARTICLE INFO

Keywords:
High speed craft
Air cavity ship
Hull design
Ship resistance
Model tests

ABSTRACT

For several years it has been studied how to obtain resistance reduction by means of air injection under the hull. The most studied applications are essentially slow hulls; however, significant results in planing hulls can be achieved. Unfortunately, for this kind of crafts, there are very few experimental data. This study has been performed to compare different cavity shapes, obtained by modifying a mother-hull of a high-speed planing yacht. The design has been obtained with the idea to use the natural low pressure under the bottom of high-speed crafts, in order to stabilize an air-layer instead of the traditional air-cushion. The experimental tests were carried out in a towing tank by varying numerous parameters, including the model speed and the flow rate of air. Results and influence of geometrical and physical parameters are discussed.

1. Introduction

1.1. Towing tank tests and methods for resistance reduction

In the last three decades, researchers in naval architecture have developed new methodologies for the reduction of drag in ships. Since 1452 Leonardo da Vinci carried out tests on different models of ships for resistance prediction, but only in 1955 Froude (1955) proposed a law whose idea was to divide the resistance in two components: frictional resistance and wave resistance. A fast method for predicting the resistance of the ship is fundamental for exploring different designs of hull form in order to reduce the drag. A method used for the prediction of resistance in order to optimize the hull forms is the Holtrop-Mennen method (Holtrop, 1984). It is an empirical method that uses the principal dimension of the ship for resistance prediction; it is usually implemented in CAD software for ship design. In the last years, the increase of computers' power allowed to use new analytical or numerical methods, a brief summary is proposed by Birk and Harries (2003). The more reliable method to predict the resistance of ship is the towing tank test, but it is time-consuming and very expensive. Towing tank tests are used only for the final decision between the best candidates, defined by empirical or numerical methods, or for new ship design. The division of resistance in two components allowed improving the hydrodynamic performance of a ship decreasing the frictional resistance and the wave resistance. Initially, researchers were focused on the reduction of the pressure component of resistance, thanks to the development of potential flow method (Dawson, 1977). Jensen developed a method for the numerical determination of the potential flow around a ship moving steadily on the free surface of an ideal fluid (Jensen, 1994). In the years, the potential flow method has been improved and it was used for optimization procedure (Dejhalla et al., 2001). Little improvements have been achieved to reduce the viscous component of resistance. Many authors proposed different concepts for reducing the viscous part of resistance but Russian scientists performed the first important step in the ship industry.

1.2. Frictional resistance reduction by means of air cavities

The initial theoretical modeling for air cavities under plate was developed by Butuzov (1968). This idea was transferred to ships with a full-scale trials of a boat with an air cavity (Butuzov et al., 1988). An alternative to the air-cavity is the microbubbles method. The pioneer in this field were McCormick and Bhattacharyya (1973). Successively Kodama proposed an experimental test with circulating water tunnel (Kodama et al., 2002). An interesting device in combination with the use of micro-bubbles application is proposed by Kumagai et al. (2015). A low pressure zone is produced rear an hydrofoil and the injection of the micro-bubbles is conducted in this zone taking in advantage the optimal environmental condition produced by the device. Merkle and Deutsch highlighted that the microbubbles method is ineffective for low velocity range (Merkle and Deutsch, 1989) and Ferrante and Elghobashi showed that, with the increase of Reynolds number, the reduction of frictional resistance decreased (Ferrante and Elghobashi,

---

* Corresponding author.
E-mail addresses: filippo.cucinotta@unime.it (F. Cucinotta), eguglie@unime.it (E. Guglielmino), fsfravara@unime.it (F. Sfravara).

http://dx.doi.org/10.1016/j.oceaneng.2017.05.028
Received 1 December 2016; Received in revised form 8 May 2017; Accepted 20 May 2017

2004). The disadvantages of the technology with microbubble, showed by authors, have pushed the studies in the direction of air-cavity. A brief review of all technologies for friction drag reduction is proposed by Ahmadzadehtalatapeh and Mousavi (2016). Ceccio (2010) proposed different applications and a detailed explanation of the distribution of the air under the hull during the injection. The economic saving is another aspect to take in account, a brief examination is reported by Mäkiharju et al. (2012). Many authors studied the principal characteristics of the cavity in various conditions, experimental and numerical tests have been conducted in various forms and a brief review is proposed here. Matveev shown the principal parameters that influence the cavity of air (Matveev, 2003) with a numerical study for the simplified configuration of a rear-ward-facing step on the lower surface of horizontal wall for choosing the correct position of propulsion and lifting devices. Another important study was about the interaction between waves and cavity, it was solved with a numerical simplified approach by Matveev (2007). A study of air-ventilated cavities under a simplified hull has been undertaken by Matveev (2012), experiment with a 56-cm-long stepped-hull model were carried in an open-surface water channel at flow velocities of 28–86 cm/s. Following the report on flat plate case with air cavity proposed in the Emerson Cavitation Tunnel of Newcastle University by Slyozkin et al. (2014), Butterworth et al. proposed an experimental test on an existing container ship model with a middle section of 2.2 m (Butterworth et al., 2015) and a 0.43 × 0.09 m^2 area for air cavity. The model experiments produced results ranging from 4% to 16% drag reduction. Always on the flat plate, application of submerged superhydrophobic (SHPo) surface is proposed by Lee et al. (2016) with an entrapped gas called plastron, the application is suitable for laminar flow. Jang et al. (2014) conducted experimental tests on the flat plate and successively on the model with also a self-propulsion test equipped with air lubrication system. Amromin (2016) reported an analysis of the interaction between the air cavity and the boundary layer. The scale factor of the air layer drag reduction has to be take in account. Indeed this effect could change significantly the results when they are transferred directly to full scale model. In this terms, an investigation of the scaling factor is reported by Elbing et al. (2013). There is a great interest also in the shipyard industries for this technology, Mitsubishi Heavy Industries developed an air lubrication system and carried out tests in order to verify the efficiency in terms of frictional resistance reduction (Mizokami et al., 2013). There are less experiment tests for planing hulls; an important example is proposed by Matveev (2015) where there is a numerical method for prediction of drag reduction validated with experimental data. The same author developed a platform, with the necessary instruments, for testing different typologies of Air Cavity Ships (ACS) in order to improve and optimize this kind of technology (Matveev et al., 2015). An experimental work with analysis of the total resistance, trim, sinkage and wave pattern for high speed craft with artificial air cavity is proposed by Gokcay et al. (2004). The experimental data are not only important for better understanding the artificial air cavity phenomena but also in order to validate, in this kind of problem, the URANSe (Unsteady Reynolds-Averaged Navier Stokes equation) methods. Validation is crucial to speed up optimization processes of the geometry of cavity and the correspondent shape of the hull. In the last years, the application of numerical methods for the resistance prediction is widely used in ship design, principally for problems of optimization shape in order to reduce the total drag. Techniques that use the URANSe are applied to avoid the towing tank tests and speed up the optimization process. A typical study is the hydrodynamic hull shape optimization as reported by Wilson et al. (2010) for standard ship or the prediction of drag of new type of ships such as catamarans (He et al., 2013). An important aspect is the simulation of the interaction between fluid and rigid body motion of the ship or any floating object. In the first case a typical example is the calculation of the trim and sinkage (Formaggia et al., 2008), in the second case typical examples are the study of offshore wind turbine (Quallen and Xing, 2016) or system such as wave energy converter (Brusca et al., 2015). In case of large deformation caused by fluid-dynamics pressure or forces the use of rigid body motion is not adequate, so new methodologies with the use of direct coupling between mechanical and fluid-dynamics solvers can be used. In this sense, analysis of shape deformation of parachute is a typical example (Takizawa et al., 2015) or the shape deformation of sails (Cella et al., 2017). In the case of ACS, the problem is the multiphase flow with different scales interaction, the big one that concerns the waves' free surface and the small one that concerns the free surface of cavity (Cucinotta et al., 2017b, 2017c). An initial approach with commercial software is reported by Maimun et al. (2016).

Objective of the research study presented in this paper is to improve the knowledge of air-cavity phenomena with a systematic study on different solution for the same type of planing hull. The approach is oriented to find the better design solution for the application of air-cavity on a yacht of 18 m. The chosen method of investigation was to use the Froude method and the towing tank facilities of the University of Naples. Four different scale models have been tested to different velocities and air flow.

2. Nomenclature

Definition	Symbol	Formula	Unit
Overall length	LOA	–	m
Waterline length	LWL	–	m
Projected Chine Length	LP	–	m
Waterline beam	BWL	–	m
Projected maximum beam	BPX	–	m
Projected beam at generic X position	BPC	–	m
Projected beam transom	BPT	–	m
Projected medium beam	BPA	$\frac{AP}{LP}$	m
Deadrise angle	β	–	°
Height of medium buttock line	HLM	–	m
Draft	T	–	m
Displacement	Δ	–	t
Longitudinal center of gravity	x_G	–	m
Wetted surface area	S	–	m^2
Projected area	AP	–	m^2
Velocity of ship	V_S	–	m/s
Velocity of model	V_M	–	m/s
Froude number	Fn		–
Length of ship	L_S	–	m
Length of model	L_M	–	m
Scale	λ	$\frac{L_S}{L_M}$	–
Number of steps	N_{ST}	–	–
Position of the step relative to the transom	L_{ST}	–	m
Area step	S_{ST}	–	m^2
Number of nozzles	N_{IN}	–	–
Dimensions - Basis x Height	B_{IN} x H_{IN}	–	m x m
Area of nozzles	S_{IN}	–	m^2
Dimensions - Basis x Height rails	B_R x H_R	–	m x m
Transversal distance between rails	TD_R	–	m
Number of rails	N_R	–	–
Volumetric flow rate	Q		m^3/s
Flow rate coefficient	CQ	$\frac{Q}{S_{IN} V_M}$	–

In general, the subscript M indicates the Model and the subscript S indicates the ship.