Ocean Engineering 117 (2016) 199–209

Contents lists available at ScienceDirect

Ocean Engineering

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

Scale effects for rudder bulb and rudder thrust fin on propulsive efficiency based on computational fluid dynamics

Shen Hai-long [a], Enock Omweri Obwogi [a,b,*], Su Yu-min [a]

[a] Science and Technology on Underwater Vehicle Laboratory, Harbin Engineering University, Harbin City, PR China
[b] Department of Marine and Maritime Operations, Jomo Kenyatta University of Agriculture & Technology, Nairobi City, Kenya

ARTICLE INFO

Article history:
Received 14 July 2015
Received in revised form
20 November 2015
Accepted 19 March 2016
Available online 31 March 2016

Keywords:
CFD
Energy saving devices
Scale effects
Propulsion efficiency
RANS equations

ABSTRACT

Current researches on energy saving devices (ESDs) have been conducted in model scale after which the corresponding full scale characteristics are determined through extrapolation. Energy gains predicted by model scale have been found to be less than the actual gains in full scale. This is can be attributed to the complexity of the flow at the stern; hence extrapolation of data is prone to an approximate estimation. Recent developments in Computational Fluid Dynamics (CFD) have made the study of these scale effects possible where simulations for both the model scale and ship full scale are carried out. The main objective in this paper is to analyze the scale effects for ESDs on propulsion efficiency based on a numerical approach. Bare hull resistance, open water, and self-propulsion tests were carried out using CFD code to solve RANS equations. A validation study was carried out and a good agreement between the CFD results and experimental data was obtained. The numerical results demonstrate that the Energy gains predicted at model scale are more than twice those predicted at full scale. The study results will hopefully be useful to Ship designers during ESDs and hull design.

© 2016 Elsevier Ltd. All rights reserved.

1. Introduction

The ever increasing fuel costs and the upcoming Energy Efficiency Design Index (EEDI) regulations have made ship-owners, operators and shipyards to focus on the ways of saving energy for ships. American Bureau of Shipping (ABS) (2013) advisory provide useful information on the status and the current state of ship energy efficiency measures. It is observed that various measures to save energy for ships include; reduction of required propulsion power, reduction of required on board equipment power, use of more efficient propulsion and on board equipment and partial substitution of fuel power by renewable energies.

The requirement for energy efficiency in ship design and operation has been continuously increasing over-recent decades. In relation to this, there is a growing interest in Energy Saving Devices (ESDs) that aim to improve the ship propulsive efficiency where two different approaches have been adopted; Pre-rotating and post propeller recovery (Mewis and Deichmann, 2013; Watson, 1998). It is evident that several ESDs have been fitted to operational ships and tried at full scale although there is limited trial data in the public domain.

The rudder bulb was devised to improve the efficiency of a ship's propeller by reducing hub drag by minimizing separation and reducing pressure pulse. Similarly, the rudder thrust fin improves propulsive efficiency by producing thrust in the rotational flow generated by the propeller. As discussed by Liu et al. (2012), the energy saving effect of the combination of rudder bulb and rudder thrust fin is better than that of rudder bulb. According to the International towing tank conference (ITTC) (1999) the combination of rudder bulb (COSTA propulsion bulb) and rudder fin (rudder-appended thrust fins) showed energy gains of 4–14% at model scale and 4–7.4% at full scale trials. Dang et al. (2012) noted that the best achievement in improving efficiency by adding an ESD to an existing vessel is more than 10% according to the model tests however, although full-scale sea trials have shown up to 5% improvement on total propulsive efficiency. With the present high fuel prices these levels of savings are sufficient to justify investment into the rudder bulb and rudder thrust fin devices.

Recent studies show that ship performance prediction has been done by model tests and by computational fluid dynamics. Model tests are however constrained by the huge capital cost requirements, high technical skills, time constrains and a number of physical and environmental parameters that can influence measurements. In addition, ESDs operate in the boundary layer of the vessel determined by viscosity, therefore scale effects are critical, hence model scale performance does not guarantee actual full scale performance (Sayma, 2009). Kawamura et al. (2012)

* Corresponding author at: Science and Technology on Underwater Vehicle Laboratory, Harbin Engineering University, Harbin City, PR China.
E-mail addresses: obwogie@yahoo.com, eobwogi@jkuat.ac.ke (E.O. Obwogi).

http://dx.doi.org/10.1016/j.oceaneng.2016.03.046
0029-8018/© 2016 Elsevier Ltd. All rights reserved.

analyzed propeller boss cap fins (PBCF) for two different propellers at model and full scale Reynolds numbers with two different inflow conditions. They noted that increased Reynolds number and presence of hull wake both positively influence the effects of PBCF. Oh et al. (2010) and Hsin et al. (2010) indicated that that there may be need to update the ITTC 1978 scaling procedures for rudder bulb and CLT propellers particularly for torque. In Heinke and Hellwig-Rieck (2011), discrepancies existed between RANS and ITTC 1978 prediction concerning the scale effects on propulsion factors for wake equalizing duct (WED) and vortex generating fin (VGF). Chen et al. (2010) suggested that scale effects on performance of energy saving devices need to be assessed through CFD and/or full scale trials. In this paper therefore, the scale effects on propulsion efficiency for rudder bulb and rudder thrust fin were analyzed based on a numerical approach.

Computational Fluid Dynamics code STAR CCM+ is used to solve Reynolds-Averaged Navier–Stokes (RANS) equations. In order to determine performance characteristics; bare hull resistance, open water, and self-propulsion simulations are carried. To study the scale effects for ESDs, these simulations were carried out at two model scales (λ=15 and 29.197) and full scale. Pre-processing of geometry was done in GAMBIT after which it was exported to STAR CCM+ for meshing and computation.

From the presented literatures Çelik and Güner (2007), Choi et al. (2010), Gomri (2012), Hirt and Nichols (1981), Li et al. (2006), Mahmood and Huang (2011), Rhee and Joshi (2003), Takayuki et al. (2003), Yong-jie et al. (2011), Yonghe et al. (2013) a moving reference frame (MRF) technique was adopted for the rotating propeller motion whereas VOF approach was used to capture the movement of the interface between the water and air fluid phases enabling the study of free surface effects. For hull resistance and self-propulsion performance predictions; Realizable k-epsilon two-layer, flat-VOF-wave, Eulerian multiphase, segregated flow and implicit unsteady physics models were used to solve RANS. For open water performance prediction SST K-omega physics model was employed. Hydrodynamic efficiency was then determined for two model scales (with scale factors, λ of 29.197 and 15) and at full scale for both the hull with rudder and propeller and hull with rudder, propeller and ESDs.

In relation to the scale effects on propulsion efficiency for ship with rudder bulb and rudder thrust fin, it was noted that the model scale simulations predicted 4.85% gains in terms of propulsive efficiency while full scale simulations indicated efficiency gains by 2.28%. The largest discrepancy was noted in determining the relative-rotative efficiency (η_r). This effect was attributed to the pronounced scale effects in the calculation of torque since η_r is a function of torque. The data obtained is validated by the experimental data from Shanghai Ship and Shipping Research Institute (SSSRI) and a comparison between the power gains predicted by the models and full scale are presented. The study results will hopefully help to understand the scale effects for energy saving devices which would be useful to ship designers.

2. Ship and propeller particulars

In the present work, experimental data used were obtained from the SSSRI towing tank tests for a single screw 35000DWT bulk carrier. A model scale with λ=29.197 was used in the test. The summary of ship and propeller particulars is as shown in Table 1. The ship model was manufactured by SSSRI based on the information on geometrical shape provided by China Ocean Shipping (Group) Company (COSCO). The model was then equipped with the design propeller from 702 Research Institute.

In this study the speed of the ship (*Vs*) will be 14 kn (7.202 m/s) which is equivalent to the design speed. However, for data validation purposes, simulation for resistance prediction in model scale was done at; *Vs*=11 kn, 12 kn, 13 kn, 14 kn and 15 kn (V_m=1.047 m/s, 1.142 m/s, 1.238 m/s, 1.333 m/s and 1.428 m/s respectively).

Table 1
Ship and propeller particulars.

Main particulars	Symbol	Units	Full scale	λ=15 Model	λ=29.197 Model
Length between P. P.	Lpp	m	172.000	11.467	5.891
Breadth	B	m	30.000	2.000	1.028
Draft	T	m	10.100	0.673	0.346
Number of blade	Z		4.000	4.000	4.000
Diameter	D	m	5.720	0.381	0.196

2.1. Computational domain

The computation domain in the RANS solver was represented by a rectangular block extending 2.5*LPP* from the ship midsection to the inlet, 4*LPP* to the outlet, 3*LPP* from the ship C/L to the side, 3*LPP* below and 1.5*LPP* above the undisturbed water line (see Fig. 1a). The coordinate origin was set at the aft perpendicular and still water surface. The inlet, top and bottom boundaries were set as velocity inlet with both magnitude and direction velocity specification. The velocity magnitude was then defined by velocity of flat-VOF-wave field function. The exit boundary was defined as a pressure outlet with a hydrostatic pressure of flat-VOF-wave field function imposed.

Fig. 1b presents the boundaries and dimensions of the computation domain for open water prediction. It consists of two regions; a rotating region around the propeller (where the rotating motion is defined by the rotating reference frame technique) and a static region for the rest of the domain (where fixed motion is applied).The set up for self-propulsion was similar to resistance prediction only that computation was done for full hull due to the presence of the propeller. A rotating region was also set around the propeller where the rotating motion defined by rotating reference frame was imposed.

2.2. Grid generation

A mesh is the discretized representation of the computational domain, which the numerical solvers use to provide a numerical solution. STAR-CCM+ provides several meshing strategies that are suitable for different applications. Structured and unstructured strategies are available for generating volume mesh. Due to complexity of the ship hull and rudder geometries, it is laborious and difficult to generate structured mesh. In the present work therefore, unstructured strategy was used. According to CD-Adapco (2014), for a given number of cells, the trimmed and polyhedral cell type meshes produce more accurate solutions when compared to a tetrahedral mesh. Unlike the polyhedral and tetrahedral models, the trimmer model is not directly dependent on the surface quality of the starting surface and as such is more likely to produce a good quality mesh for most situations.

Trimmed cell model uses OCTREE triangulation to generate a volume mesh. In this triangulation, squares containing the boundaries are recursively subdivided until desired resolution is obtained. Irregular cells (or triangulation) are generated near the surface where square intersect the boundary. This triangulation requires least surface representation and is highly automated. However, it cannot match surface grid and results to low quality grid near surfaces. In order to improve the overall quality of an existing surface and optimize it for the volume mesh models, the surface remesher can be used to re-triangulate the surface. To

Download English Version:

https://daneshyari.com/en/article/1725108

Download Persian Version:

https://daneshyari.com/article/1725108

Daneshyari.com