Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 194 (2017) 128 – 135

Procedia Engineering

www.elsevier.com/locate/procedia

10th International Conference on Marine Technology, MARTEC 2016

Numerical Study on Propeller Performance for a Vessel in restricted water

Mehdi Nakisa [a,c,]*, Fatemeh Behrouzi[a], Adi Maimun[b] , Rahimuddin Samad[d] ,Yasser M. Ahmed[e]

[a]*Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310 (UTM), Johor, Malaysia*
[b]*Marine Technology Centre (MTC), Universiti Teknologi Malaysia (UTM), Johor, Malaysia*
[c]*Department of Mechanical Engineering, Science and Research Branch, Islamic Azad University, Bushehr, Iran*
[d]*Department of Naval Architecture, Engineering Faculty, Hasanuddin University, Makassar, Indonesia*
[e]*Faculty of Engineering, Alexandria University, Egypt*

Abstract

Ship-bank hydrodynamic interaction effect in shallow water on ship manoeuvrability, steering performance and navigation in restricted waters has been a main challenge to safe navigation. The objectives of this research are firstly to investigate the effect of non-symmetric sloped bank and shallow water on hydrodynamic characteristics of propeller for a LNG ship, secondly to determine the flow wake pattern at propeller plane in different water depth and ship-bank distance. The numerical and experimental research methodologies were applied to achieve the objectives. The data contributes to the surface-ship resistance and propulsion model-scale database for computational fluid dynamics validation, as part of an international collaborative project between Malaysia International Shipping Corporation Berhad (MISC) and Hydrodynamic Research Group of MTC in UTM Institute (HRG-MTC-UTM). The tests were conducted in Marine technology Centre (MTC) – Universiti Teknologi Malaysia (UTM) to validate the computational fluid dynamic simulations using ANSYS-CFX 14. The thrust deduction factor decreased due to suction effect in stern region of ship above Also propeller thrust and torque increased. These results can be applied for as a valuable guideline for evaluating the manoeuvring and navigation of the vessel in restricted waters.

Peer-review under responsibility of the organizing committee of the 10th International Conference on Marine Technology.

Keywords: Numerical Modelling; Propeller Performance; Towing Tank Tests; Deep and Shallow Water conditions; Bank Effect.

* Corresponding author. Tel.: + 6075534761.
E-mail address: mnakisa2003@yahoo.com

Peer-review under responsibility of the organizing committee of the 10th International Conference on Marine Technology.
doi:10.1016/j.proeng.2017.08.126

1. Introduction

In restricted waterways, ship operation is affected by boundaries of the channels, ports and harbours such as the bottom, horizontal restrictions due to banks and quay walls. Lataire and Vantorre (2008) examined the different bank geometries to investigate the ship-bank interaction. They discovered that the ship manoeuvring behaviour is considerably affected by waterways restrictions, both in vertical and lateral dimensions, due to close proximity of the ship to the bottom and of the side walls, respectively [1].

Ship-bank interaction effect varies depending on bank shape, water depth, ship speed and ship-bank distance. In Lataire and Vantorre (2008), seven different bank geometries are installed in towing tank to simulate manoeuvres in low water depths in Flanders Hydraulics Research-Ghent University in a towing tank of 88m length x 7m breadth x 0.5m depth. Their fully automated towing tank can execute captive tests up to 35 tests per day [1].

1.1. Experimental survey

The relationship between ship's centre and half draft of bank ($YT/2 - Yship$) and the lateral hydrodynamic force which are shown in Fig. 1 [1]. Lataire et al., 2007 determined the ship-bank interaction using surface piercing and with submerged horizontal platform to find the effect of bank distance on ship behaviour in case of sway force and yaw moment [2]. In [3] and [4], flooded banks and surface piercing banks were studied. It was discovered that sway force and yaw moment were affected significantly in the case of a bank submerged under a low depth of water. Indeed, one of the advantages of a submerged bank is the test accuracy because the waves due to the bank geometry are damped and the reflexed waves take longer time to come in contact with the ship hull and propulsion system as compared to surface piercing.

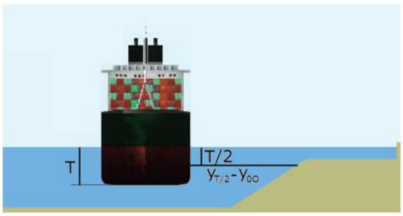

Fig.1. Ship bank distance in half draft [1].

Zhou et al. (2013) investigated hydrodynamic interactions between ship and bank when ship is sailing along a bank in restricted waters [5]. They applied computational fluid dynamics (CFD) to numerically simulate the flow around the ship appended with a rudder whilst navigating parallel to a bank.

Model scale experiments have been conducted by Duffy (2002) to study ship bank interaction in ship navigation. Parameters such as speed of ship, vessel draft, bank height, slope of bank, water depth and ship-bank distance were investigated. Empirical equation was developed from experimental data to estimate sway force and yaw moment produced in the case of surface piercing banks and submerged banks. The tests were performed at Australian Maritime College (AMC) towing tank facility [6].

On the other hand, Duffy (2002) could develop a parameter that superposition yaw moment and sway force produced by port and starboard banks linearly [6]. To produce empirical equations which can help to estimate the sway force and yaw moment, a number of regression analyses were carried out on the experimental results. Research of Duffy (2002) was extended by Maimun et al. (2009) to study the manoeuvring of Liquefied Natural Gas (LNG) ship in shallow water with ship-bank interaction effects [7].

Norrbin (1971, 1974 and 1985) carried out extensive model tests and simulation on manoeuvring in general and effects of bank in particular (Fig. 2). The model tests with a propelled tanker model (L = 5.024 m, B = 0.852 m, T = 0.339 m, CB = 0.821) were conducted to deduce lateral forces and yawing moments acting on a ship due to interaction with a vertical bank based on geometrical ship characteristics, water depth, ship-bank distance and propeller loading. The results showed that the propeller effect initially is amplified if the forward speed increased, up to a certain critical value. The application point of the additional force due to propeller action appeared to move forward with decreasing under keel clearance and increasing speed, and move aft with decreasing bank clearance [3, 4 and 5].

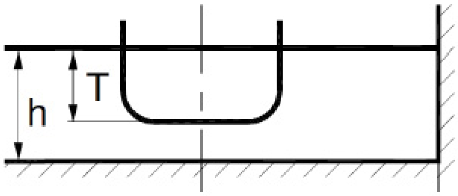

Fig.2. Vertical bank configuration [3, 4 and 5].

1.2. Computational fluid dynamic simulation survey

CFD has indeed made significant progress in replacing expensive model tests and this is proven by the extensive use of viscous flow and potential flow CFD codes in design. Model tests are now only limited to the much later stages in the project. In marine applications, CFD has also been used to simulate the flow around marine bodies such as hulls, propellers and ship appendages. In following parts, literature review is focused on evaluation of propeller working in open and confined water conditions.

Fig. 3 and Table 1 show the suitable computational domain and dimensions for simulations that they used. Hexa-structured meshes and hybrid-unstructured meshes were compared in the study and the turbulence model used in the commercial CFD solver was Shear Stress Transport (SST).

Morgut and Nobile (2012) found that for the propeller in open water propulsion characteristics, the hexa-structured and hybrid-unstructured meshes can guarantee similar levels of accuracy [11]. Nevertheless hybrid-unstructured meshes seemed to exhibit a more diffusive character than hexa-structured meshes, and also they were suited for detailed investigations of the flow field [11, 12, 13, 14 and 15].

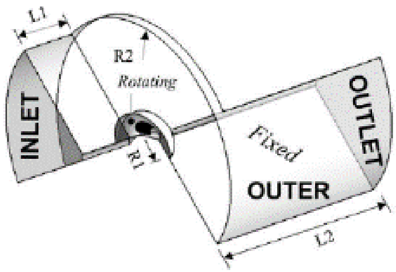

Fig. 3. Computational domain of propeller model [11].

Table 1. Dimension of computational domain [11].

Values	P5168		E779A	
	Rotating	Fixed	Rotating	Fixed
R1	0.72D		0.70D	
Lmid	0.75D		0.76D	
R2		5D		5D
L1		3D		3D
L2		5D		5D

2.1 Computational Fluid Dynamic simulation

Nowadays, CFD methods have been developed in completed units in one set of code. Ansys 14.0 has been utilized to simulate the ship bank interaction effect on propeller performance and flow around the ship's hull in shallow water. There are three main steps involved for CFD simulations which are Pre-processing, Solver and Post-processing. The shear stress transport (SST) turbulence model had been used in this study, because it gave the best results in comparison with other turbulence models. The equations are shown as follows (Eq. 1 and 2):

Equation of κ:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left(\Gamma_k \frac{\partial k}{\partial x_j}\right) + G_k - Y_k + S_k \tag{1}$$

Equation of ω:

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \omega u_i) = \frac{\partial}{\partial x_j}\left(\Gamma_\omega \frac{\partial \omega}{\partial x_j}\right) + G_\omega - Y_\omega + D_\omega + S_\omega \tag{2}$$

Where G_κ and G_ω express the generation of turbulence kinetic energy due to mean velocity gradients and ω. Γ_κ and Γ_ω express the active diffusivity of κ and ω. Y_κ and Y_ω represent the dissipation of κ and ω due to turbulence. D_ω expresses the cross-diffusion term, S_κ and S_ω are user-defined source terms [16 and 17]. Figs. 4 and 5 show the mesh elements of two computational domains. The main domain includes flexible free surface (between air and water) with ship model and propeller domain respectively:

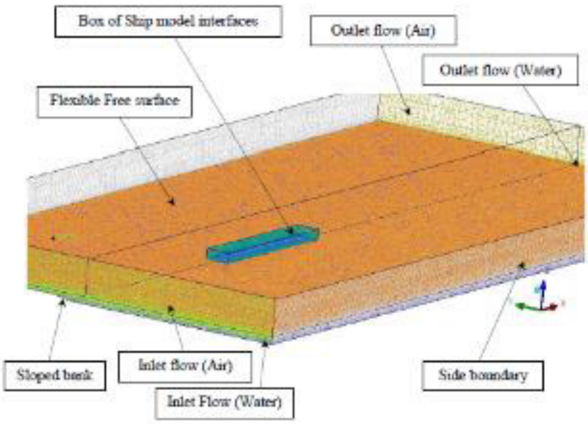

Fig. 4. Mesh elements of Stationary computational domain

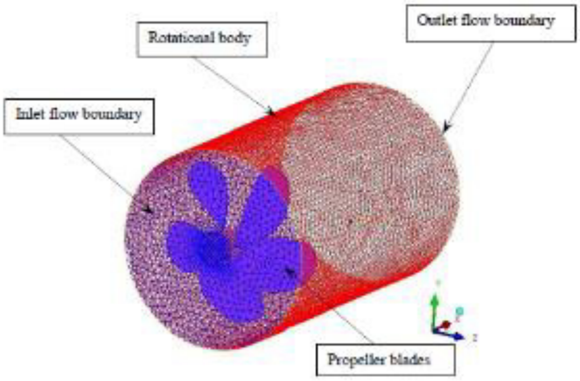

Fig. 5. Mesh elements of the rotational domain

2.2 Model test and validation

Hydrodynamic propeller performance and wake pattern at propeller plane of the LNG model were experimentally investigated in open water, near bank and in shallow water conditions to validate the numerical simulation. The experimental works were performed in Marine Technology Centre (MTC) - Universiti Teknologi Malaysia (UTM) which is shown in Figs. 6 and 7.

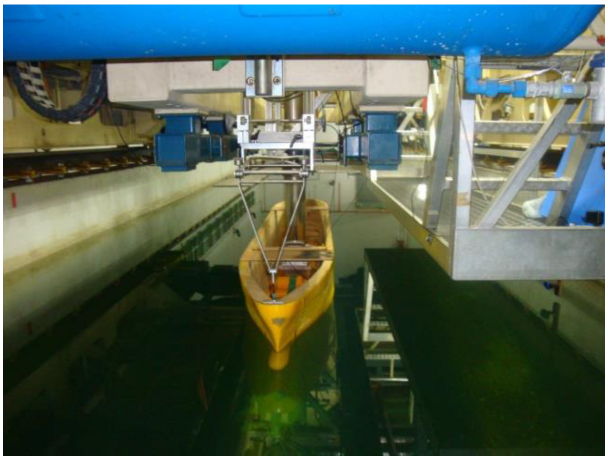

Fig. 6. The LNG model attached to the carriage in towing tank of UTM-MTC.

Fig. 7. LNG carrier model test in near bank and shallow water conditions.

3.1 Results and discussion

The self-propulsion test has been simulated to determine the changing of the propeller thrust (T) due to the propeller load variation in open, shallow and limited water conditions numerically. The thrust of the propeller behind the LNG ship model in open, shallow are shown in Figures 8 and 9 respectively.

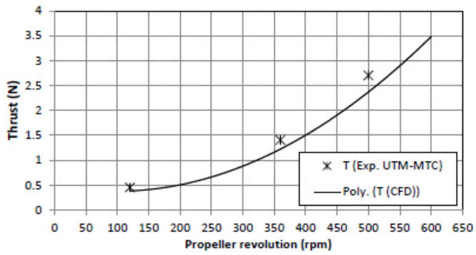

Fig. 8: Self-propulsion test results to validate the CFD calculation results of hydrodynamic thrust (T) for the propeller behind the LNG ship model in open water condition with constant ship speed (Vm=0.35m/s, Vs=6knots) versus variable propeller revolution speed.

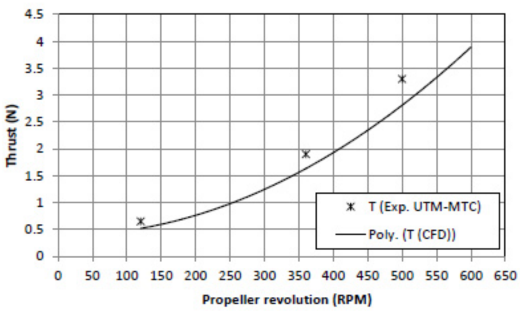

Fig. 9. CFD calculation of hydrodynamic thrust (T) of the propeller behind the LNG ship model in shallow water condition (H/T=1.2) with constant ship speed (Vm=0.35 m/s, Vs=6knots) versus propeller revolution speed.

Nominal wake pattern at propeller plane of the LNG ship model is investigated numerically in open water and restricted water conditions. Figures 10 show the half body velocity contours of axial velocity at the propeller plane in open water condition. The nominal wake pattern in deep and unrestricted water is symmetric due to the symmetrical incoming flow to the propeller disc.

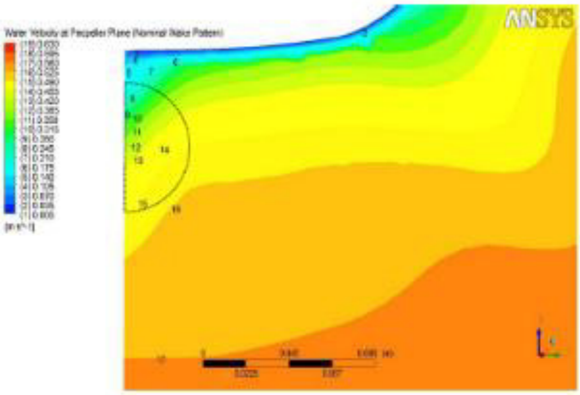

Fig. 10. CFD calculation of nominal wake pattern in open water condition for the LNG ship model at Vs=10knots.

Conclusions

This research has investigated the hydrodynamic propeller performance of LNG vessel near bank and in shallow water. Investigation conducted using a numerical simulation method. The conclusions are listed as follows:

i. The water depth to draft ratio (H/T) and ship-bank distance (Y/B) effected significantly on propeller performance. For a certain propeller revolution (RPM) and ship speed, in the lower water depth and bank distance, the thrust and torque increased due to sea bead and wake fraction effects.

ii. Ship resistance increased in lower water depth and ship-bank distance, because flow velocity increased under the ship bottom. This increment of resistance occurred due to turbulence flow under the ship bottom in shallow water depth and also the resistance increased dramatically near the bank.

Acknowledgements

This research was supported by Universiti Teknologi Malaysia (UTM) – Marine Technology Centre (MTC).

References

[1] Lataire, E., and Vantorre, M. (2008). Ship-bank interaction induced by irregular bank geometries. Symposium on Naval Hydrodynamics, 27th, Proceedings. 2, 511–524.

[2] Lataire, E., Vantorre, M., Laforce, E., Eloot, K., and Delefortrie, G. Navigation In Confined Waters: Influence Of Bank Characteristics On Ship-Bank Interaction. 2nd international conference on marine research and transportation. 28-30 June 2007, Ischia Naples Italy.2007.

[3] Norrbin, N.H. Bank effects on a ship moving through a short dredged channel. Proceedings 10th Symposium on Naval Hydrodynamics, Cambridge, MA, USA. 24-28 jun .1974.71-88.

[4] Dand, I. (1981). On ship-bank interaction. Royal Institution of Naval Architects. 25- 40.

[5] Zou, L., and Larsson, L. (2013). Computational fluid dynamics (CFD) prediction of bank effects including verification and validation. Journal of Marine and Science Technology. 18,310–323.
[6] Duffy, J. T. (2002). Prediction of Bank Induced Sway Force and Yaw Moment for Ship Handling Simulator. Ship Hydrodynamics Centre, Australian Maritime College (AMC).
[7] Maimun, A., Priyanto, A., Rahimuddin, S., Baidowi, A., and Nurcholis. Manoeuvring of a Vessel in Shallow Water with Ship-Bank Interaction Effect, Proceedings of International Conference in Ocean Engineering, ICOE 2009 IIT Madras, Chennai. India. 1-5 Feb.2009.
[8] Norrbin, N.H. Bank effects on a ship moving through a short dredged channel. Proceedings 10th Symposium on Naval Hydrodynamics, Cambridge, MA, USA. 24-28 jun .1974.71-88.
[9] Norrbin, N. H. Bank clearance and optimal section shape for ship canals, In: 26th PIANC international navigation congress, Brussels, Belgium. Section 1, Subject 1.1985. 167–178.
[10] Morgut, M., and Nobile, E. (2012). Influence of grid type and turbulence model on the numerical prediction of the flow around marine propellers working in uniform inflow. Ocean Engineering. 42, 26-34.
[11] Zhang, Z. R. (2010). Verification and validation for RANS simulation of KCS container ship without/with propeller. Journal of Hydrodynamics, Ser. B. 22(5), 932-939.
[12] Zadravec, M., Basic, S., and Hribersek, M. (2007). The influence of rotating domain size in a rotating frame of reference approach for simulation of rotating impeller in a mixing vessel. Journal of Engineering Science and Technology. 2(2), 126-138.
[13] Li, D. Q., Leer-Andersen, M., Ottosson, P., and Tragardh, P. Experimental investigation of bank effects under extreme conditions. In Practical Design of Ships and Other Floating Structures. Proceedings of the Eighth International Symposium on Practical Design of Ships and Other Floating Structures. 1.2001.
[14] Kerwin, J. E. and Lee, C. S. (1978). Prediction steady and unsteady marine propeller performance by numerical lifting surface theory. transaction of the society of naval architect and marine engieeners.86.
[15] Streckwall, H. (1986). A method to predict the extent of cavitation on marine propellers by lifting-surface-theory. International Symposium on Cavitation. Sendai, Japan.
[16] Kim, H. T. and stern, F. (1990). Viscous flow around a marine propeller-shaft configuration with infinite pitch rectangular blades. Journal of propulsion and power. 6(4), 434-444.
[17] Seo, J. H., Seol, D. M., Lee, J. H., and Rhee, S. H. (2010). Flexible CFD meshing strategy for prediction of ship resistance and propulsion performance. International Journal of Naval Architecture and Ocean Engineering. 2(3), 139-145.