Three-dimensional numerical analysis of restricted water effects on the flow pattern around hull and propeller plane of LNG ship

M. Nakisa, A. Maimun, A.Y.Sian, Yasser M. Ahmed, A. Priyanto, Jaswar, F. Behrouzi

Abstract**—**This paper aims to show bank and shallow water hydrodynamic force and moment effects on hull of Liquefied Natural Gas (LNG) ship that Lateral and vertical hydrodynamic forces are generated due to influence of confined water by using Computational Fluid Dynamics (CFD) methods. The simulation method is a numerical solution of the Navier-Stokes equation based on a finite volume method. Liquefied Natural Gas (LNG) carrier frequently navigates through noticeable banks with shallow water for its maneuvers. The flow pattern around a travelling or navigating ship in restricted water is affected significantly by the water depth and vicinity of the wall of a quay or canal. Sailing the ship in vicinity of bank may consequence in making hydrodynamic loads such as sway force and yaw moment on ship hull also changing velocity and pressure contours around the hull and the generation of asymmetric flow around the ship, hence may lead to potentially dangerous situations. In this research work the behaviour of a LNG carrier in confined water where there is a confined in water depth and in waterway width has been studied experimentally and numerically. From the results of pressure and velocity measured, the distribution of hydrodynamic lateral forces was obtained. Acting of hydrodynamic forces and moments on the hull ship and wake pattern behind ship hull has been investigated using CFD. The numerical results have been validated by comparison with model testing in Marine Teknologi Center (MTC) of Universiti Teknologi Malaysia (UTM).

Mehdi Nakisa is PhD Student in the Mechanical Engineering Faculty, Department of Marine Engineering, Marine Technology Center, Universiti Teknologi Malaysia (UTM), Skudai, Johor Bahru, 81310; tel. +60-755-36542; e-mail: nmehdi5@live.utm.my).

Adi Maimun is Professor in the Mechanical Engineering Faculty, Department of Marine Engineering, Marine Technology Center, Universiti Teknologi Malaysia, Skudai, Johor Bahru, 81310; (corresponding author, phone:+60-755-34761;fax:+60-755-66159;email: Adi@fkm.utm.my)

Yasser Mohamed Ahmed is Senior Lecturer in the Mechanical Engineering Faculty, Department of Marine Engineering, Marine Technology Center, Universiti Teknologi Malaysia(UTM), Skudai, Johor Bahru, 81310; tel. +60-755-34644; e-mail: Yasser@mail.fkm.utm.my).

Ang Y. Sian is PhD Student in the Mechanical Engineering Faculty, Department of Marine Engineering, Marine Technology Center, Universiti Teknologi Malaysia(UTM), Skudai, Johor Bahru, 81310; tel. +60-755-35957; e-mail: Angang503@yahoo.com).

Agoes Priyanto is Senior Lecturer in the Mechanical Engineering Faculty, Department of Marine Engineering, Marine Technology Center, Universiti Teknologi Malaysia(UTM), Skudai, Johor Bahru, 81310; tel. +60-755-34744; e-mail: Agoes@fkm.utm.my).

Jaswar is Senior Lecturer in the Mechanical Engineering Faculty, Department of Marine Engineering, Marine Technology Center, Universiti Teknologi Malaysia(UTM), Skudai, Johor Bahru, 81310; tel. +60-755-34645; e-mail: Jaswar@fkm.utm.my).

Fatemeh Behrouzi is Master Student in the Mechanical Engineering Faculty, Department of Thermo-Fluid, Universiti Teknologi Malaysia (UTM), Skudai, Johor Bahru, 81310; tel. +60-755-36542; e-mail: Bfatemeh5@live.utm.my)

Keywords**—** Bank effect, Shallow water, CFD, LNG carrier.

I. INTRODUCTION

NOWADAYS, focusing on the ship navigation in restricted waters such as canals, ports and channels with increasing the size and dimensions of LNG ships, container ships, but then, confined waters for navigation aren't increased at the same level. As a result, ships are influenced by lateral and vertical hydrodynamic forces which produced by canal wall and bed. Hydrodynamic effect of restricted water (bank-shallow) on the ship manoeuvring is called "ship-bank effect".

Effects of such interaction can results in characteristics such as ship alter course and lose steerage efficiency, double the squats of vessels, developing an angle of heel, drawn smaller ship towards larger vessel or worst, change in the propulsion efficiency, change in the propeller performance, change on the rudder performance, effect on manoeuvrability of ships and collided or capsized.

Hydrodynamic ship behaviours in restricted water are significant influenced by banks and shallow water while she navigates in confined water. LNG Tanker "Tenaga Kelas Satu "owned by Malaysian International Shipping corporation berhad (MISC Berhad) was grounded at Lake Charles in Jun. 2004 and Starboard bow below waterline touched the bank channel after failing to steer back since the ship turn to starboard. On the other hand, the same incident occurred for another LNG tanker "Tenaga Kelas Empat" owned by MISC Berhad at Suez Canal in Nov. 2007. [1]

Influence of ship-water velocity ratio, depth of water, propeller rotation rate, bank geometry, rudder angle and ship hull types on hydrodynamic forces of model tests conducted, also all coefficients and parameters of motion formulation determined by experimental tests and extent each items affected by confined channel width and finite depth. [2]

Ship hydrodynamics characteristics are significant affected by vertical and slope walls of restricted waterways. There are noteworthy hydrodynamic consequences in experimental tests with model tanker to determine the yaw moment and sway force on ship manoeuvring. [3]

Channels with varying width are causing asymmetric flow field around ship's hull and vertical and horizontal hydrodynamic forces in presence of shallow water and close bank to the ship. Several characteristics of asymmetric forces on ship's hull due to the bank and shallow water effect are caused by varying width of channel. Results of simulations are shown that it is noticeably large to the possible dangers of grounding and collision for ship traveling in varying width of channel. [4]

Changing the geometry of bank has significant effect on ship's hull hydrodynamic forces. Several kinds of bank such as vertical bank, submerged bank, surface piercing bank with varying in horizontal situation between ship and bank, slope, water depth above the flooded bank was conducted in experimental tests to determine the yaw moment, surge and sway forces on LNG tanker's hull. [5, 6 and 7]

LNG carrier often travels through restricted waters with visible banks to load and unload its hazardous cargo. When ship deviates from the center of channel, the ship unavoidably goes close the bank, and results in non-symmetric flow around the ship hull. As a consequence of Bernoulli's theorem, pressure difference is generated between port and starboard sides of vessel. A low pressure region generated due to the accelerated flow between the vessel and bank and manifest into a suction forces at stern of vessel attracting the vessel to bank, while a cushioning effect is induced at the bow which will result in a yaw moment swinging the bow away from bank. Such situations affect manoeuvring and course keeping which can lead to potentially dangerous situations, especially for LNG tanker.

It is considerable that the application of numerical methods using computational Fluid Dynamics (CFD) with finite volume method (FVM) not only conducted for maritime fields but also it is spread out for different cases such as prediction of wind flow around the high-rise buildings for urban street canyon [8], and turbulence models for isothermal and thermal flows in idealized street canyon [9], simulation of single-phase and two-phase flow dynamics in the HLTP for microalgae culture [10], Analysis of air velocity distribution in a laboratory batch-type tray air dryer by computational fluid dynamics [11], also the Lattice Boltzmann Method (LBM) has been extensively in different field of computational fluid dynamics as microfluidics, bio-fluidics, particulate flows and multiphase flows [12]. Computational Aerodynamic Investigations of a Car [13], Application of Euler-Euler Model for two phase flow field for radial turbine using Numerical Simulation [14], pressure distribution of fluid flow over ridges of circular using LES (Large Eddy Simulation), for different kind of shapes such as Parabolic and Rectangular [15], simulation of ventilation effects on indoor radon in a detached house [16], Investigation of the flow characteristics for a solenoid valve using computational fluid dynamic with industrial applications[17], aerodynamic characteristics of wing of WIG catamaran vehicle during ground effect[18], flow field phenomena about lift and down force generating cambered aero-foils in ground effect [19].

However the test is expensive and takes long time, Model test remain the primary source of data for marine hydrodynamic test. using Computational Fluid Dynamics (CFD) techniques in expecting the hydrodynamic forces influence on vessel is gradually increasing in the maritime community, this include further study of bank induced hydrodynamic forces in proximity of bank in complicated situations. CFD also applied in investigation of bank induced hydrodynamic forces on hull of a ship in confined waterway.

The results presented generally confirmed the feasibility of CFD technique to study on bank effect. Application of CFD in hydrodynamic analysis is particularly interesting since they can be used for a wide range of different application. Although these methods are still not wholly reliable, their use as a design tool will become widely accepted and used in the near future. [20].

Present study focus on examination of ship-bank interaction effects on flow pattern around the hull and nominal wake pattern in propeller plane of LNG tanker in restricted water using CFD simulation. Commercial CFD package, ANSYS-CFX, which using 3D Reynolds-Averaged Navier-Strokes (RANS) solver based on finite volume method (FVM) is applied in this study.

II. MODELING

A. LNG ship characteristics

The ship model studied in this research work for numerical simulations and experiments is a "Tenaga" Class LNG tanker, at scale of 1/80. Fig. 1 and Table I are shown the body lines and the main characteristics of the LNG ship.

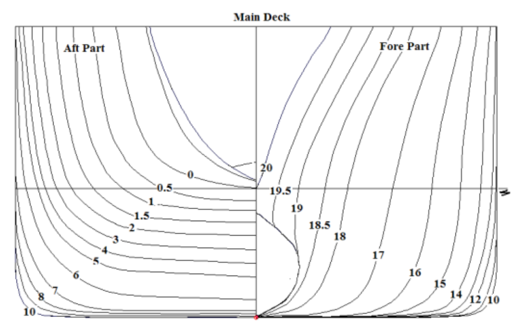


Fig. 1 Body plan of the LNG ship

The wooden model was fabricated in Marine Teknologi Center (MTC) - University Teknologi Malaysia and tested in towing tank in case of shallow water test.

Same Froude number in model and full scale ensures the model and full-scale ships exhibit similar behaviour. In general, the Froude number Fn is defined as:

$$Fn = \frac{V}{\sqrt{gL_{wl}}} \quad (1)$$

Where V is the velocity of the ship, g is the acceleration due to gravity, and L_{WL} is the length of the ship at waterline level.

TABLE I
Main characteristics of LNG ship and model

Main Characteristics	Symbol	Full Scale	Model	Unit
Length Overall	L_{OA}	280.62	3.50775	[m]
Length of Waterline	L_{WL}	268.414	3.355175	[m]
Length Between Perpendiculars	L_{BP}	266	3.325	[m]
Breadth of Waterline	B	41.6	0.52	[m]
Draught	T	11.13	0.139125	[m]
Wetted Surface (Bare hull)	S	13970.15	2.182837	[m2]
Water Density	ρ	1,025	1,000	[kg/m3]
Gravity Acceleration	g	9.81	9.81	[m/s2]
Block Coefficient	C_B	0.746	0.746	----
Scale Factor	λ	1	80	----
Material	---	Steel	Wood	----

III. Numerical and Mathematical Simulations

A commercial RANS code Ansys-CFX has been used in this study for simulating the flow around the LNG hull at the different speeds. CFX code utilizes the finite volume method for simulation. Figs. 2, 3, 4, 5 and table II show the computational domain arrangements and mesh information for the LNG carrier in restricted and open water, respectively.

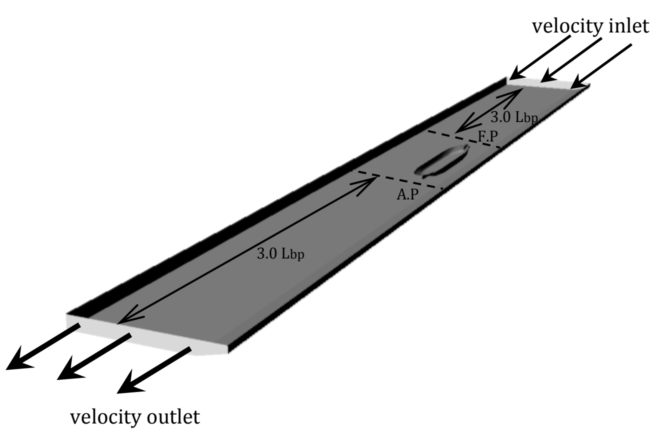


Fig. 2 Computational Domain in restricted water

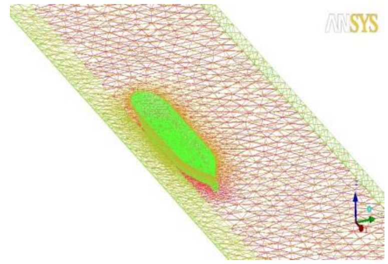


Fig. 3 Computational Domain in restricted water

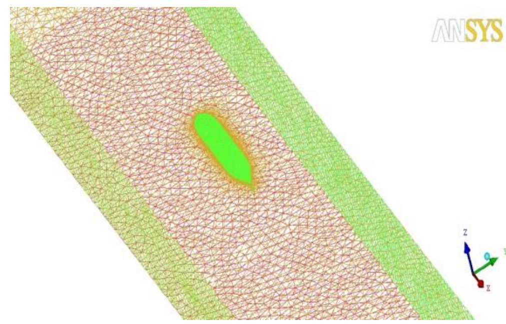


Fig. 4 computational domain in open water

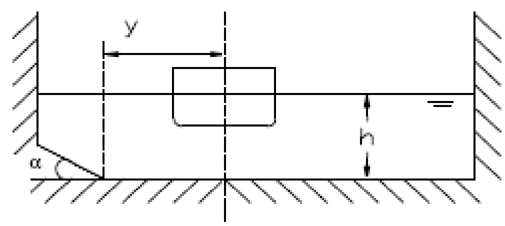


Fig. 5 arrangement of confined water

Table II
Mesh information of numerical domain

Computational Domain	Total Elements	Total Nodes
open water	393893	80634
restricted water	171629	51389

IV. Governing Equation

In Cartesian tensorial form the general Reynolds Average Navier-Stockes (RANS) formulation for continuity is written as,

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0 \qquad (2)$$

Momentum formulation is become as follows:

$$\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} =$$

$$-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}-\frac{2}{3}\delta_{ij}\frac{\partial u_i}{\partial x_i}\right)\right]+$$

$$\frac{\partial}{\partial x_j}\left(-\rho\overline{u_i'u_j'}\right)+f_{bi} \qquad (3)$$

In the above equation u_i is ith Cartesian component of total velocity vector, μ is molecular viscosity, $(-\rho\overline{u_i'u_j'})$ is Reynolds stress, δ_{ij} is Kronecker delta and p is static pressure. The Reynolds stress should be demonstrated to near the governing equations by suitable turbulent model. For solution the RANS equation and turbulence velocity time scale, it is used by Boussinesq's eddy-viscosity supposition and two transport equations. The body force is expressed by fbi.

For determination the 3D viscous incompressible flow around the ship's hull is used the ANSYS-CFX14.0 code. The parallel version of CFX concurrently calculates the flow formulations using numerous cores of computers.

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:

Equation of κ:

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left(\Gamma_k\frac{\partial k}{\partial x_j}\right)+G_k-Y_k+S_k \qquad (4)$$

Equation of ω:

$$\frac{\partial}{\partial t}(\rho\omega)+\frac{\partial}{\partial x_i}(\rho\omega u_i)=\frac{\partial}{\partial x_j}\left(\Gamma_\omega\frac{\partial \omega}{\partial x_j}\right)+G_\omega-Y_\omega+D_\omega+S_\omega \qquad (5)$$

Where G_κ and $G_\omega$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.[21]

The forces and moments acting on the hull can be approximated by the following polynomials of v' and r' by the following expressions [22].

$$X_H=\frac{1}{2}\rho L^2U^2[X'_{\dot{u}}\dot{u}'+X'_{vr}v'r'+X'_{vv}v'^2+X'_{rr}r'^2]+\frac{1}{2}\rho L^2U^2R'_{TM} \qquad (6)$$

$$Y_H=\frac{1}{2}\rho L^2U^2[Y'_{\dot{v}}\dot{v}'+Y'_{\dot{r}}\dot{r}'+Y'_v v'+Y'_{vvv}v^3+Y'_{vvr}v'^2r'+Y'_{vrr}v'r'^2+Y'_{rrr}r'^3] \qquad (7)$$

$$N_H=\frac{1}{2}\rho L^3U^2[N'_{\dot{v}}\dot{v}+N'_{\dot{r}}\dot{r}'+N'_v v'+N'_r r'+N'_{vvv}v^3+N'_{vvr}v'^2r'+N'_{vrr}v'r'^2+N'_{rrr}r'^3] \qquad (8)$$

The primes in Eqs. (6) – (8) refer to the non-dimensional quantities, defined as the following:

$$v'=\frac{v}{U};\quad \dot{v}'=\frac{\dot{v}L}{U^2};\quad r'=\frac{rL}{U};\quad \dot{r}'=\frac{\dot{r}L^2}{U^2} \qquad (9)$$

$$X'=\frac{X}{\frac{\rho}{2}L^2U^2};\quad Y'=\frac{Y}{\frac{\rho}{2}L^2U^2};\quad N'=\frac{N}{\frac{\rho}{2}L^3U^2} \qquad (10)$$

$$R'=\frac{R}{\frac{\rho}{2}L^2U}\text{ ; R = Ship Resistance} \qquad (11)$$

N is sum of yaw moments acting on the ship and N'_v , $N'_r, N'_{vv}, N'_{rr}, N'_{vvr}, N'_{vrr}, N'_{rrr}$ are hydrodynamic coefficients for the yaw moment, also Y is sum of forces acting on the ship in the transverse direction and Y'_v , $Y'_r, Y'_{vv}, Y'_{rr}, Y'_{vvr}, Y'_{vrr}, Y'_{rrr}$ are hydrodynamic coefficients for sway force. X is sum of forces acting on the ship in the longitudinal direction.

The computational setting for using the ANSYS-CFX is tabulated in Table III as follows:

TABLE III
Computational settings

Parameter	Setting
Computing	64-bit Desktop pc 6GB of RAM
Simulation type	Steady state
Mesh type	Unstructured hybrid(tetrahedral/prism)
Turbulence model	k-ω (Shear stress transport)
Wall modeling	Automatic wall function based on a law of the wall formulation
Advection scheme	CFX high resolution

V. Results and Discussion

A series of model experiments to measure the ship-bank interactive forces were conducted to validate the numerical calculations. The experiments were conducted in the towing tank at Marine Technology Centre of Universiti Teknologi Malaysia (UTM) that shows in Figs. 6, 7 and 8.

Fig. 6 Towing tank and towing carriage in UTM

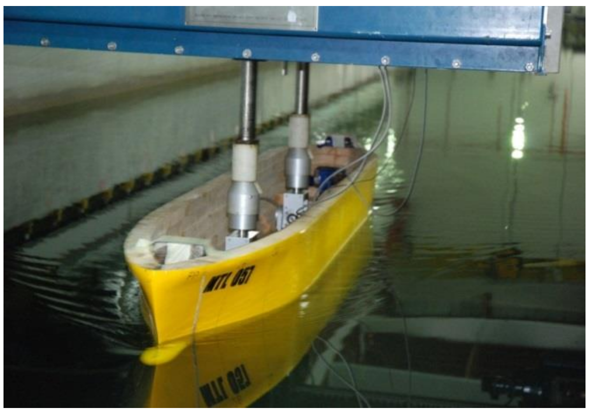


Fig. 7 Ship model test in open water

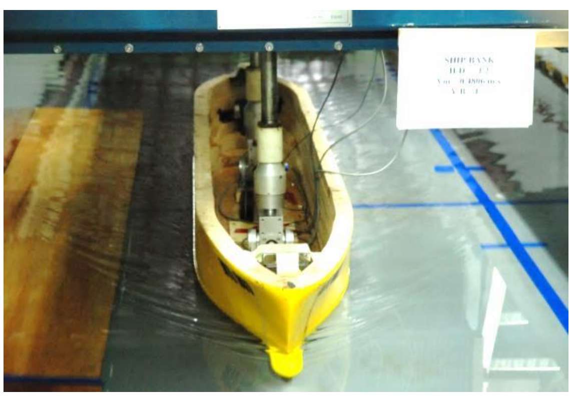


Fig. 8 Ship model in testing situation to study the effect of bank and shallow water

The measured hydrodynamic forces and moments while various ship speed, ship-bank distance, and water depth are shows in Fig. 9.

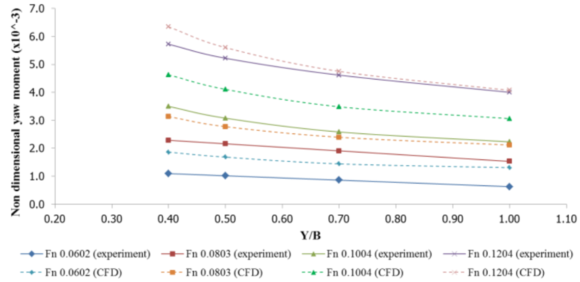


Fig. 9 Effect of ship speed on yaw moment at H/D =1.6

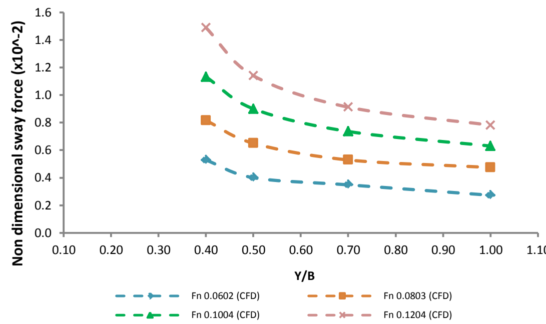


Fig. 10 Effect of ship speed on sway force at H/D =1.6

Fig. 10 shows that when ship navigate in nearer to the bank, the sway force effect of bank would be more significant in higher speed.

The vessel experienced a bow out moment, which becomes bigger as the ship-bank distance decreases. It is also noticed that for a given ship-bank distance, the bank effect increase with higher ship speed. In general, the bank effect reduces with the increase of ship-bank distance and the reduction of ship speed.

The streamlines on the LNG ship's hull model surface predicted by CFX code has shown a good agreement with streamlines that generated by Paint Smear Test (PST) on the hull surface model test which conducted in MTC-UTM and as can be seen in Figs. 11 and 12.

Fig. 11 Paint Smear Test velocity streamlines

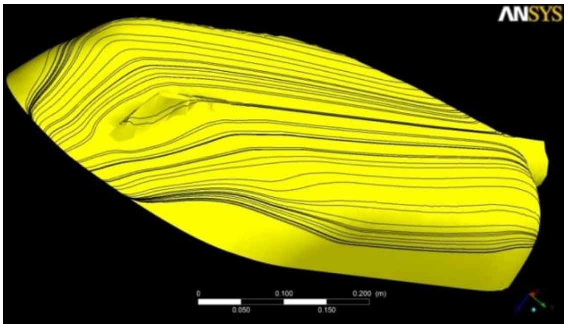
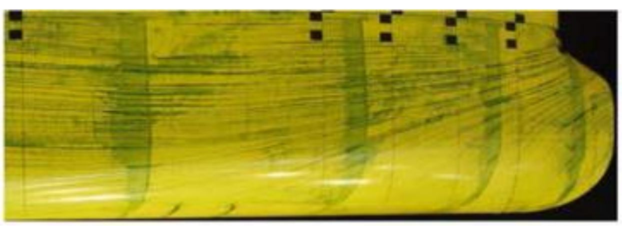


Fig. 12 CFD velocity streamlines

Test and computational fluid dynamic results in case of limiting streamlines on the fore of hull surface of Tenaga class LNGC model [23] are shown in Fig. 13. For the best design of the bow thruster position, grid angle, bilge keel stabilizer and find the wake pattern in propeller plane use this important information. The streamlines direction is downward at the fore region of the hull ship surface. The most important factors that affected on the ship speed performance are the streamlines angle at the bow thruster zone. The computational results represent the streamlines are agreed with the test.



(a) Experimental: Limiting streamline

(b) Simulation: Limiting streamline

Fig. 13 Limiting streamlines on the hull (a) test and (b) CFD

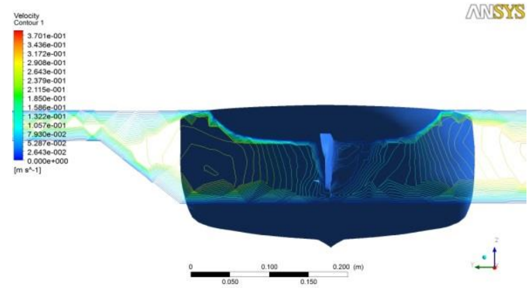


Fig. 14 Velocity line contours at inflow propeller plane

Effect of slop bank and shallow water on velocity contours at inflow propeller plane is shown in Fig. 14. The velocity line contours show that the flow velocity in the bank effect side is more than the other ship side. It means, the pressure between bank and ship's hull is decreased and this phenomenon makes absorb each other and it may lead to ship hull bank connecting.

For better understanding of the relation between lateral force and flow field, the computed pressure coefficient on the hull surface is shown in Fig. 15 using color contours. The bank effect side and the open water side are named inside and outside positions, respectively.

Inside (Bank Effect)

Outside (without bank)

Fig. 15 Inside and outside positions of the hull surface

Contour maps of the computed pressure distribution of inside of the hull surface and outside hull surface are shown in in Fig. 16 (a) and (b), respectively. Inside hull surface has lower pressure distribution than outside hull surface, means that the bank and shallow water affected on inside hull surface and the pressure between bank and hull caused to absorb the hull to bank and the shortage of pressure under the ship bottom makes grounding, as well.

a) Inside of the hull surface

b) Outside of the hull surface

Fig. 16 Measured distributions of pressure coefficients on the hull surface

Fig. 17 shows that the streamline around the ship hull in open water is symmetric on both sides on the ship hull at the streamline around her hull are asymmetric near bank as shows in Fig. 18. In addition, the pressure contours are symmetric in case of deep water and asymmetric under the effect of bank and shallow water, as shown in Figs. 19 and 20. The high velocity under the hull bottom in shallow water, it might lead to hull grounding.

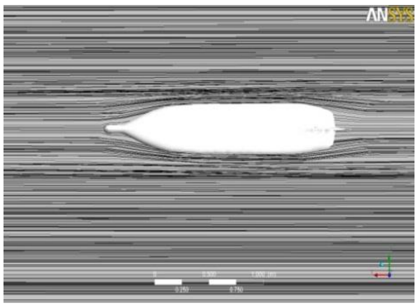


Fig. 17 streamlines around the ship in deep water

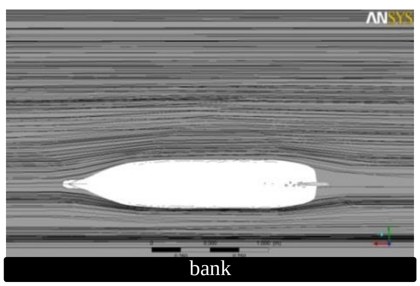


Fig. 18 Streamlines around the ship in confined water

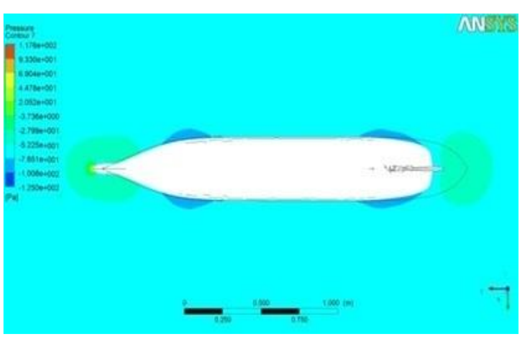


Fig. 19 Pressure contours around the ship in deep water

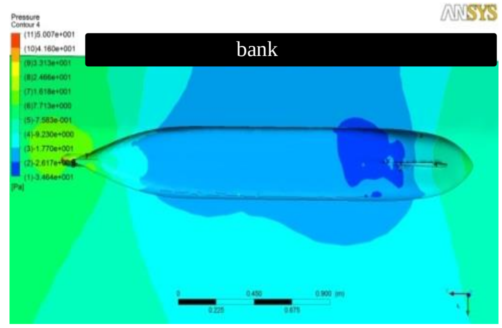


Fig. 20 Pressure contours around the ship in confined water

VI. Conclusion

CFD simulations and experiments have been conducted for studying the bank effect on a LNG tanker in open and restricted water conditions. In according to the presented computational results based on RANS equations, the following conclusions are taken:

1- The proximity of bank and shallow water has a significant effect on the hydrodynamic pressure and velocity contours around the ship hull.

2- The hydrodynamic interactive force and moment increase with reducing of ship-bank distance. For a given ship bank distance, the bank effect is more pronounced with an increasing of ship speed.

3- In according to high velocity under the bottom in shallow water, it may lead to ship hull grounding.

4- Several factors may contribute to the modeling errors: neglect of free surface, air drag, non-free sinkage and trim, turbulence modeling, absence of propeller and rudder, and boundary conditions on solid walls.

Acknowledgments

The authors would like to express their sincere gratitude to MOHE–UTM- vote No. 00G44 (Universiti Teknologi Malaysia- UTM) for its supporting of this research project.

References

[1] A., Maimun, Priyanto, A., Rahimuddin, Sian, A.Y., Awal, Z.I., Celement, C.S., Nurcholis and Waqiyuddin, M., “Amathematical model on manoeuvrability of a LNG tanker in vicinity of bank in restricted water”, Journal of Safety Science, 53, pp.34-44, 2013.

[2] M., Fujino, “Experimental studies on ship manoeuvrability in restricted waters”, Part 1, International Shipbuilding Progress, Vol.15, No. 168, pp.279-301, 1968.

[3] N.H., Norrbin, “Bank clearance and optimal section shape for ship canals”, 26th PIANC International Navigation Congress, Brussels, Section 1, Subject 1, pp.167-178, 1985.

[4] H., Yasukawa, “bank effect on ship maneuverability in a channel with varying width”, the Japan Society of Naval Architects and ocean Engineering, No.81, pp.85-100, 1991.

[5] J.T. Duffy, “Prediction of bank induced sway force and yaw moment for ship handling simulator”, Australian Maritime College, pp.1-22, 2002.

[6] I.W., Dand, "some mesurements of interaction induced by Surface-piercing and Flooded Banks", National Maritme Institute Report, 1981, R110

[7] N. H., Norrbin, "Bank Effects on a Ship Moving Through a Short Dredged Channel", Proceedings 10th Symposium on Naval Hydrodynamics, Office of Ship Research, 1974

[8] F. Behrouzi, NOR Azwadi, M. Nakisa, A. Witri, "Numerical prediction of wind flow around the high-rise buildings by two equations turbulence models for urban street canyon", WSEAS/ International Conference on Proceedings of the 15th International Conference on Mathematical and Computational Methods in Science and Engineering (MACMESE '13)-Kuala Lumpur, Malaysia, April 2-4, 2013, pp.152-156.

[9] A. Witri, Nor Azwadi, S. Mohamed," Prediction of k-ε Turbulence Models for Isothermal and Thermal Flows in Idealized Street Canyon", WSEAS/ International Conference on Proceedings of the 15th International Conference on Mathematical and Computational Methods in Science and Engineering (MACMESE '13)-Kuala Lumpur, Malaysia, April 2-4, 2013, pp.173-177.

[10] U.Kumar Deb,K.Chayantrakom, Y. Lenbury,"Comparison of Single-phase and Two-phase Flow Dynamics in the HLTP for Microalgae Culture", NAUN Journal /international journal of mathematics and computers in simulation, Issue 5, Volume 6, 2012, pp.496-503

[11] D. A. Tzempelikos, A. P. Vouros, A. V. Bardakas, A. E. Filios, and D. P. Margaris, "Analysis of air velocity distribution in a laboratory batch-type tray air dryer by computational fluid dynamics", NAUN Journal/ international journal of mathematics and computers in simulation, Issue 5, Volume 6, pp.413-421, 2012.

[12] N. Azwadi, E. P. Kermani, L. Jahanshaloo," Force Evaluation on Different Shape of Particles using Lattice Boltzmann Curved Boundary Methods", WSEAS/ International Conference on Proceedings of the 15th International Conference on Mathematical and Computational Methods in Science and Engineering (MACMESE '13)-Kuala Lumpur, Malaysia, April 2-4, 2013, pp.163-167.

[13] M. Desai, S. A. Channiwala, H. J. Nagarsheth, "experimental and computational aerodynamic investigations of a car", WSEAS transactions on fluid mechanics, issue 4, volume 3, pp. 359-368, Oct. 2008.

[14] S. Schuster, Friedrich, K. Benra, H. J. Dohmen," Application of Euler-Euler Model for Numerical Simulation of a Radial Turbine Working in the Two-Phase Flow Regime", WSEAS transactions on fluid mechanics, Issue 4, Volume 5, pp. 276-286, Oct. 2010.

[15] A. J. Templeman, M. G. Rasul, "Large Eddy Simulation of Pressure Distribution of Fluid Flow Over Ridges of Circular, Parabolic and Rectangular Shapes", WSEAS transactions on fluid mechanics, issue 2, volume 6, pp. 113-122, Apr. 2011.

[16] K. Akbari, J. Mahmoudi, R. Oman, "Simulation of Ventilation Effects on Indoor Radon in a Detached House", WSEAS transactions on fluid mechanics, Issue 4, Volume 7, pp. 146-155, Oct. 2012.

[17] T. Kim, S. Yang, S. Kang, "Numerical Study on the Flow Characteristics of a Solenoid Valve for Industrial Applications", WSEAS transactions on fluid mechanics, Issue 3, Volume 5, pp. 155-164, Jul. 2010.

[18] A. Maimun, S. Jamei, A. Priyanto, NOR Azwadi, "Aerodynamic Characteristics of Wing of WIG Catamaran vehicle During Ground Effect", WSEAS transactions on fluid mechanics, Issue 3, Volume 5, pp. 196-205, Jul. 2010.

[19] J. W. Vogt, T. J. Barber, E. Leonardi, " Flow Field Phenomena about Lift and Downforce Generating Cambered Aerofoils in Ground Effect", WSEAS transactions on fluid mechanics, Issue 3, Volume 3, pp. 234-243, Jul. 2008.

[20] D.C., Lo, Su., Dong-Taur and chen., Jan-Ming, Application of computational fluid dynamics simulations to the analysis of bank effect in restricted waters, Journal of Navigation, Vol.62, pp. 477-491, 2009.

[21] A. Maimun, A., M. Y. Ahmed., A. Priyanto, Y. S. Ang., M. Nakisa., Assessment of Ship-Bank Interactions on LNG Tanker in Shallow Water, Department of Marine Technology, University Technology Malaysia, Johor Bahru, Malaysia, The 6th Asia-Pacific Workshop on Marine Hydrodymics-PHydro2012Malaysia,September 3-4, 2012

[22] Y.J. Sung, Yum, D.J., Rhee, K.P.,. "An Analysis of The PMM Tests Using a System Identification Method". In: The 7th International Marine Design Conference, Kyung-Ju, Korea, 2000.

[23] E. C. Jung, H. K. Jung, G. L. Hong, J. C. Bong, H. L. Dong, Computational predictions of ship-speed performance, Mar Sci Technol , 14:322–333, 2009.

Adi Maimun bin Hj Abdul Malik

Prof. Dr. Adi Maimun is corresponding author in this paper and WSEAS13 session chairman, Professor (Naval Architecture) at the Marine Technology Department, Faculty of Mechanical Engineering, Universiti Teknologi Malaysia (UTM). In 1976 received a government scholarship to join the Cadetship (deck) programme with Ocean Fleets Ltd., Liverpool, UK. In 1979 obtained a Diploma in Nautical Science from Riversdale College, Liverpool and 2nd Mates Masters of Foreign-going Certificate from DOT, UK. In July 1983 graduated from University of Strathclyde, UK with a B.Sc. degree in Naval Architecture. Joined Universiti Teknologi Malaysia in August 1983 as Assistant lecturer. Obtained M.Sc.and Ph.D. in Marine Technology (in 1985 and 1993 respectively) from University of Strathclyde. Head of Marine Laboratory (1986-1989), Head of Dept. Marine Technology (2000-2007); Deputy Dean (Development), Faculty of Mechanical Eng.(2007-2011). Fellow Member - Royal Institution of Naval Architects, UK (Membership No. 00167001). Ordinary Member – Malaysian Society for Engineering and Technology (Membership No.: 00148). Graduate Member - The Institution of Engineers, Malaysia (Membership No.: 07310). Chartered Engineer, Engineering Council, UK (2008). EIA Subject Consultant (No. SS0963) – Marine Traffic & Navigational, Department of Environment, Ministry of Natural Resources and Environment, Malaysia.