

RANS Simulation of Viscous Flow around Hull of Multipurpose Amphibious Vehicle

M. Nakisa, A. Maimun, Yasser M. Ahmed, F. Behrouzi, A. Tarmizi

Abstract—The practical application of the Computational Fluid Dynamics (CFD), for predicting the flow pattern around Multipurpose Amphibious Vehicle (MAV) hull has made much progress over the last decade. Today, several of the CFD tools play an important role in the land and water going vehicle hull form design. CFD has been used for analysis of MAV hull resistance, sea-keeping, maneuvering and investigating its variation when changing the hull form due to varying its parameters, which represents a very important task in the principal and final design stages. Resistance analysis based on CFD (Computational Fluid Dynamics) simulation has become a decisive factor in the development of new, economically efficient and environmentally friendly hull forms. Three-dimensional finite volume method (FVM) based on Reynolds Averaged Navier-Stokes equations (RANS) has been used to simulate incompressible flow around three types of MAV hull bow models in steady-state condition. Finally, the flow structure and streamlines, friction and pressure resistance and velocity contours of each type of hull bow will be compared and discussed.

Keywords—RANS Simulation, Multipurpose Amphibious Vehicle, Viscous Flow Structure.

I. INTRODUCTION

ANY amphibious vehicle design inherently involves optimization in hull form to increase the manoeuvrability and hull performance with fairing the flow structure around it, as competing requirements and design parameters force the design to evolve, and as designers strive to deliver the most effective and efficient platform possible within the constraints of time, budget, and performance requirements. A significant number of applications of computational fluid dynamics (CFD) tools to hydrodynamic optimization, mostly for reducing calm-water drag and wave patterns, demonstrate a growing interest in optimization. One difficulty with designing such new concepts is the lack of experience from which to draw from when performing design studies. Thus, optimization techniques may be particularly useful.

Resistance characteristic of amphibious boat is one of the most important topics in Naval Architecture, Offshore and Ocean Engineering. Today, several of the CFD tools play an

important role in the amphibious vehicle and other vehicles hull form design. CFD has been used for analysis of multipurpose amphibious vehicle hull resistance, sea-keeping, manoeuvring and investigating its variation when changing the hull form due to varying its parameters, which represents a very important task in the principal and final design stages. In preliminary stage, model test is expensive and time concise. In vehicle hydrodynamics where the accurate result is never possible and getting the resistance consequences of a hull form, optimization based on CFD solutions quantitative accuracy of integral results such as resistance is imperative.

As the speed and memory capacity of the computers increased and more sophisticated RANS codes were developed more realistic simulations were able to be performed. These advances are well documented in the proceedings of several international conferences on the application of CFD techniques to ship and multipurpose vehicle flows which have been held every few years since 1990, most notably in Tokyo [1], [3] and in Gothenburg [2].

Resistance co-efficient with free surface flows of ships like Wigley and Series 60 model and other Floating vessels computed [4]. Recently fluent code simulation has been implemented around another type of naval hull named DTMB5415 [5]. In this simulation wave profile for different findings of mesh has been showed using Volume of Fluid (VOF) model. Resistance estimates and their error estimation compared with the experimental values in the presence on free surface flows have been simulated [6].

Reynolds-averaged Navier-Stokes method has been implemented with the numerical solution of free-surface wave flows around surface-piercing cylindrical structures using an unstructured grid [7]. Numerical tests were also performed by [8] who proposed a VOF-based technique to simulate the flow around the foil and the validations suggested that the most efficient solutions were found when the high resolution interface capturing (HRIC) schemes are employed. In the open literature, there are only a few published papers on the design principles of amphibious vehicles. It is investigated several waterjet systems for Marine Corps applications [9].

The present research work emphasizes on simulation of three shapes of bow hull of multipurpose amphibious vehicle to compare the flow structure and streamlines around the hull to evaluate the hydrodynamic resistance.

II. MODELING AND GOVERNING AMPHIBIOUS VEHICLE FLOW EQUATIONS

The coordinate system (x, y, z) for calculating the viscous drag is defined to represent the flow patterns around hull form

Mehdi Nakisa, Fatemeh Behrouzi, and Ahmad Tarmizi are with the Mechanical Engineering Faculty, Universiti Teknologi Malaysia (UTM), Skudai, Johor Bahru, 81310 (phone: +60-755-36542; e-mail: nmehdi5@utm.live.my, bfatemeh5@utm.live.my).

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

Yasser Mohamed Ahmed is Senior Lecturer in the Mechanical Engineering Faculty, Universiti Teknologi Malaysia (UTM), Skudai, Johor Bahru, 81310; tel. +60-755-36644; e-mail: Yasser@mail.fkm.utm.my).

as positive y in the opposite flow direction, positive x in port side and positive z upward where the origin at the aft perpendicular of the hull form, as shown in Fig. 1. Side and perspective views are shown in Figs. 2 and 3 as well.

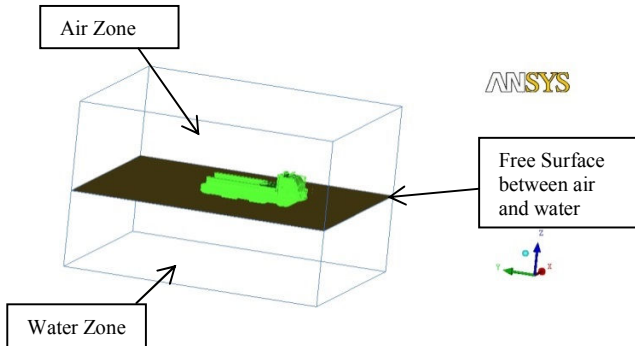


Fig. 1 Three-dimensional co-ordinate systems and computational domain for a hull design

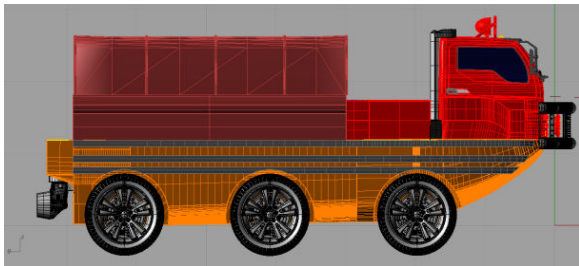


Fig. 2 Side view of multipurpose amphibious vehicles

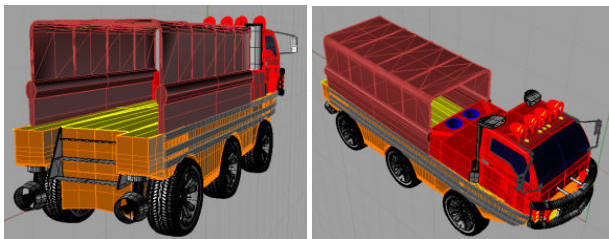


Fig. 3 Perspective view of MAV

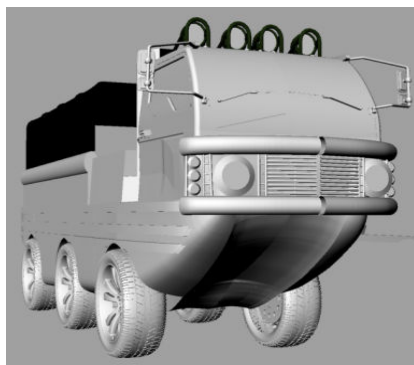


Fig. 4 U-shape hull bow

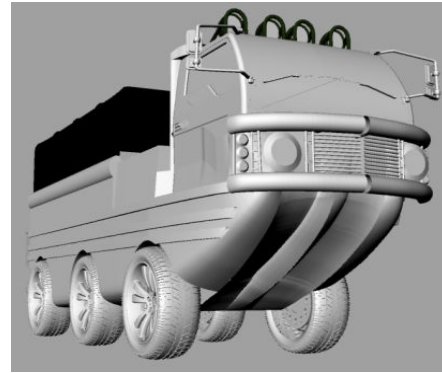


Fig. 5 V-shape hull bow

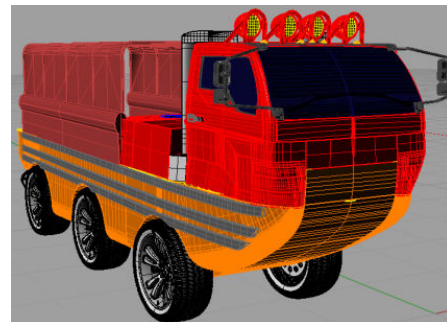


Fig. 6 Flat-shape hull bow

TABLE I
 CHARACTERISTICS OF MULTIPURPOSE AMPHIBIOUS VEHICLE

Loading Condition	Actual size	Model Size	Unit
LWL	6.607	1.65175	m
Beam	2.024	0.506	m
Draft	0.99	0.2475	m
Displaced volume	5.314	0.08303	m ³
Wetted area	31.719	0.33212	m ²
Prismatic coeff.	0.559	0.559	---
Waterplane area coeff.	0.665	0.665	---
LCG from midships	2.726	0.6815	m
Transom draft	0.025	0.00625	m
Max sectional area	1.438	0.08987	m ²
Deadrise at 50% LWL	19.33	19.33	deg.
Hard chine or Round bilge	Round bilge	Round bilge	-----
Headwind	0	0	kts
Scale	1	4	-----
Air density	0.001	0.001	Tonne.m ³
Kinematic viscosity	1.1883E-06	1.1883E-06	m ² s ⁻¹
Water Density	1.025	1.00	Tonne.m ³

The MAV is equipped with watertight compartments to achieve floatation capability. The vehicle is also equipped with additional water pumps in order to pump out the uncontrolled water ingress during the river crossing mission. Three geometry designs of MAV are shown in Figs. 4-6 and Characteristics of Multipurpose Amphibious Vehicle are given in Table I. Appendages, which are not a part of the main body such as wheels, drive trains etc. are considered as watertight compartments and added separately in stability calculations. In addition to floatability, the vehicle should also be stable in a floating condition.

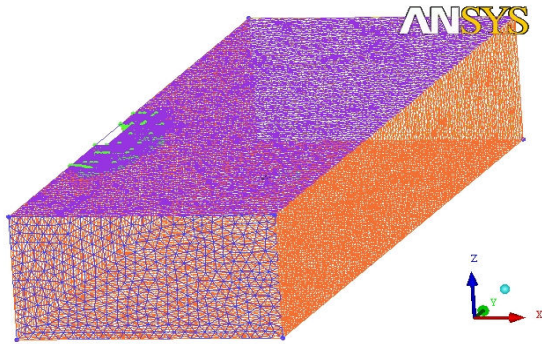


Fig. 7 Multipurpose amphibious vehicle computational domain

TABLE II
MESH INFORMATION OF NUMERICAL DOMAIN

Computational Domain	Total Elements	Total Nodes
water	1,636,197	374511

Fig. 7 and Table II show the computational domain and mesh elements which is modeled and simulated in analysis14.0 using Finite Volume Method (FVM).

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

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

where V is the velocity of the MAV, g is the acceleration due to gravity, and L_{WL} is the length of the Vehicle at waterline. In Cartesian tensorial form the general Reynolds Average Navier - Stokes (RANS) formulation for continuity is written as,

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

Momentum formulation is become as follows:

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \right) \right] + \frac{\partial}{\partial x_j} (-\rho \overline{u_i' u_j'}) + f_{bi} \quad (3)$$

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 f_{bi} .

For determination the 3D viscous incompressible flow around the ship's hull is used the ANSYS-Fluent14.0 code. The parallel version of Fluent 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 :

$$\frac{\partial}{\partial t}(\rho \kappa) + \frac{\partial}{\partial x_i}(\rho \kappa u_i) = \frac{\partial}{\partial x_j} \left(\Gamma_\kappa \frac{\partial \kappa}{\partial x_j} \right) + G_\kappa - Y_\kappa + S_\kappa \quad (4)$$

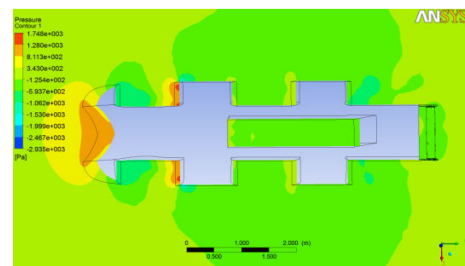
Equation of ω :

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

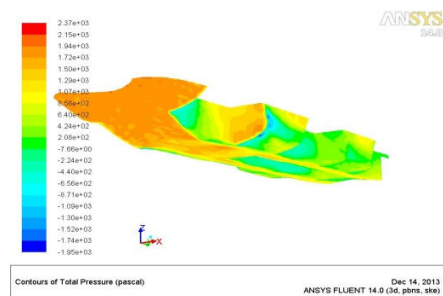
where G_κ and G_ω express the generation of turbulence kinetic energy due to mean velocity gradients and ω . Γ_κ and Γ_ω express the active diffusivity of κ and ω . Y_κ and Y_ω represent the dissipation of κ and ω due to turbulence. D_ω expresses the cross-diffusion term, S_κ and S_ω are user-defined source terms [10].

III. RESULTS AND DISCUSSION

Pressure distributions around three shapes of bow hull of multipurpose amphibious vehicle are shown in Figs. 8-10. The pressure resistance on the U-shape, V-shape and Flat-shape of bow hull of vehicle are accumulated on the front of the multipurpose amphibious vehicle.

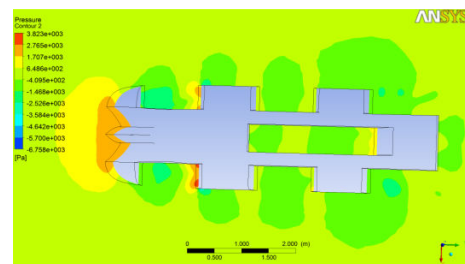


(a) Top view

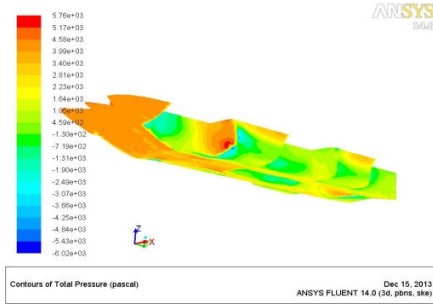


(b) Bottom view

Fig. 8 Pressure distribution around U-shape of bow hull

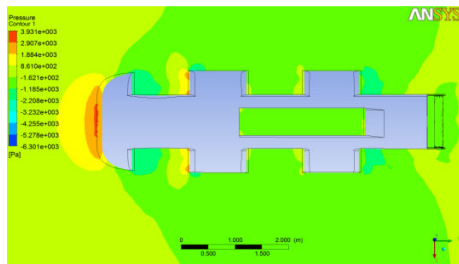


(a) Top view

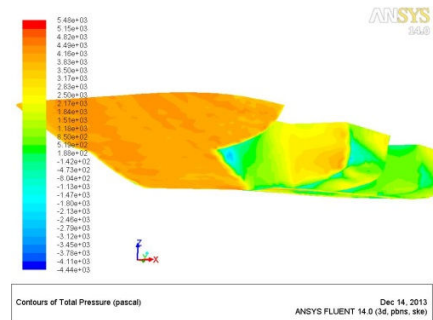


(b) Bottom view

Fig. 9 Pressure distribution around V-shape of bow hull



(a) Top view



(b) Bottom view

Fig. 10 Pressure distribution around Flat-shape of bow hull

Considering to pressure distributions contours, the U-shape bow hull vehicle has lowest pressure and drag resistance than the V-shape and Flat-shape bow hull of multipurpose amphibious vehicle. In service speeds (10-12 knots) the induced waters and waves are guided to go underneath of the U-shape of hull bow and both sides of hull. Wave fraction resistance in V-shape of hull bow has more significant effect for increasing the total resistance. In addition, this phenomena cause to increase the frictional resistance, added resistance and pressure resistance.

The resistance curves plotted against Froude number which are shown in Figs. 11-13, gives the optimum design for the U-shape of hull bow design in speeds more than 10-12knots, because in these range of speeds, the Multipurpose Amphibious Vehicle take up and the wetted surface will be decrease and friction resistance will be decrease as well. Fuel consumption reduction and speed increasing are related to total resistance in service speed range as well.

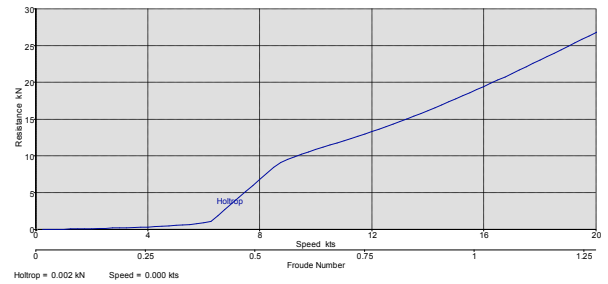


Fig. 11 Total resistance for U-shape of hull bow

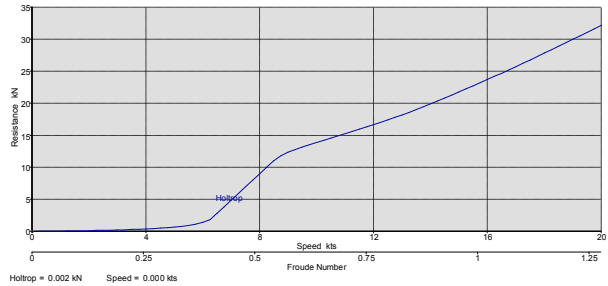


Fig. 12 Total resistance for V-shape of hull bow

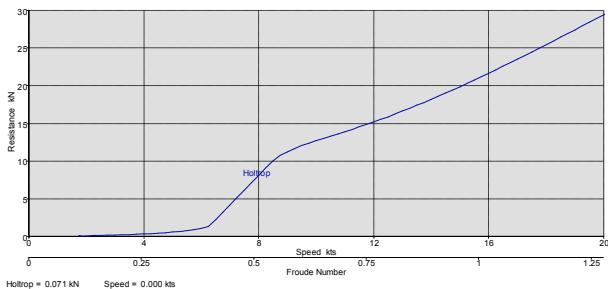


Fig. 13 Total resistance for Flat-shape of hull bow

The streamlines of flow structure around hull of multipurpose amphibious vehicle are shown in Figs. 14-16. The streamlines accumulated on the both sides of bow hull area.

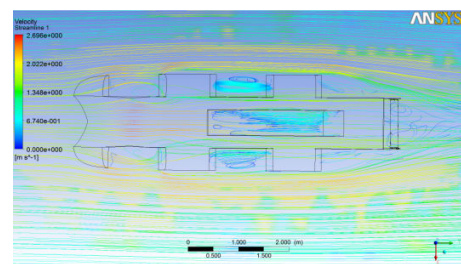


Fig. 14 Streamlines around U-shape of bow hull

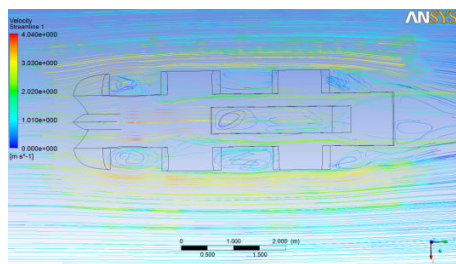


Fig. 15 Streamlines around V-shape of bow hull

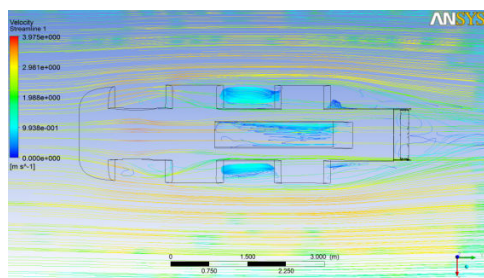


Fig. 16 Streamlines around flat-shape of bow hull

The streamlines around U-shape of bow hull design are more accumulated into the U-form, therefore the flow cause to flow up the whole body and decrease the draft, pressure and frictional resistance in the service speed in compare of V-shape and Flat-shape of bow hull designs.

IV. CONCLUSION

The research work has investigated the performance of the three different shapes designed of bow hull and optimized the best performance in accordance of the total resistance during the operating of the Multipurpose Amphibious Vehicle in various speed ranges. However, the added resistance is mainly dependent on the shape of the hull bow of designed vehicle.

When the U-shape of hull bow of MAV faced the water and wave, it forced the vehicle to flow up, which resulted to reduce the draft of the water and wave resistance, in addition, the wetted hull, friction resistance, pressure resistance, power of the vehicle, fossil fuel consumption and wave breaking resistance of the U-shape hull bow decreased compared with the others bow shapes. Meanwhile, the U-shape of the hull bow has reduced the total resistance to 20.3% and 13.6% compared with the V-shape and flat shape respectively. Though out, the U-shape of the hull bow is capable to increase the amphibious operating life and speed of vehicle.

ACKNOWLEDGMENT

The authors would like to thank the Faculty of Mechanical Engineering in Universiti Teknologi Malaysia (UTM) for financial support under MOHE-UTM-Vote No. 01G36.

REFERENCES

- [1] T. Hino, "Proceedings of the CFD Workshop Tokyo 2005", Tokyo, Japan, 2005.
- [2] Y. Kodoma, H. Takeshi, M. Hinatsu, T. Hino, S.Uto, N. Hirata and S. Murashige, "Proceedings of the 1994 CFD Workshop", Ship Research Institute, Japan, 1994.

- [3] L. Larsson, F. Stern and V. Bertram, "Benchmarking of Computational Fluid Dynamics for Ship Flows: The Gothenburg 2000 Workshop", Journal of Ship Research, 2003, vol. 47, iss. 1, pp. 63-81.
- [4] R.A. Repetto, "Computation of Turbulent Free-Surface Flows Around Ships and Floating Bodies", PhD. Thesis, Technical University Hamburg-Harburg, Argentina, 2001.
- [5] D.A. Jones and D.B. Clarke, "Fluent Code Simulation of Flow around a Naval Hull: The DTMB 5415", Maritime Platforms Division, Defence Science and Technology Organisation, Australian Government, Department of Defence, 2010.
- [6] D.F. Mucientes, "Numerical Simulation of Free Surface Viscous Flows around Single and Multiple Hulls", M.S Thesis, Department of Mechanical and Aerospace Engineering, California State University, Long Beach, December, 2010.
- [7] S.H. Rhee, "Unsteady Reynolds Averaged Navier-Stokes Method for Free-Surface Wave Flows Around Surface-Piercing Cylindrical Structures", Journal of Waterway, Port, Coastal and Ocean Engineering, 2009.
- [8] S.H. Rhee, B. P. Makrov, H. Krishinan, and V. Ivanov, "Assessment of the Volume of Fluid Method for Free Surface Wave Flow", Journal of Marine Science and Technology, 2005, vol. 10, pp. 173-180.
- [9] J.G. Sticker, A.J. Becnel and J.G. Purnell, Advanced Waterjet Systems. Nav. Eng. J. 106 (5), 1994, pp. 100-109.
- [10] 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 Hydrodynamics-PHYDRO2012Malaysia, September 3-4, 2012.



Prof. Dr. Adi Maimun bin Hj Abdul Malik obtained his B.Sc in Naval Architecture from the University of Strathclyde, Glasgow in 1983. He joined Universiti Teknologi Malaysia (UTM) as a tutor the same year. He later return to Strathclyde University for his Master's and Ph.D. in Marine Technology and obtained his degrees in 1985 and 1993 respectively.

He is currently serving as Professor of Naval Architecture at the Dept. of Aeronautics, Automotive and Ocean Engineering, Faculty of Mechanical Engineering. He was Head of Marine Laboratory (1986-1989), Head of Panel for Marine Technology (1999-2000), Head of Department for Marine Technology (2000-2007) and the Deputy Dean (Development) for the Faculty of Mechanical Engineering (2007-2011), Head of Marine Hydrodynamics Research Group for Transportation Research Alliance, UTM (2010-present).

Dr. Adi Maimun specializes mainly in the field of Marine Vehicles/Structures Dynamics using CFD, Time domain simulations and experimental work. He had taught, conducted research and consultancy work in the said field and had published over 50 papers in conferences and journals.

He is currently a Fellow Member of the Royal Institution of Naval Architects (UK) and a Chartered Engineer (UK). He had served as committee member for number years for the Malaysia Joint Branch of Royal Institution of Naval Architects (RINA) and Institute of Marine Engineers Science and Technology (IMarEST). In the international field he had served in the International committee board for the conferences of MARTEC (since 2002), APHYDRO (since 2002) and OMAE (2008). Local organizing chairman for MARTEC 2004 and APHYDRO 2010 and IMSE2013. Session chairman for WSEAS13 (2-4 April 2013, Kuala Lumpur).