International Journal of Mining, Metallurgy & Mechanical Engineering (IJMMME) Volume 3, Issue 3 (2015) ISSN 2320–4060 (Online)

Numerical Simulation for Stern Flow Characteristics of Twin Skeg Container Vessel

A. Maimun, Y.M. Ahmed, Zulhazazi, Sunarsih and Mehdi Nakisa

***Abstract*—**This paper employs computational tools to examine the propulsive improvement of a twin skeg container ship by observing the streamline pattern and wake distribution of propeller plane. A single skeg container ship was used for the study as a mother ship. The mother ship was then modified to a twin skeg ship. Arrangement of the twin skeg is based on the distance between skegs and angle of the skeg. Based on the skeg flow analysis, an optimize twin skeg arrangement was selected. RANSE code Ansys CFX fitted with Shear Stress Transport (SST) turbulence was used to carry out the simulation. Grid generator ICEM CFD was used to build hybrid grids for RANSE code solver.

***Keywords*—**Container ship, CFD, propulsive improvement, twin skeg.

I. Introduction

SHIP efficiency and fuel cost are inherently key considerations for ship owners and ship designers. Improvement of ship efficiency will lead to a better environmental control as well as fulfillment of the classification society requirement. Twin skeg design which increases propulsive efficiency is advantageous from the point of view of hydrodynamic and thus becoming an alternative design.

In the study of hull form design and ship flow analysis, computational method has been widely used to obtain preliminary result prior to validation through experiment. Solely depending on the result of experiment might be reliable but it consumes extensive time and cost. Using CFD, local flow characteristic which is difficult to obtain during model test will be readily generated through such method.

Computational method is applicable to analyse ship resistance, propulsion and manoeuvring. Among past researches that utilise CFD are assessment and prediction of ship-propulsion performance [1], [2] and analysis of bare hull, rudder and propeller [3].

In this work, flow analysis of hull form design of container ship is presented using CFD code (CFX). This study emphasized the skeg arrangement to achieve a propulsive improvement. Self-propulsion performance was observed when the geometrical variation of skeg distance and angle were applied on the ship stern. Through investigation, an optimized skeg arrangement design was determined.

It was assumed that each skeg variation was operated by the same dimension of propeller plane. By neglecting the propeller influence in the simulation, prediction of propulsion performance was considered to be sufficient by investigating the streamline pattern and velocity distribution induced on propeller plane.

A. Maimun is with the Marine Technology Centre, Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310 Skudai, Malaysia (e-mail: adi@fkm.utm.my).

Y.M. Ahmed, is with the Marine Technology Centre, Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310 Skudai, Malaysia (e-mail: yasser@mail.fkm.utm.my).

Zulhazazi was graduated from Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310 Skudai, Malaysia (e-mail: zulhazazi@gmail.com).

Sunarsih and Mehdi Nakisa is a post graduate student at Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310 Skudai, Malaysia (e-mail: bekhairunnisa@gmail.com, nmehdi5@live.utm.my)

II. Description Of Model

The study explored single skeg container ship which is modified to be twin skeg using Rhinoceros 5.0 by referring to the skeg design of [4]. Table 1 shows the particulars of the model employed while Fig. 1 and Fig. 2 display the mother ship of a single skeg container model and a redesigned twin skeg container model at distance of 45.4 percent of breadth of midship from longitudinal centerline respectively.

TABLE 1
PRINCIPLE DIMENSION

Length	Beam	Draft	Wetted Surface Area	Volume displace
3.84 m	0.644 m	0.211 m	3.128 m^2	0.286 m^3

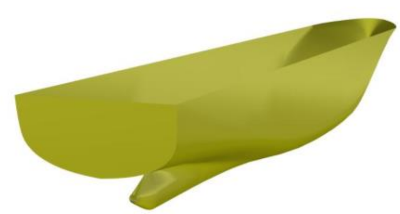


Fig. 1 Single skeg container model

Fig. 2 Twin skeg container model

III. Mathematical Model

Mathematical description of free surface flow in CFX is based on the homogenous multiphase Eulerian–Eulerian fluid approach. Both fluids (water and air) in this approach share the same velocity and other relevant fields such as temperature, turbulence, etc., which is separated by a distinct resolvable interface. The local equations governing the motion of unsteady, viscous and incompressible fluid (either liquid or gas) namely Navier–Stokes is given as

$$\frac{\partial}{\partial t}(\rho)+\frac{\partial}{\partial x_i}(\rho u_i)=0.0 \tag{1}$$

$$\frac{\partial}{\partial t}(\rho u_i)+\frac{\partial}{\partial x_i}(\rho u_i u_j)=-\frac{\partial \rho}{\partial x_i}+\frac{\partial}{\partial x_i}\left(-\rho\overline{u'_i\, u'_j}\right)$$

$$+\frac{\partial}{\partial x_i}\left[\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}-\frac{2}{3}\partial_{ij}\frac{\partial u_l}{\partial x_l}\right)\right] \tag{2}$$

where

$$\left(-\rho\overline{u'_i u'_j}\right)=\mu_t\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)-\frac{2}{3}\left(\rho k+\mu_t\frac{\partial u_i}{\partial x_i}\right)\partial_{ij} \tag{3}$$

And

$$\rho=\sum_{\alpha=1}^{2} r_\alpha\rho_\alpha\,;\ \mu=\sum_{\alpha=1}^{2} r_\alpha u_\alpha\,;\ \sum_{\alpha=1}^{2} r_\alpha=1 \tag{4}$$

The SST turbulent model was used in CFX code formulated by

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left(\Gamma_k\frac{\partial k}{\partial x_j}\right)+G_k-Y_k \tag{5}$$

$$\frac{\partial(\rho\omega)}{\partial t}+\frac{\partial}{\partial x_t}(\rho\omega u_i)=\frac{\partial}{\partial x_j}\left(\Gamma\omega\frac{\partial k}{\partial x_j}\right)+G_\omega-Y_\omega+D_\omega \tag{6}$$

where Γk and Γω, Gk and Gω, and Yk and Yω respectively represent the effective diffusivity for k and ω, generation of turbulence kinetic energy and ω due to mean velocity gradients, and dissipation of k and ω due to turbulence respectively. Meanwhile Dω represents the cross-diffusion term.

IV. Computational Grid

A commercial code, RANSE (ICEM CFD) was used to generate hybrid mesh and the distance of the first grid point of the ship surface was maintained for each case, y+ ≈ 8 which is within a log-law region. The computational domain of the model which is extended to 1.5L forward, 2.5L backward, 1.5L aside and 1.2L downward under the keel of the ship model has been meshed with unstructured tetrahedral mesh elements. The air layer was extended to 0.125L above the still water surface [5]. Fig. 3 displays the computational domain used in the simulation.

Smaller element size was used for the container model and unstructured tetrahedral was built in the region around the model. The smaller the size of element used the higher the model element number produced. Furthermore, small size of element used on and near the ship hull surface is designate to obtain a better result of velocity distribution on propeller plane. In the meantime, prism layer was built around the model with a total of five layers to produce better free surface effect. Fig. 4 depicts grid of the unstructured tetrahedral and prism layer of the model.

V. Computational Method And Boundary Condition

The current study considered the flat bottom as sea bed without natural irregularities and excluded the dynamic mesh hence treated the ship model without propeller influence. Attention was given only to starboard side due to symmetric properties of the ship hull. Computation was run on an i7 processor, 4 core hyper threading, 3.6GHz and 16 GB RAM with the effect of free surface and surface without wave and current. The water region width was set to fourth of ship lengths as to eliminate the wall effect and no-slip boundary condition was imposed on the hull surface. The static pressure and initial location of the free surface were accordingly defined by the function of water volume fraction at the outlet boundary and volume fraction of water and air at both inlet and outlet boundaries. The scalable wall function was used with turbulence model while the reference pressure was set to the atmospheric pressure. The flow was considered steady in ANSYS CFX calculations with utilisation of finite volume method and high resolution numerical scheme for discretization process and advection terms respectively. The pressure and velocity were correspondingly interpolated using linear and trilinier numerical schemes. Finally, root mean

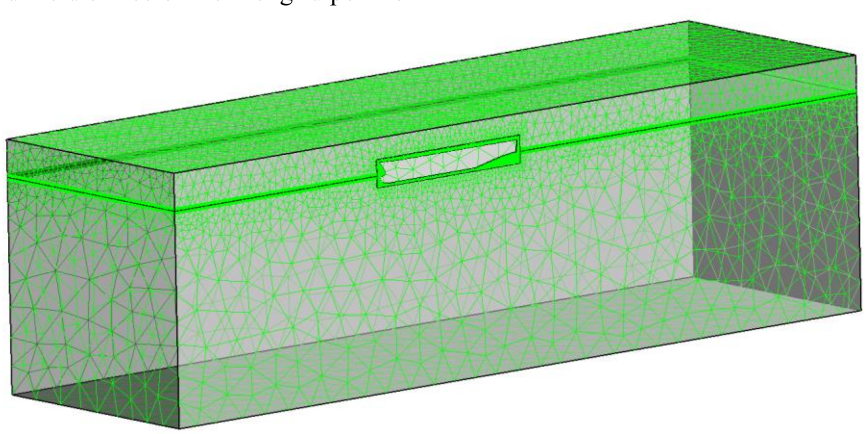


Fig. 3 Computational domain used in the simulati

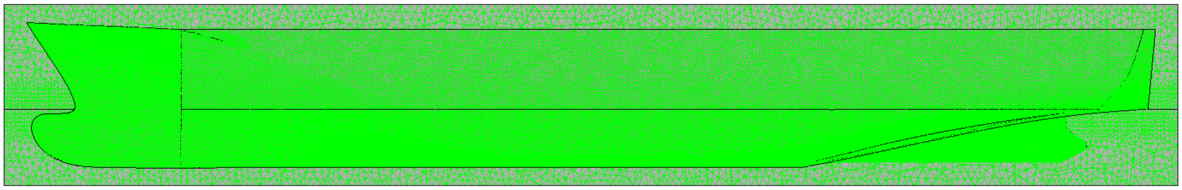


Fig. 4 Grid of unstructured tetradehral and prism layer on ship model

square (RMS) criterion with a residual target value of $1x10^{-05}$ was used to check the convergence of the solutions.

VI. Variation Of Skeg Arrangement

Two design parameters in stern arrangement were selected for variation i.e. distance between two skegs and the inclination angle. These parameters were used to investigate the flow pattern in the skeg stern area that may impact the self-propulsion performance of the ship model.

Stern flow characteristics of three different models namely Ship A, Ship B and Ship C were investigated where position of skeg for each ship model was listed in Table 2. The distance between the two skegs was specified to be 34.5 percent from ship centreline for first model and increased accordingly about 5 percent for the following models.

TABLE II
Skeg Distance Of The Ship Model

Ship	Percentage of skeg distance from centreline/ship half breadth
A	34.5
B	40
C	45.5

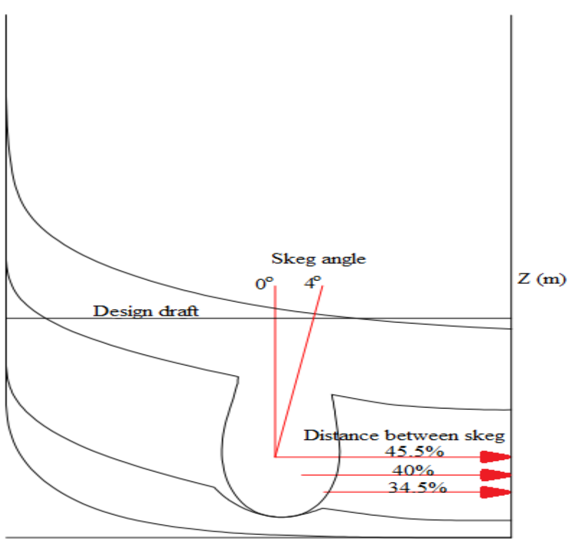


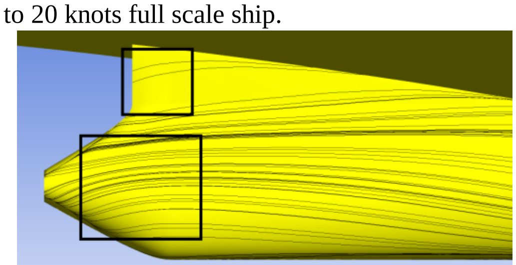
Fig. 5 Variation in stern skeg arrangement

For the skeg inclination angle, the ship model indicating the best flow characteristic was then modified by making an inclination angle to the neutral position of the skeg which is about 7 degree. Fig. 5 provides the schematic view of the variation in stern skeg arrangement applied to the model.

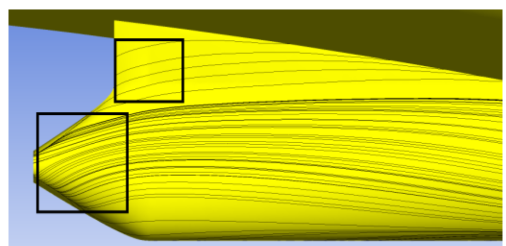
VII. Results

A. Effect of the Skeg Distance on the Stern Flow

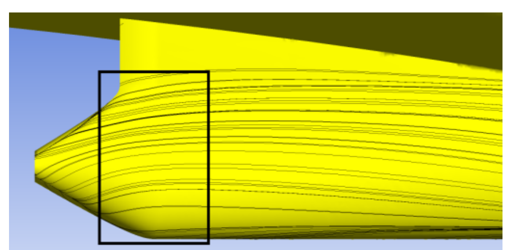
Numerical flow analysis of the model scale ships was performed and computational data was acquired using RANSE code. All models were run at model speed of 0.154 m/s which equals to 20 knots full scale ship.



Ship A



Ship B

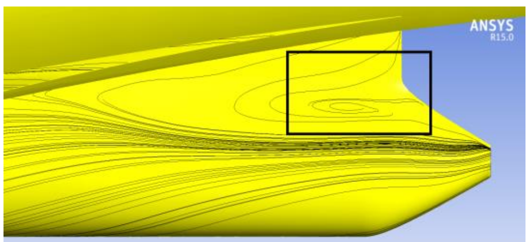


Ship C

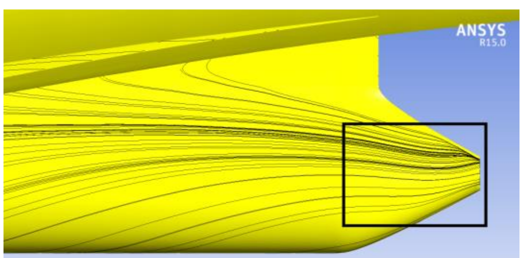
Fig. 6 Streamline pattern at outer skeg surface

Numerical streamline pattern on the outer and inner part of the skeg surface are presented in Fig. 6 and Fig. 7, respectively. As shown in Fig. 6, the flow pattern of all models was likely moving towards the propeller plane without twisting the streamline. Boxed region of Ship C indicates almost consistency in flow pattern at the stern bulb. Smooth distribution as shown by Ship C results in smaller dead water area. On the contrary, Ship A shows inconsistency in its flow pattern where the flow tends to dense at the stern bulb but less at the upper part. Differently, flow line at the upper part bulb of Ship B is straightened out. From the point of view of ship self-propulsion, it can be thus deduced that streamline distribution over the outer sides of the stern skeg for each models is to be straightforward since it implies less flow separation in the stern area.

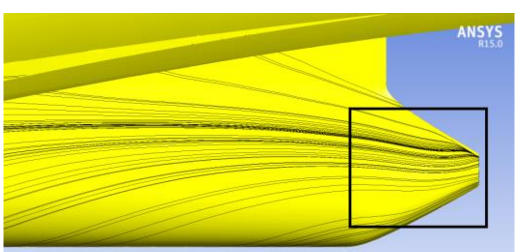
While for the inner side of the stern skeg, by referring to Fig. 7, it can be seen that the streamlines along the upper part of the stern bulb of Ship A which possesses the smallest skeg distance shows twisting pattern and reverse flow which indicate large flow separation in the zone. This separation and the generated distorting flow over the stern bulb lead to a negative impact on the propeller which is reducing the incoming water velocity into the propeller plane. On the contrary, as observed in the boxed area of Ship B and Ship C, flow lines go down towards the shaft hole without twisting. Summarily, the skeg distance applied to these two models improves the propulsion efficiency as both can accelerate the flow into the propeller plane.



Ship A



Ship B



Ship C

Fig. 7 Streamline pattern at inner skeg surface

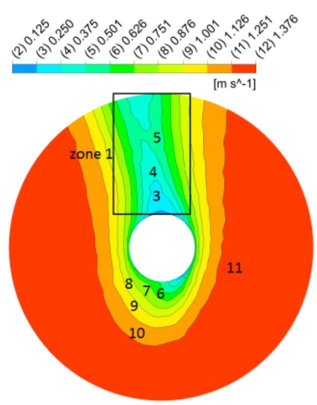
In addition to the interrogations onto the flow characteristics in the stern skeg area, wake distribution on the propeller plane was also investigated. Table 3 summarizes the advance velocity Va and wake fraction w of the three ship models. Keeping the ship speed constant at 1.454 m/s, the wake fraction values were compared among the models. The values are important as to determine the interaction between the ship hull and the water inflow velocity. Wake fraction formula employed is based on Taylor expression as follows.

$$w = \frac{v_s - v_a}{v_s} \tag{7}$$

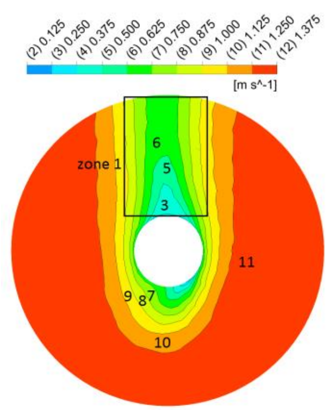
According to the CFD code result, numerical flow characteristics in the stern region of the ship models are as demonstrated in Fig. 8. The figure represents the computational wake distribution in the starboard propeller plane of Ship A, Ship B and Ship C respectively. The velocity distributions presented in the figure are important as to compare the flow characteristics among the three skeg distances.

TABLE III
WAKE FRACTION OF THE SHIP MODEL

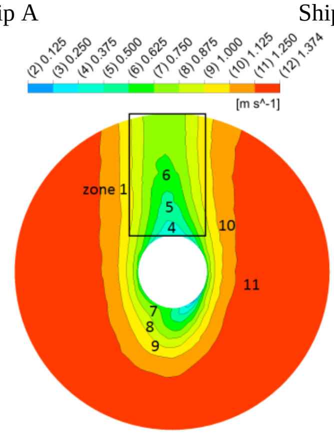
Ship	Vs	Va	Vs-Va	w
A	1.454	1.125	0.329	0.226
B	1.454	1.142	0.312	0.214
C	1.454	1.151	0.304	0.209



Ship A



Ship B



Ship C

Fig. 8 Distribution of computational velocity in the propeller plane

The figure tells that the wake distribution of Ship A is unbalance as for zone I the flow tends to move away to the top left hand side. This can be explained by relating the flow characteristic of the inner skeg surface of Ship A where the streamline at the top part goes backward (reverse flow) as well as due to some flow separation occurred. By comparing zone I of the three ship models, Ship C exhibits higher velocity flow which is represented by the colour contours.

B. Effect of the skeg angle on the stern flow

Another Skeg angel of 4 degree was modified from the selected candidate Ship C which original skeg designed without any angle. The new 4 degree skeg angle ship was assigned as Ship D. Applying the same procedure, explorations were conducted to the new assigned ship and the flow pattern and wake distribution were then compared with Ship C. The result of the wake distribution is displayed in Fig. 9.

The figure indicates that the velocity of wake distribution of the two planes shows less difference. Furthermore, as the wake value of the original selected designed Ship C is lower than the modified Ship D, the earlier was absolutely won over the latter and thus is finally selected as the best design candidate.

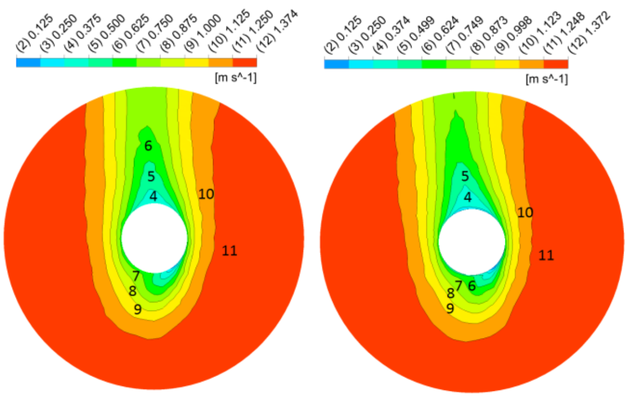


Fig. 9 Distribution of computational velocity into propeller plane of Ship C and Ship D

C. Comparison between single skeg and optimized stern skeg

For the purpose of such comparison, Ship C was chosen to be compared with the mother ship which is the single skeg container ship assigned as Ship E. Table 2 provides the wake fraction of the ships assessed. It is clearly shown that Ship C possesses lower wake value with higher water inflow velocity.

TABLE IV
WAKE FRACTION FOR SHIP C AND SHIP E

Ship	Vs	Va	Vs-Va	w
C	1.454	1.151	0.304	0.209
E	1.454	1.037	0.418	0.287

Figure 10 finally compares the velocity distribution on the propeller plane of twin and single skeg ship models of Ship C and Ship E respectively. As displayed by the figure, at zone 11 Ship C indicates wider area of high velocity. However, comparing the top part of the propeller plane, Ship E shows low of flow velocity especially in region 4 and 5 while ship C shows a better flow of velocity in region 6 and 7 about 0.750 m/s.

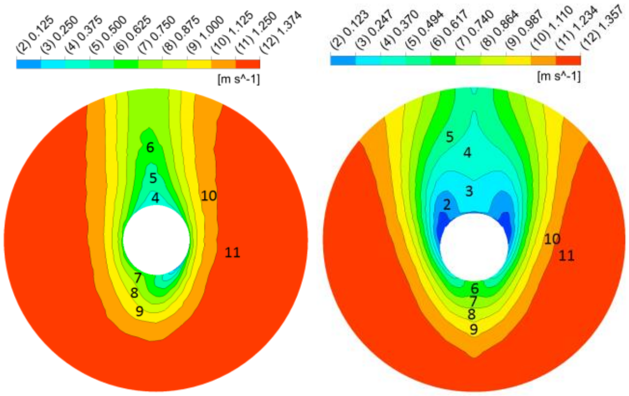


Fig. 10 Distribution of computational velocity into propeller plane of Ship C and Ship E

VIII. CONCLUSION

Two hydrodynamic characteristics that can be effectively used to measure propulsive performance are straightforward flow lines along the stern bulb over the inner and outer skeg surface and the wake fraction of the propeller plane. Numerical analysis has been extensively used in the assessment of the complicated ship stern flows that impact the propulsive efficiency of twin skeg ships.

Effects of the variation of skeg distance and angle on the behaviour of stern flow have been investigated to optimize the stern hull form of twin skeg ships in viewpoint of self-propulsion performance. Investigation was started with single skeg container ship which was modified to be twin skeg. The streamline and wake fraction were then compared among the variation of skeg distance.

Finally streamline and wake fraction of the selected twin skeg model was compared with single skeg one through the same computational method. The selected twin skeg model shows better wake fraction about w=0.209. Minor change in stern hull form of the twin skeg model appears and although it is a small fraction compared to the total hull geometry it might result in critical change in stern flow and propulsion performance.

REFERENCES

[1] J. Kim, I.R. Park, K.S. Kim, and S.H. Van, "RANS computation of turbulent free surface flow around a self-propelled KLNG carrier", J SNAK, vol. 42, no. 6, pp. 583-592, 2005.

[2] [2] H.T Kim., H.T Kim, and S.H. Van, "Numerical analysis of flow characteristics of a twin-skeg container ship with variation of stern hull shape", J SNAK, vol. 44, no. 6, pp. 551–563, 2007.

[3] [3] J.J. Kim, K.J. Paik, Y.C. Lee, and H.T. Kim, "A numerical study of turbulent viscous flow around a self-propelled ship", in Proceeding of the annual spring meeting SNAK., 2006. pp 548–557.

[4] [4] J.E. Choi, K.-S. Min, J.H. Kim, S.B. Lee, and H.W. Seo, "Resistance and propulsion characteristics of various commercial ships based on CFD results", Ocean Engineering, vol. 37, pp. 549-566, 2010.

[5] [5] Y.M. Ahmed. "Numerical simulation for the free surface flow around a complex ship hull form at different Froude numbers", Alexandria Engineering Journal, vol. 50, pp. 229-235, 2011.

A. Maimun is a professor at Universiti Teknologi Malaysia. He 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.

Y.M. AHMED is a Senior Lecturer at Universiti Teknologi Malaysia. He gives lecture and conduct research in the field of Hydrodynamics, Dynamic Marine Vehicle, Sea Keeping and Ship Resistance Propulsion. He is specialize in computational fluid dynamic using various computational method.

Zulhazazi was graduated from Universiti Teknologi Malaysia in Mechanical Engineering-Marine Technology. He is currently a research assistant at Marine Technology Centre

Sunarsih is a Post Graduate Student at Universiti Teknologi Malaysia. She is currently studying marine propeller hydrodynamics towards development of manoeuvring model in restricted waterways. She has experience in the area of Marine Propulsion Plant, Marine Engineering and Systems, and Fuzzy Multi Criteria Decision Making Method.

Mehdi Nakisa is PhD student in Faculty of Mechanical Engineering-Universiti Teknologi Malaysia(UTM). He is currently studying on propeller performance and ship hydrodynamics characteristics in different water ways geometries numerically and experimentally as a member of hydrodynamic research group in marine technology center (MTC) under Prof. Dr. Adi Maimun. His experience is hydrofoil catamarans, propeller performance and ship maneouvrability in restricted waters using Ansys commercial code simulation.