

Ship Modelling and Analysis Using Open Source Computer Aided Engineering (CAE) Part 2 : Resistance Prediction of Passenger Ship Hull Form Model Using OpenFOAM

Ummu Saiyidah Najihah binti Zainudin*, Abu Hasan Abdullah and Nasrudin Haji Ismail

School of Mechanical Engineering, Faculty Of Engineering,
Universiti Teknologi Malaysia, 81310 UTM, Johor Bahru

*Corresponding email: ummusaiyidah95@gmail.com

Article history

Received
6th September 2020
Revised
5th April 2021
Accepted
1st June 2021
Published
14th August 2021

ABSTRACT

The prediction of hull resistance has become an important part in the ship design industry. There is also of interest to investigate the wave pattern of the hull to get desirable resistance properties. To make an accurate prediction of hull resistance is very essential because it related to choose an appropriate engine power of the propulsion unit. Thus, this research is to predict the performance of passenger ship hull model (MTC 092) which developed by Marine Technology Centre's (MTC) by simulating wave patterns around this hull form. Secondly is to analyze the computed flow field which is expected to yield hull resistance and free-surface wave pattern which will be compared with the model testing results. Initially, a conceptual framework study of the predictions of wave pattern and hull performance by using OpenFOAM was done from previous research. Next, the passenger ship hull model is modelled by using FreeShip Plus. Then, the model is exported into the OpenFOAM by undergoing the resistance simulation to determine the wave pattern and resistance value respectively. The result in terms of hull performance is focusing on the resistance part which will be analyze using $k-\omega$ SST method and it will be compared with the resistance experiment result. The wave pattern of passenger ship hull model in simulation will be observed specifically on plan view. In conclusion, the contribution of this study be able to increase the type research by using OpenFOAM from previous study. The suggestion for ship design industry is that the using of computational fluid dynamics (CFD) toolbox in predict of the hull form performance shall be expanded in Malaysia so that they could improve their vessel design process, reduce the production time and cost.

Keywords : *Free-surface, Wave pattern, OpenFOAM, Hull Resistance, Ship hull form.*

1.0 INTRODUCTION

Shipbuilding industry in Malaysia lacks in advanced capability, especially in ship design and hull performance area. Over the decade, Jasak (2009), Yang et al. (2009), Kim et al. (2010) and Seo et al. (2010) contend that the advancement of computational analysis of ship hull performance by using computational fluid dynamics (CFD) is being widely adopted and the results produced are being applied to the actual design of ship in shipbuilding industry. However, ship design and hull performance area is concurred by commercial software whose demand a high initial cost and high maintenance cost. As a result of that, small and medium

sized shipbuilding enterprises (SME) in Malaysia was burdened with limited cost and budget because of this situation.

Park et al. (2013) said owing to the high cost of commercial software packages, interest in free computational fluid dynamics (CFD) codes has recently increased. Due to the reason, the idea of this study is to use fully packaged of Open Source Software (OSS) in evaluating and optimizing the ship hull form and the study of its generated wave patterns. As the motivation for using Open Source or free CFD is primarily due to the cost and time related. It is crucial because the CFD process can be sufficiently accurate as experimental, but low in computational cost and total time taken.

The above description reflects the aims of this study which are to develop multiple work flow of integrating open source software modules in ship performance analysis so that an efficient work flow of hull development solution can be determined. Besides that, to determine the potential workflow set of open source software for hull form design and evaluation on selected number of model and to build a database that will archive the details of ship models at Marine Technology Centre (MTC) and keeping relevant design parameters which have been studied, analyzed and simulated by expert user who draw on open source solution.

In order to achieve the objective of this study, an integration of alternative workflow sets of open source software platform will be done in designing, meshing, developing and testing the hull ship which is a complete process for any ship design and hull performance test. The key of workflow set in this integration process is the freedom to incorporate the hull form design and hull performance test with Marine Technology Centre's (MTC) which own 20- year collection of model testing data. This requires the ability to access the code and compare them with the integration results.

The expected results of this research are an effective workflow complete packaged of hull development which is passenger ship model and resistance performance analysis is determined. After that, the potential workflow module sets of open source software for hull form design and evaluation on selected number of model are determined. Lastly, a database that will archive the details of ship models at Marine Technology Centre (MTC) is build.

2.0 THEORY

2.1 The Volume of Fluid Method

The volume of fluid method is used to approximate free boundaries in finite difference numerical simulations. Hirt and Nichols (1981) said this method is simple but powerful and shown to be more flexible and efficient than other methods for treating complicated free boundary configurations. Meanwhile, Sugalski (2014) state that a volume of fluid is a numerical technique to track and localize the interface of different fluids in CFD.

In this volume of fluid method, a phase fraction which is used to determine the amount of each fluid of the cells. In this case study Hirt and Nichols (1981) and Sugalski (2014) shared the same thought which is floating object at sea, the cells that containing water will have an α -value equal to 1 and those that only contain air will have α -value equal to 0. Sugalski (2014) mentioned the volume, phase fraction equation for the phase fraction α is written as

$$\frac{\partial \alpha}{\partial t} + \nabla(\alpha V) = 0 \quad (1)$$

where t is time, V is velocity.

After that, to determine the density of the air or water mixture within each cell using Equation 1, the phase fraction α is used to solve the Navier-Stokes equations written as below,

$$\rho = \alpha\rho_i + (1 - \alpha)\rho_g \quad (2)$$

$$\frac{\partial\alpha}{\partial t} + \nabla(\alpha V_i) = 0 \quad (3)$$

$$\frac{\partial\alpha}{\partial t} + \nabla((1 - \alpha)V_g) = 0 \quad (4)$$

$$V = \alpha V_i + (1 - \alpha)V_g \quad (5)$$

where ρ_i is the density of water and ρ_g is the density of air. Term of i, g represents liquid and gas phase.

An example of two fluids divided using the volume of fluid method is shown in Figure 1, where the water filled cells with $\alpha = 1$ are red and the ones with $\alpha = 0$ are blue. Figure 1 also shown the water level or α -value of the surface cells. However, Refvik (2016) contend the VOF method only calculates an approximation of the free-surface, not an exact water level.

The red zone represents $\alpha = 1$ and red zone $\alpha = 0$, while the blue surface and white areas with $0 < \alpha < 1$ represent interface.

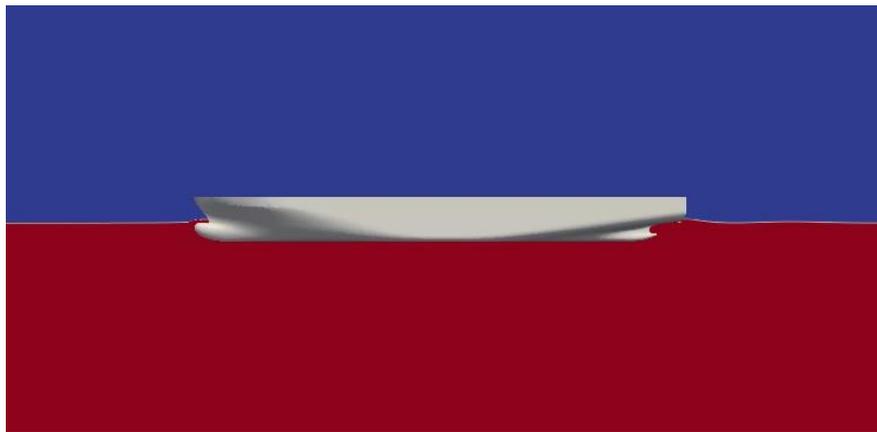


Figure 1: Visualization from paraview of the phase fraction close to a ship model.

2.2 The Turbulence Models

The two equation turbulent models, the standard k - model and the k - model are the most commonly used in naval architecture. In these two equation models, Frisk and Tegehall (2015) state that the transport equations for the turbulent kinetic energy and its dissipation, s or ω , are used to obtain the turbulent viscosity.

According to Wilcox (2008), the standard $k - \omega$ model is good equation in prediction free flows with small pressure gradients, but limits its applicability to high Reynolds number flows. Other than that, Moore (2010) has been observed that the standard $k - \omega$ model cannot be used to describe the wake behind a moving hull accurately. Meanwhile based on Frisk and Tegehall (2015), the $k - \omega$ model is valid in the low turbulent Reynolds number region close to walls, however the $k - \omega$ model is sensitive to the choice of boundary conditions and initial conditions.

At the hull surface and in the free stream outside the boundary layer Axner (2014) said that both of turbulence models should be used together to improve prediction of the principal shear stresses which called $k - \omega$ SST model. After that, Frisk and Tegehall (2015) figure out that $k - \omega$ models have given results that differ from the experimental result and have been proved that the SST $k - \omega$ model has shown a good result for many types of complex flows.

2.3 Boundary Conditions

The boundaries of the computational domain used in the study are illustrated in Figure 2.

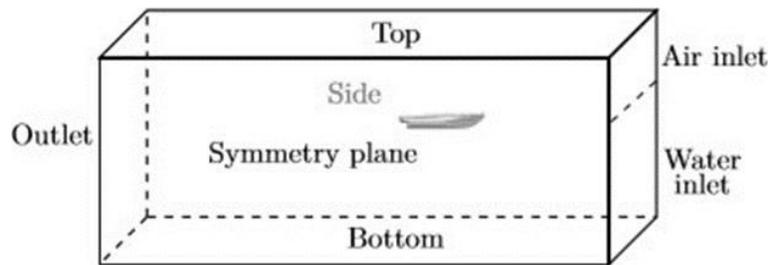


Figure 2: Boundaries of the computational domain.

At the inlet which located in front of the hull Frisk and Tegehall (2015) said the velocity of incident air and water was set to the hull speed and the turbulent flow variables were set by specifying values of the turbulent intensity and turbulent viscosity ratio. After that the outlet located behind the hull was set to a pressure outlet. The boundary conditions are,

$$Top : \frac{\partial U}{\partial n} = 0, p = 0, \alpha = 0 \quad (6)$$

$$Hull : U = 0, \frac{\partial p}{\partial n} = 0, \frac{\partial \alpha}{\partial n} = 0 \quad (7)$$

$$Inlet : U = U_{\infty}, p = 0, \frac{\partial \alpha}{\partial n} = 0, \alpha_0^1 \quad (8)$$

$$Outlet : \frac{\partial U}{\partial n} = 0, p = 0, \frac{\partial \alpha}{\partial n} = 0 \quad (9)$$

where n is the outer normal vector and p is the pressure.

The computational domain was built as a rectangular block around the hull in deep water. A large domain was created in order to avoid effects from the domain boundaries to affect the flow near the hull. Figure 3 shows the dimensions agree well with the minimum recommendations of ITTC. In Figure 3, the computational domain is illustrated and its dimensions are expressed in terms of the overall hull length, L_{OA} .

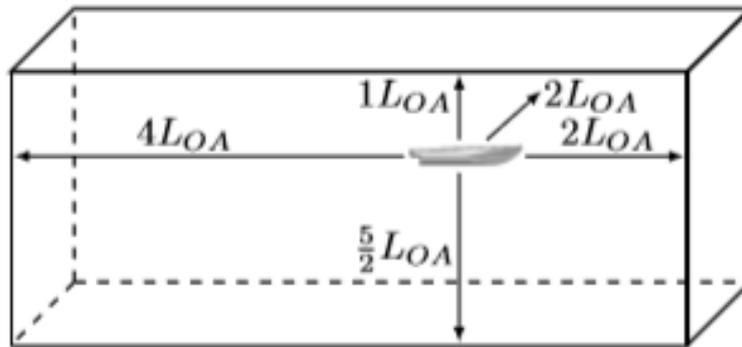


Figure 3: Dimensions of the computational domain (ITTC).

3.0 THE TURBULENCE MODELS

3.1 FreeShip Plus

FreeShip Plus is an open source software used in surface modeling program for the design of ships. However, this FreeShip Plus software designed for only thin hull ship type. Ueng et al. (2008) thin ship hull can be defined based on their lengths (L), widths (B), drafts (D) and the wave length (λ) of the ship. The thin hull ship shapes satisfy the following constraints:

$$B \ll L \quad (10)$$

$$\frac{D}{\lambda} = \frac{D}{L} \quad (11)$$

FreeShip Plus software feature modelling a ship surface. Besides that, FreeShip Plus also can act as a post-processing since it can calculate ship resistance, design hydrostatics and etc. After that, table 1 below is the list of file types that can be imported or exported in FreeShip Plus software which can be used in the process of import or export the model.

Table 1: The list of file types that can be used in import and export process.

File Type	Import	Export
Part	√	√
Carlson .hul file	√	
Import/Export .fef file	√	√
Surface	√	
Chines	√	
VRML	√	

PlyCad files	√	
Michlet waves	√	√
IGES		√
DXF 3D mesh		√
DXF 2D polylines		√
DXF 3D polylines		√
STL file		√

Lastly, FreeShip Plus software allows the user to view the completed format of a lines plan of the ship. The lines plan of the ship can be viewed in two different modes which are wireframe mode and the filled mode as shown in Figure 4 below:

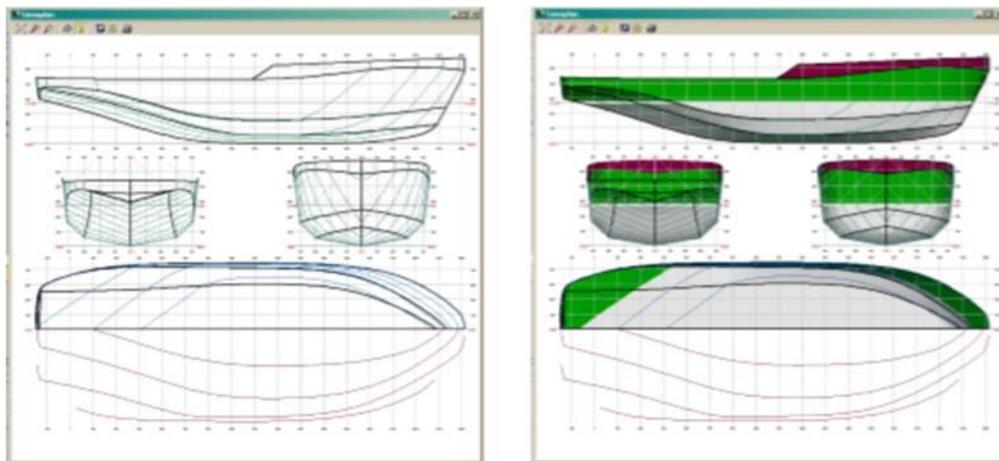


Figure 4: The lines plan of the ship in FreeShip Plus (FreeShip Plus Manual).

3.2 OpenFOAM

System directory files contained the controlDict text file which there are entries regarding the simulation settings. The simulation settings that control for what time step to start and stop the simulation. Besides that, Jasak et al. (2007) stated the settings also control time step to write the temporary results also known as time dumps. This control setting is a steady state solver which refers to number of iterations instead of seconds of simulation time. Once the solution is converged, the data can be used for analysis.

However, when the simulation starts OpenFOAM produces information regarding the solving of different equations which can deduce whether the solution has converged or not. The first information is the initial residual which the initial residue should be approached zero when the solution is convergence. According to Jasak et al. (2007), second is the final residual is always remained small and will always be less than the initial residual. The last one is the number iterations will show how many times the matrix was solved for the current equation. The number iteration should start to decrease once the solution approach convergence.

Mesh generation is an important part of OpenFOAM. The accuracy and convergence of the solutions depending to the mesh quality. However, OpenFoam has its own built mesh generation utility called blockMesh. BlockMesh is text based and works by defining hexahedral blocks in a local, right -handed coordinate system. This block can build more complex geometries. There is one challenge with the blockMesh which is keeping track of the multiple coordinate points within a complex geometry. This problem can be negated by using a CAD software to produce the complex geometry described with STL surfaces. OpenFoam can read these STL files and convert them into mesh as shown in Figure 5. This solution can be used as the technique or strategy for this project.

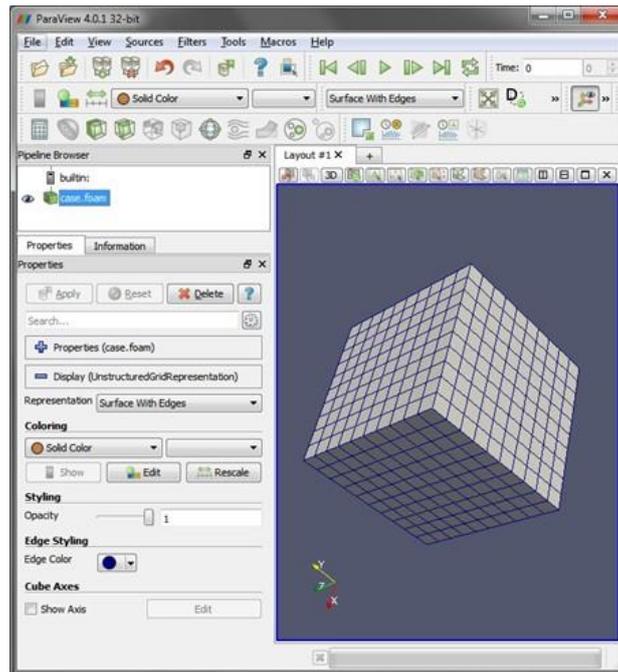


Figure 5: Mesh generation in OpenFoam.

4.0 RESULTS AND DISCUSSION

The overall simulation results are tabulated in Table 2 and 3 according to the model and total resistance differences between experimental and simulation. Based on the hydrostatic data of passenger ship hull model, the percentage of difference between the hydrostatic data from experimental and hydrostatic data from FreeShip Plus was calculated. The differences between these two data as shown in table 2 below:

Table 2: Ship hydrostatic data particular difference from real model.

Hydrostatic Data	Unit	Ship Model	FreeShip Plus Model	Differences (%)
Displacement	<i>t</i>	0.0944	0.094	0.00
Wetted Surface Area	<i>m</i> ²	1.5050	1.470	2.33
WL length	<i>m</i>	2.7400	2.816	2.77
Beam	<i>m</i>	0.5130	0.511	0.39

After that, OpenFOAM was calculated the forces components acting on passenger ship hull model, namely pressure and viscous. Total between these two forces is total resistance. The ship model was simulated following the experimental data and compared with OpenFOAM resistance result. For passenger ship hull model OpenFOAM - resistance case, the data obtained by OpenFOAM simulations at different velocities are shown in table 3 below. Meanwhile, figure 6 show the graph of total resistance by OpenFOAM and experimental data for passenger ship hull model.

Table 3: Resistance difference from OpenFOAM software and Towing Tank experiment.

Froude Number (F_n)	Vs (knot)	Towing Tank (KN)	OpenFOAM (KN)	Differences (%)
0.38	14	30.77	35.00	13.750
0.43	16	43.73	45.40	3.820
0.48	18	57.44	57.44	0.157
0.54	20	65.31	69.65	6.650
0.59	22	73.30	84.80	15.690

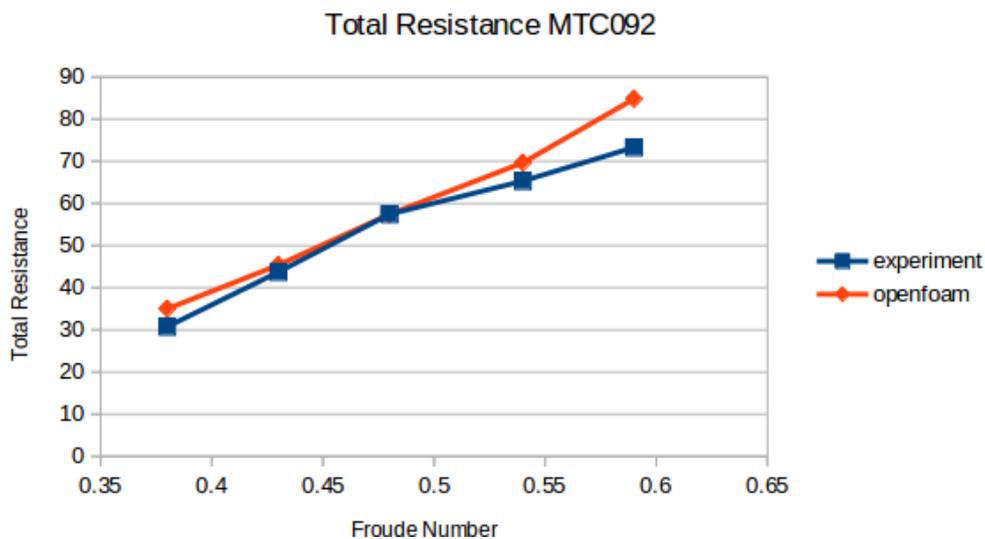


Figure 6: Total resistance for passenger ship hull model.

As shown in table 3 and figure 6, the differences total resistance of passenger ship hull model (MTC 092) between experimental and numerical simulations are close which in between range 0.157 % to 15.69 %. The curves of numerical simulation results are increasing linearly towards the Froude number but as can see at total resistances obtain from numerical simulation is an overestimate from experiment result.

This is because of in numerical simulation the ship model is in a static condition. The study not considering the trim of the ship as in the towing tank experiment. Based on Molland et al. (2017), trim angle is very important and it has significant influence on resistance. Meanwhile, the 2.77 and 2.22 percentage difference of level of water length and wetted surface area between experimental model and ship simulation model might affect differences of the total resistance of experimental and ship simulation model.

Next, the results of the wave pattern that visualized in OpenFOAM simulation was observed at the plan view as shown in figure 7 to figure 11 at different speed of passenger ship hull model. The figures below emphasized that as the speed of the simulation increases the amplitude and the length of the wave increase.

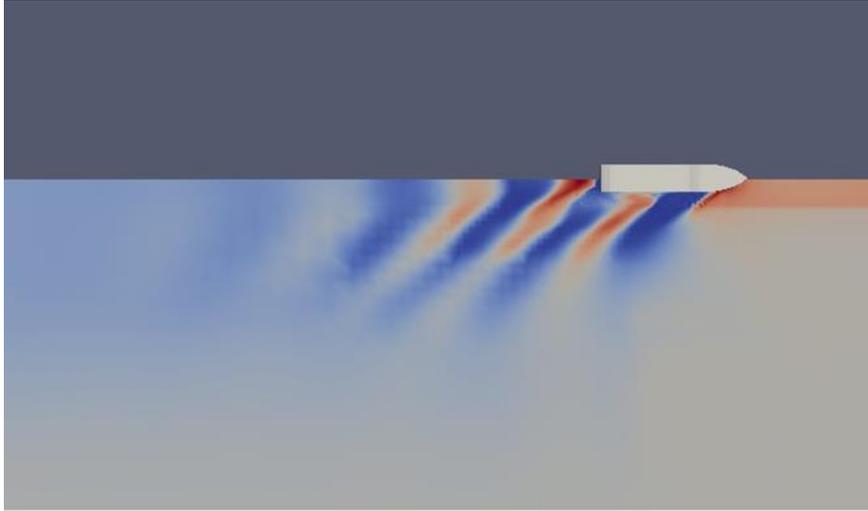


Figure 7: Wave pattern at $F_n=0.38$.

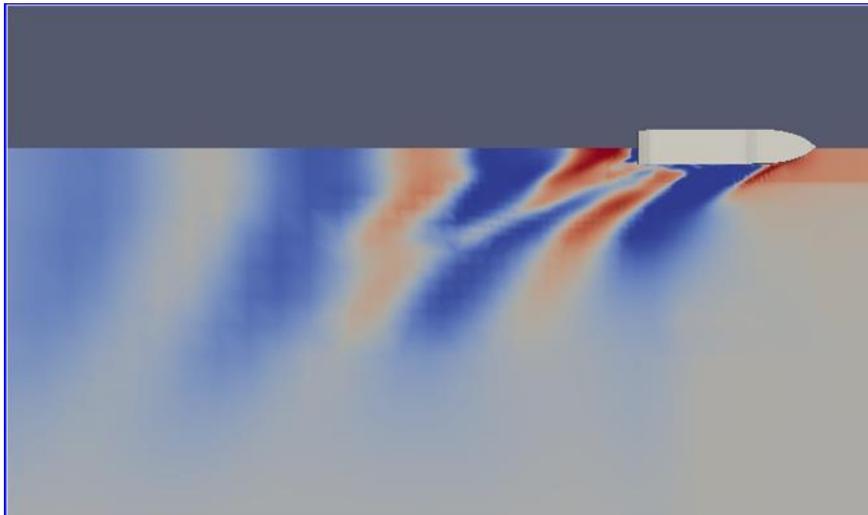


Figure 8: Wave pattern at $F_n=0.43$.

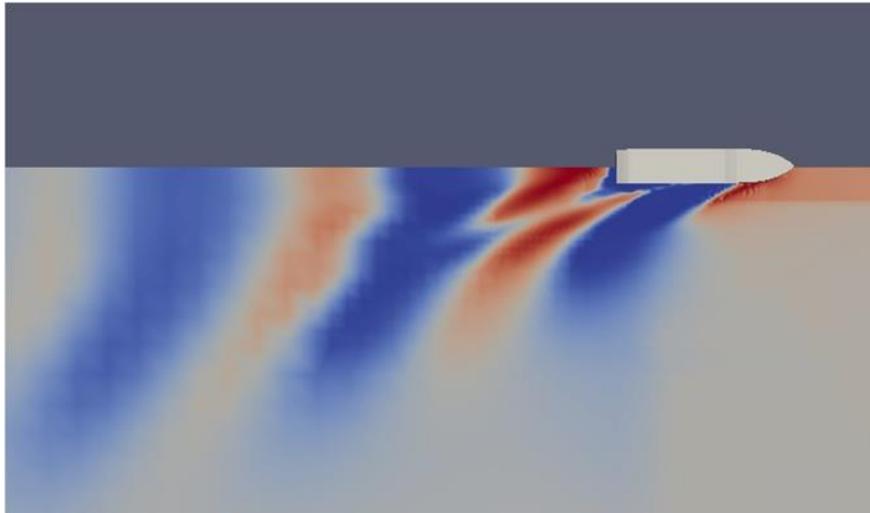


Figure 9: Wave pattern at $Fn=0.49$.

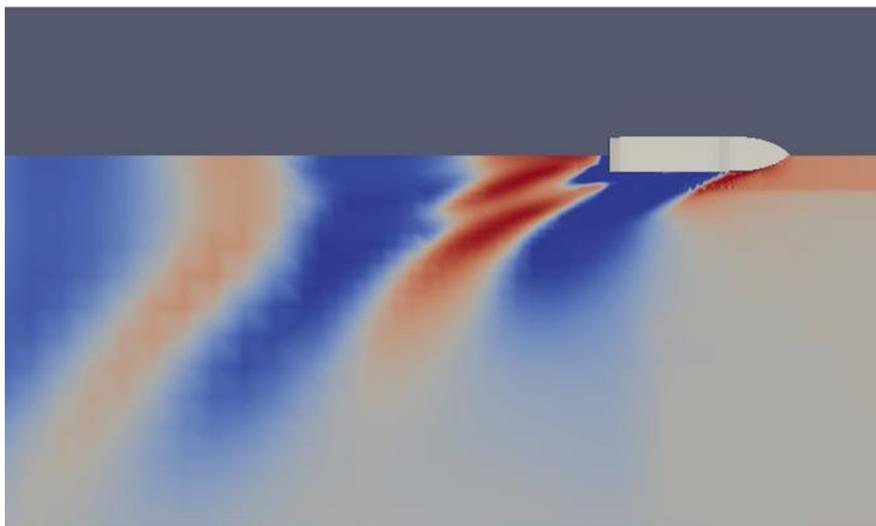


Figure 10: Wave pattern at $Fn=0.54$.

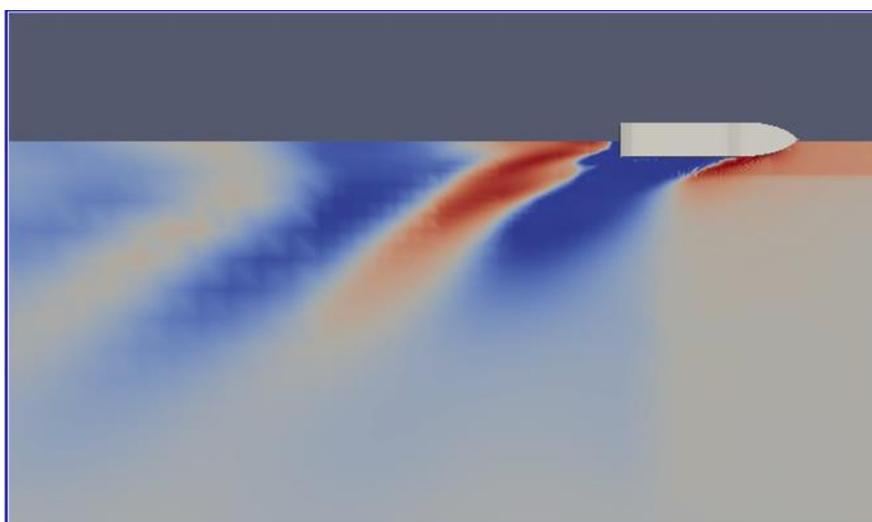


Figure 11: Wave pattern at $Fn=0.59$.

Numerical simulations using OpenFOAM are a feasible method for predicting the resistance of the passenger ship hull model, despite the difference between experimental data and numerical simulation resistance. In the study, the difference numerical simulation to the experimental result resistance is very close. Based on the analysis that have been done, the least number of difference in this simulation is 0.157 % at Froude number equal to 0.49. Meanwhile, the highest number of difference in this simulation is 15.69 % at Froude number equal to 0.59. The resistance curves of numerical simulation had increasing linearly.

These variations between experimental results and numerical simulation may be due to the difference hydrostatic data passenger ship hull model that have been modelling in FreeShip Plus compared to the experimental hydrostatic data of passenger ship hull model. Since the difference of displacement value between manual hydrostatic data and FreeShip Plus data is 2.77 % and 2.33 % for waterline length and wetted surface area, hence it is possible to affect the difference between experimental data and numerical simulation resistance result.

Lastly, the action that need to be taken for this cases are re-modelling the ship model to improve the differences of the hydrostatic data below than 1 % and then run numerical simulation of ship model with considering the trim angle, since the trim angle is very significant to the ship hull resistance.

ACKNOWLEDGEMENTS

The authors gratefully acknowledge OSCAE. Initiative effort at the Faculty of Mechanical Engineering, UTM, to raise awareness and use of open source hardware and software for Computer Aided Engineering (CAE) works and this study under full use of the towing tank vessel MTC 092 hull model at Marine Technology Centre (MTC).

REFERENCES

- [1] Axner, L., Gong, J., Chiarini, A., & Mascellaro, L. (2014). SHAPE pilot Monotricat SRL: Hull resistance simulations for an innovative hull using OpenFOAM. PRACE Partnership for Advanced Computing in Europe, 1-8.
- [2] Frisk, D., & Tegehall, L. (2015). Prediction of High-Speed Planing Hull Resistance and Running Attitude (Doctoral dissertation, Master Thesis, Chalmers University of Technology).
- [3] Hirt, C. W., & Nichols, B. D. (1981). Volume of fluid (VOF) method for the dynamics of free boundaries. *Journal of computational physics*, 39(1), 201-225.
- [4] Jasak, H. (2009). OpenFOAM: open source CFD in research and industry. *International Journal of Naval Architecture and Ocean Engineering*, 1(2), 89-94.
- [5] Jasak, H., Jemcov, A., & Tukovic, Z. (2007, September). OpenFOAM: A C++ library for complex physics simulations. In *International workshop on coupled methods in numerical dynamics* (Vol. 1000, pp. 1-20). IUC Dubrovnik, Croatia.
- [6] Kim, M. C., Park, W. G., Chun, H. H., & Jung, U. H. (2010). Comparative study on the performance of POD type waterjet by experiment and computation. *International Journal of Naval Architecture and Ocean Engineering*, 2(1), 1-13.
- [7] Moore, C. S. (2010). *Principles of Naval Architecture Series-Intact Stability*. Society of Naval Architects and Marine Engineers.
- [8] Park, S., Park, S. W., Rhee, S. H., Lee, S. B., Choi, J. E., & Kang, S. H. (2013). Investigation on the wall function implementation for the prediction of ship resistance. *International Journal of Naval Architecture and Ocean Engineering*, 5(1), 33-46.
- [9] Refvik, H. (2016). *Computational Fluid Dynamic (CFD) Study Investigating Wave Loads on a Torus at Deep Sea* (Master's thesis, Norwegian University of Life Sciences, s).
- [10] Seo, D. W., Lee, S. H., Kim, H. C., & Oh, J. K. (2010)a. A Numerical study for the efficacy of flow injection on the diminution of rudder cavitation. *International Journal of Naval Architecture and Ocean Engineering*, 2(2), 104-111.

- [11] Sugalski, K. (2014). Fishing vessel hull design and towing resistance calculation by the CFD methods. *Zeszyty Naukowe/Akademia Morska w Szczecinie*.
- [12] Wilcox, D. C. (2008). Formulation of the kw turbulence model revisited. *AIAA journal*, 46(11), 2823-2838.
- [13] Yang, J., Rhee, S. H., & Kim, H. (2009). Propulsive performance of a tanker hull form in damaged conditions. *Ocean Engineering*, 36(2), 133-144.