

PAPER • OPEN ACCESS

A Numerical Ventilation Problem on Fridsma Hull Form Using an Overset Grid System

To cite this article: Samuel *et al* 2021 *IOP Conf. Ser.: Mater. Sci. Eng.* **1096** 012041

View the [article online](#) for updates and enhancements.



240th ECS Meeting ORLANDO, FL

Orange County Convention Center Oct 10-14, 2021



Abstract submission due: April 9

SUBMIT NOW

A Numerical Ventilation Problem on Fridsma Hull Form Using an Overset Grid System

Samuel¹, D J Kim², A Fathuddiin¹, and A F Zakki¹

¹Department of Naval Architecture, Faculty of Engineering, Diponegoro University, Semarang, 50275, Indonesia

²Naval Architecture and Marine Systems Engineering, Pukyong National University, Busan, 48513, South Korea

Abstract. Numerical ventilation problem (NPV) is a problem that often occurs during the numerical simulations process. This problem mostly occurs when the vessel has a high Froude Number, causing inaccurate ship resistance predictions. Numerical ventilation can be considered as one of the major sources of error in numerical simulations that require further analysis. The present paper aims to validate a numerical resistance prediction using Fridsma's hull form. An overset grid system is implemented in this study to solve using the Volume of Fluid method. The method for improving the accuracy of ship resistance prediction are refinement method, visualization method, and phase replacement method. In this paper, the RANS equation is used to describe the turbulence model using $k-\epsilon$. On most occasions, the Volume of Fluid model uses multiphase Euler flow, assuming air and water as a phase. This research concludes that mesh refinement method is able to solve numerical ventilation problem with such a good result.

1. Introduction

Numerical Ventilation Problem (NVP) is a problem that occurs during a high-speed craft, mostly planing hull numerical simulation. A mixed-flow is generated below the hull causing inaccurate ship resistance prediction. A ship is called planing hull when the ship is mostly supported by hydrodynamic lift rather than hydrostatic, in which the Froude Number is $Fr \geq 1$ [1]. Studies regarding NVP were conducted by Avci et al. in 2018 to analyze resistance using visualization reference [2]. Furthermore, in the same year, Wheeler et al. researched to investigate resistance on planing hull and solving NVP using source term [3]. Additionally, Gray-Stephens researched in 2019 to investigate high-speed planing hull using refinement method as a way to reduce NVP [4].

Gerald Fridsma highly contributed to experimental testing of planing hull to predict ship resistance in 1969 [5]. This research is commonly used because of simple geometry and the highly accurate result so that Gerald Fridsma's experiment is used as validation reference of CFD simulation. Generally, the Navier-Stokes equation was used in CFD simulations to solve fluid flow around a planing hull. Three main methods to solve numerical simulation: FEM (Finite Element Method) and FVM (Finite Volume Method). In 2015, FEM was used in research to predict catamaran ship resistance by ignoring ship dynamic motions [6]. Meanwhile, in 2018, the same research was conducted to predict the total resistance of a bulbous bow catamaran using FEM [7]. Furthermore, in 2019 Finite Volume Method (FVM) was used in a research conducted to predict the resistance of a



high-speed craft and to see the flow of fluids in regular waves [8] considering heave and pitch motions. As a report from Yousefi et al. statistically to predict high-speed craft resistance the most widely used FVM [9].

According to a commonly experienced problem in high-speed craft, this research is conducted to reduce Numerical Ventilation Problem and providing alternative solutions to improve ship resistance predictions.

2. Method

2.1. Hull form Characteristic

This part of the paper presents the results of comparisons between CFD codes STAR-CCM+ and planing hull experiments in calm water. The numerical simulation was performed tested by Fridsma’s hull form. Table 1 illustrates the main dimension of Fridsma’s hull form. The shape of the hull form described by mathematical, as seen in Figure 1. The hull form is represented graphically by lines drawing using NURBS to obtain curves, surfaces, and volume.

Table 1. Fridsma experiment data [5]

Parameters	Unit	Value
L/B		5
L	m	1.143
B	m	0.229
T_{AP}	m	0.081
β	Degree	20
τ_o	Degree	1.569
LCG from AP	m	0.457
VCG from keel	m	0.067
Δ	Kg	10.890
$I_{yy} = I_{zz}$	Kg.m ²	0.235

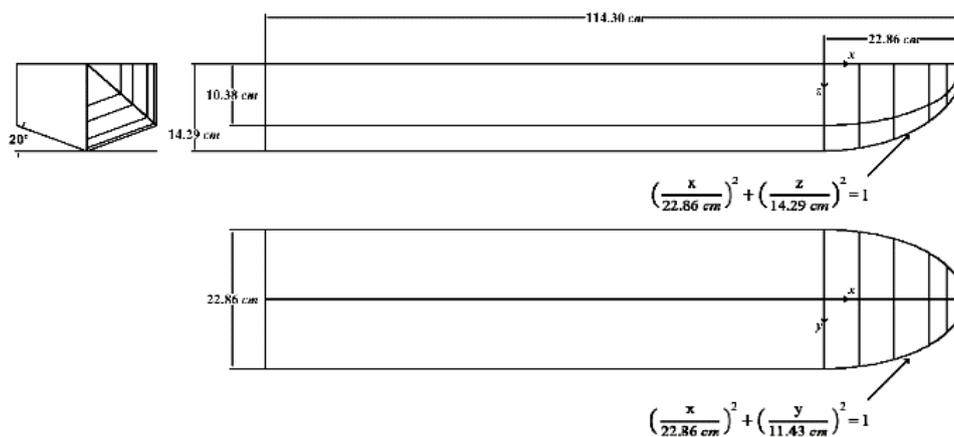


Figure 1. Fridsma hull form [5]

2.2. Numerical Implementation

The Overset grid is a donor-acceptor cell grid method. In overset grid mesh, there is more than one geometry such as background as a donor and overset as an acceptor. Active cells are found in every geometry boundary functioning as an intermediary between donor and acceptor cells, while passive cells only found in background [10]. The overset grid systems have better captures the large motions of the planing hull at high Froude numbers. Towing tank dimension and fluid dynamics boundary condition are explained in Figure 2. The boundary condition for this simulation used to represent a towing tank using the Cartesian coordinate divided into two-part, which are background and overset. The background is defined as velocity inlet are top, bottom, inlet, and back. Pressure outlet is located behind the hull, The longitudinal mid-ship was set symmetry plane. To reduce the time consuming required for computation, half hull model is used.

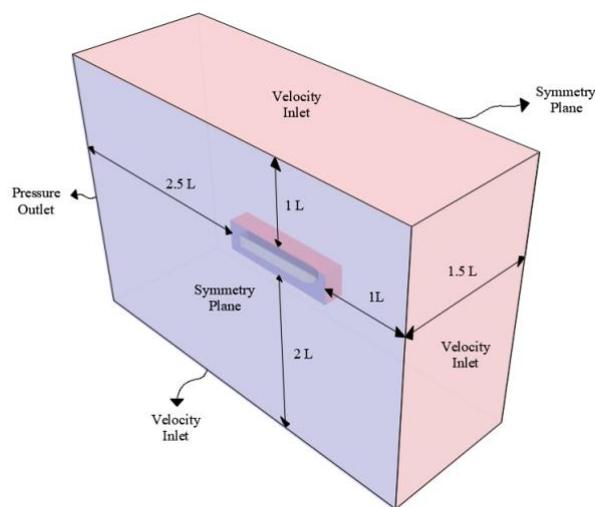


Figure 2. Towing tank dimension and boundary conditions

Mesh density is focused on the ship's hull and water surface so that the result is more accurate. Mesh density locally placed in x,y,z coordinates using isotropic or anisotropic. In the overset grid method, mesh density for both acceptor and donor must be the same grid. If the difference between both mesh density is significant, it can cause the simulations unable to start because of grid mesh incompatibility. In this simulation, mesh density was divided into a few parts shown in table 2 and illustrated in Figure 3.

Table 2. Mesh density

Region name	Mesh dimension
Far Field	0.7874 L
Block 1	0.0492 L
Overlap	0.0246 L
Overset	0.0246 L
Hull	0.0062 L
Free Surface	0.0062 L

Wall function (y^+) is a non-dimensional distance between walls and fluid flows. The purpose of wall function is to predict the boundary layer which has an important part in drag calculations. It is often used to describe how coarse or fine a mesh is for a particular flow pattern. The case study was conducted by Avcı et al., y^+ range between 45 - 60 to achieve high accuracy results. Y^+ calculations according to ITTC [11], defined:

$$\frac{y}{L} = \frac{y^+}{Re \sqrt{\frac{C_f}{2}}} \quad (1)$$

Where y is the thickness of the first layer, L is ship's length, Re is Reynold number and C_f is friction coefficient estimation of the object's surface.

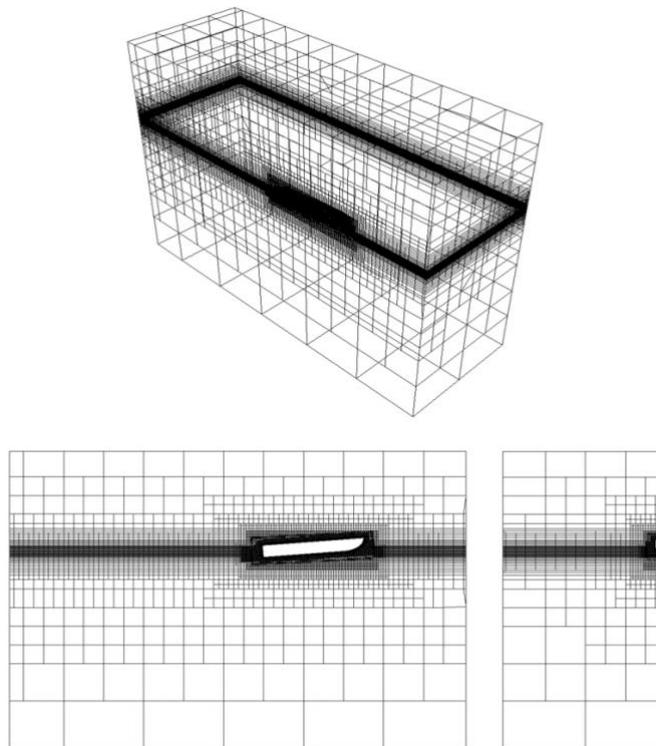


Figure 3. Illustration of mesh density

Time-step is used in unsteady flow. Time-step is a periodic interval for every iteration calculation. The lesser the time step value, the higher the accuracy is, and the other way around. The lesser the time-step value, the more time required to solve a case. To determine time-step in CFD simulations it depends on the ship's speed. The higher ship's speed is, the lower time-step required. Based on calculation recommended by ITTC [11] in Equation 2, ship length noted as L and ship speed noted as U .

$$\Delta t_{ITTC} = 0.005 \sim 0.01 \frac{L}{U} \quad (2)$$

In this research, the ship's motion is limited to y -axis rotational motion (trim) and z -axis translational motion (heave). Other ship motions are not required to achieve the goal of this research.

3. Numerical Ventilation Study

There is no doubt that Numerical Ventilation Problem (NVP) often occurs during high-speed craft simulations, especially high-speed craft. Therefore, mesh refinement are applied in area around ship bow, water surface around overset and background, these applications of mesh refinement are expected to reduce NVP. This method was based on Gray-Stephens et.al [4] which accomplished NVP using modified High-Resolution Interface-Capturing (HRIC) method and mesh refinement to improve courant number quality on the around the ship bow area. Highlighting previous studies De Luca et al. [12] and Mancini et al. [13] are doing the same problem to solve NVP.

Outlining the past-present history of the study of NVP using two methods, the first method is to multiply by 1.15 the total drag from the simulation that is experiencing NVP which means there are 15% ventilation flows around the Wetted Surface Area of the ship. The second method is phase replacement, this is done by replacing mixed-phase with water-phase [2]. The replacement factor depends on the mixed flow intensity. If the air intensity on the mixed-phase is larger, then the replacement factor used will be higher too. In research conducted by Wheeler et al. used a source term to reduce mixed-flow intensity below the hull [3]. Using 0.4 - 0.8 filter to compare the total drag results.

4. Result and Discussion

In this research the drag prediction using CFD trend results, shown in Figure 5 (a). The CFD simulation is shown in the non-dimensionalized unit. It is shown that the unit used is drag/displacement (R/Δ) for every Froude number representing the vessel's speed. The trim results in this study are shown in the degree (ϕ) in Figure 5 (b) to represent the angle of the ship. Non-dimensionalized unit h/B is used to represent heave motions shown in Figure 5 (c), where h is the distance between initial position and ship's final position on LCG , while B is ship's breadth. The planing hull is treated as two degrees of freedom, as free heave and pitch. Generally, total drag occurs between $0 > Fr > 1$ are considered normal, but after reaching $Fr > 1$ it shows significant difference caused by an inability of CFD simulations to solve Numerical Ventilation Problem (NPV). When the hull speed increases, the volume of air diffused below the ship's hull is increasing too, caused by high trim value. The volume fraction of water diffused air below the hull. Practically, these problems are causing unnecessary 30% drag depends on the vessel's speed [14]. Research conducted by Avci et al. shows the total drag unfit the experiment results [2]. This error is similar to the research conducted by wheeler et al. which used Fridsma Hull Form as a validation reference [3]. Based on the result, treatment will be applied to reduce errors caused by Numerical Ventilation Problem. Numerical ventilation is a phenomenon that occurs erroneously in a simulation due to the under-resolution of the mesh, time-step, or other numerical issues [10].

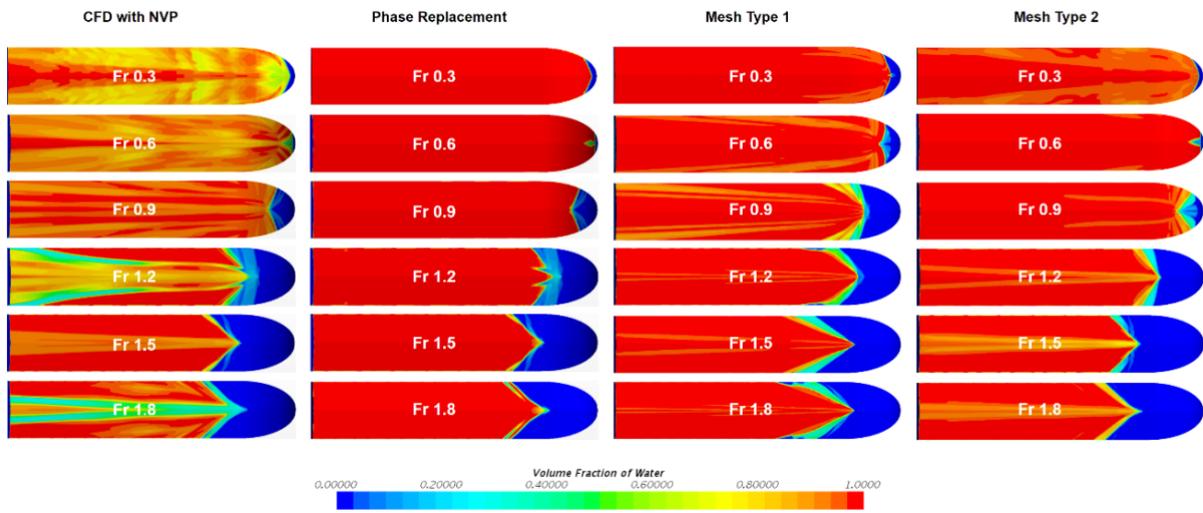
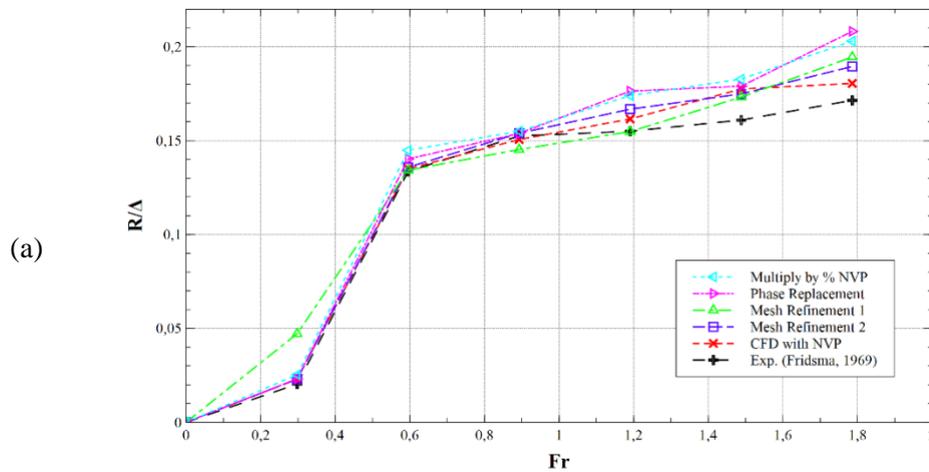


Figure 4. Comparison of each method

In this research, a Froude Number greater than 1 indicates not accordance with the experiment result. In other words, an error occurs during planing condition. As a result indicates that NVP occurs at intervals Froude Number 1.2 - 1.8, it can be seen in Figure 4.



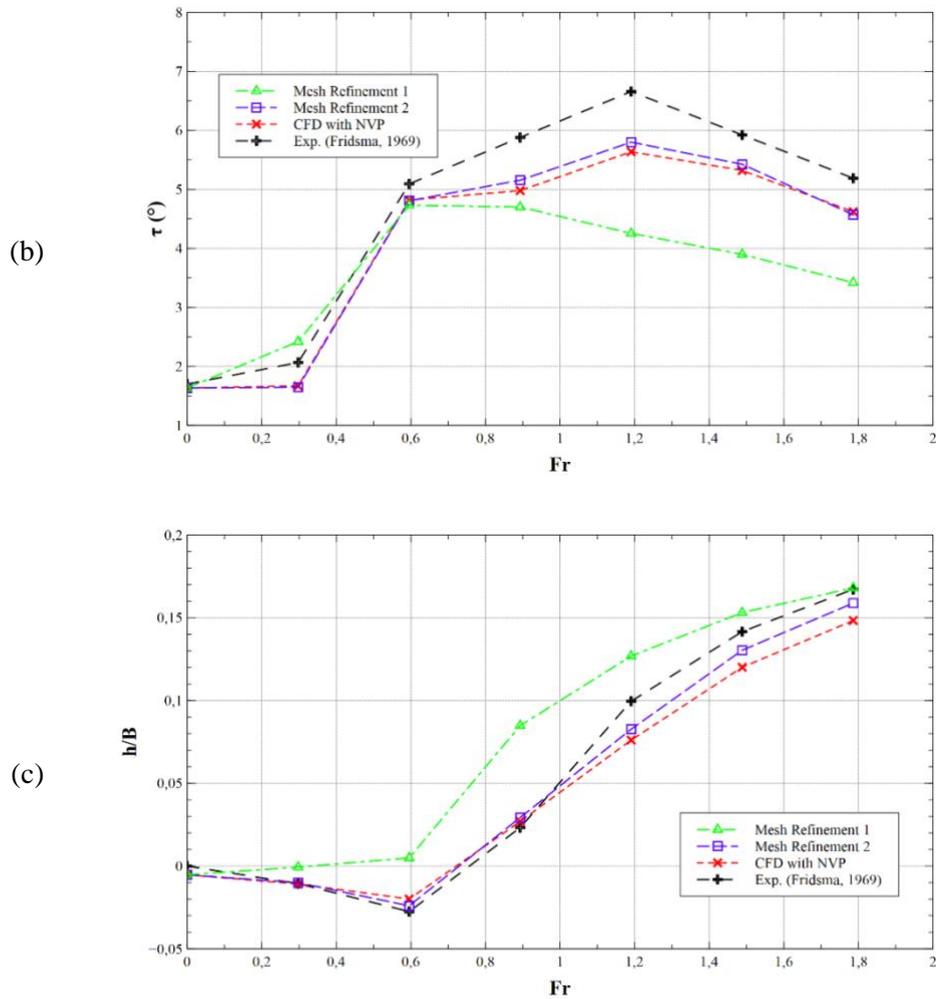


Figure 5. Result comparison of (a) Resistance, (b) Heave, and (c) Trim

4.1. First Method

The first method to solve NPV is to multiply by 1.12 of the total drag force as the intake percentage of air visualized from the volume fraction of water. That is to say, if the visualizations of volume fraction of water are not in water fraction form, then 12% of the total drag is multiplied by 1.12. Figure 6 illustrates the NPV area of 12% at Fr 1.8. So it can be stated that area multiplier depends on NPV areas. Figure 4 shows the differences in every ship's speed. It should be noted that NPV cases do not affect trim and heave predictions, hence this research is mainly focused on total resistance predictions.

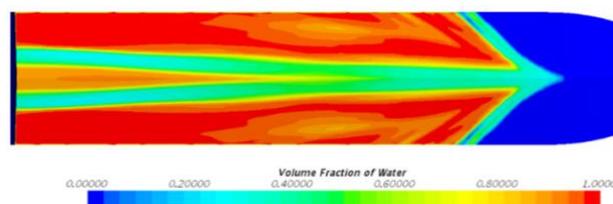


Figure 6. NVP in Fr 1.8

4.2. Second Method

The second method is using phase replacement method, this will replace the air phase with the water phase. The volume fraction of water coefficient which has a value of less than 1 will be replaced. The result indicates good performances based on how the colors under the hull are. Figure 7 indicates significant change showing volume fraction of water visually. On heave and trim measurement, this method does not affect so much of the simulation results. The result heave and trim using this method can be seen in Figure 4 (b) and Figure 4 (c). The steps of phase replacement are written below [10]:

- Before starting the simulations, activate "Multiphase Interaction" in Physic.
- Stop the simulations after reaching the convergence state.
- Visualize "Volume of Fraction of Water"
- On the "Multiphase Interaction", create new phase interaction
- Choose "VOF-VOF Phase Interaction Model" and "VOF Phase Replacement Model"
- Use "Field Function" on the Tools menu to replace phase.
- Create new "scalar" formula in Field Function and define as " $\{\text{VolumeFractionAir}\} < 0.50$ ".
- Back to "Multiphase Interaction" menu, choose "Air" for "Primary Case" and "Water" for "Secondary Case". This will replace the air phase with the water phase.
- Continue simulation for one time-step, and re-visualize "Volume Fraction of Water"

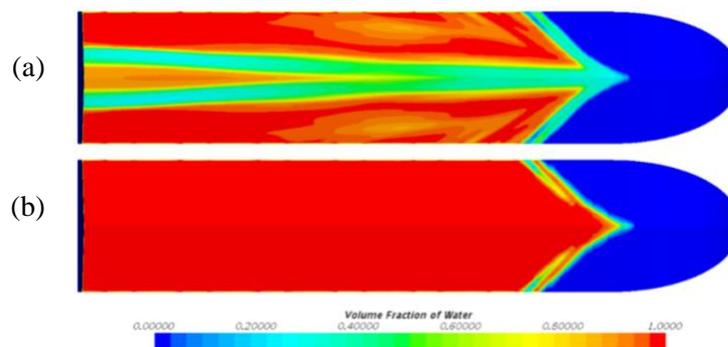


Figure 7. Volume fraction of water comparison in Fr 1.8 (a) NVP under the hull (b) after phase replacement method

4.3. Third Method

The third method is using mesh refinement, which is an important method for adapting meshes in order to increase the accuracy of the solution within certain sensitive or turbulent regions of simulation. Mesh refinement is conducted by improving mesh quality on the water surface. In this research, there are two methods of mesh refinement are used, refinement 1 is focused around ship's bow as shown in Figure 8(a) and refinement 2 is focused on around ship's as shown in Figure 8(b).

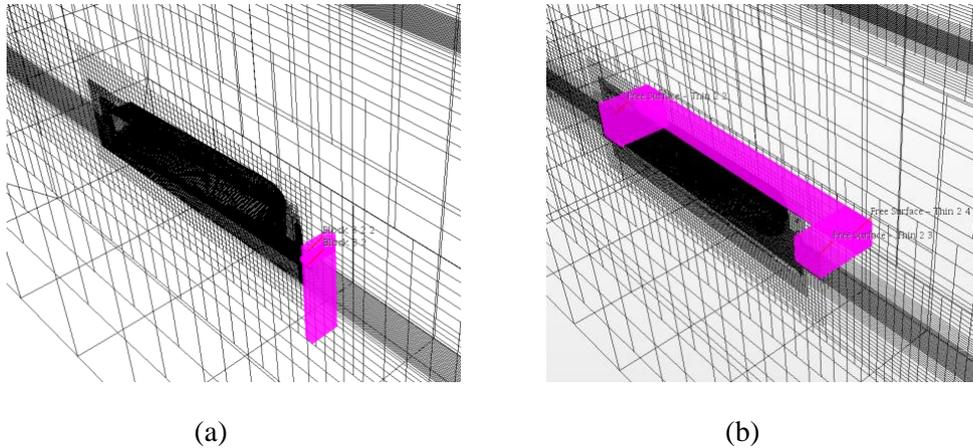


Figure 8. Visualization of (a) mesh refinement type 1 and (b) mesh refinement type 2

Reporting customizations performed of resistance, heave, and trim show different performances on both refinement methods. Refinement 1 indicates there is a large gap in heave and trim prediction. While refinement 2 shows the best results in terms of resistance, heave, and trim. The refinement 2 shows similar trends with the experiment results. Figure 9 shows the visualization volume fraction of water in Froude Number 1.8 with different refinement methods. The obtained visualization show that refinement 1 is greater than refinement 2, shown in red color which is more dominant at the bottom of the ship. Figure 10 (a) shows that improving mesh refinement able to reduce NVP. Meanwhile, Figure 10 (b) shows that there is some area that has an NPV under the ship’s hull.

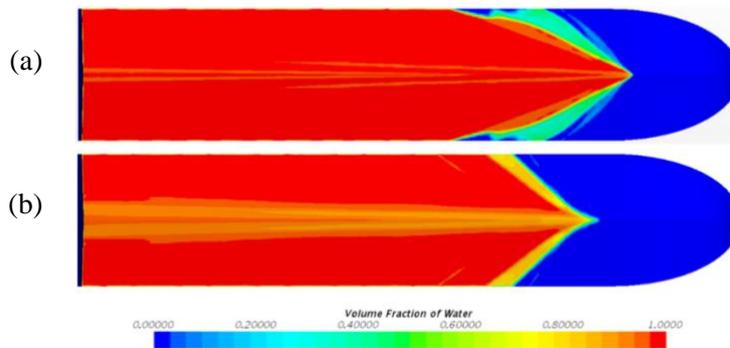


Figure 9. Volume fraction of water comparison in Fr 1.8 (a) mesh refinement 1 and (b) mesh refinement 2

Referring to Figure 11, it can be seen that fine mesh can reduce diffused flow that is the main cause of NV. As was stated before that increasing mesh quality in one specific area (ship’s bow) can cause inaccurate resistance. Thus, meshes around the overset boundary, are very likely to get more accurate results. While in terms of reducing NVP, it is recommended to improve mesh quality on areas around the ship’s bow and incoming flow angle.

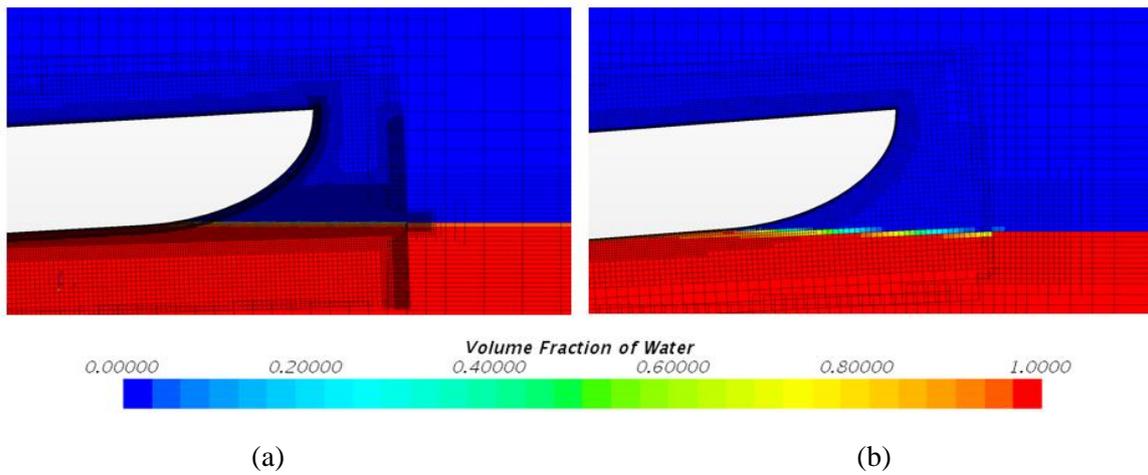


Figure 10. Volume fraction of water comparison in Fr 1.8 (a) mesh refinement 1 and (b) mesh refinement 2.

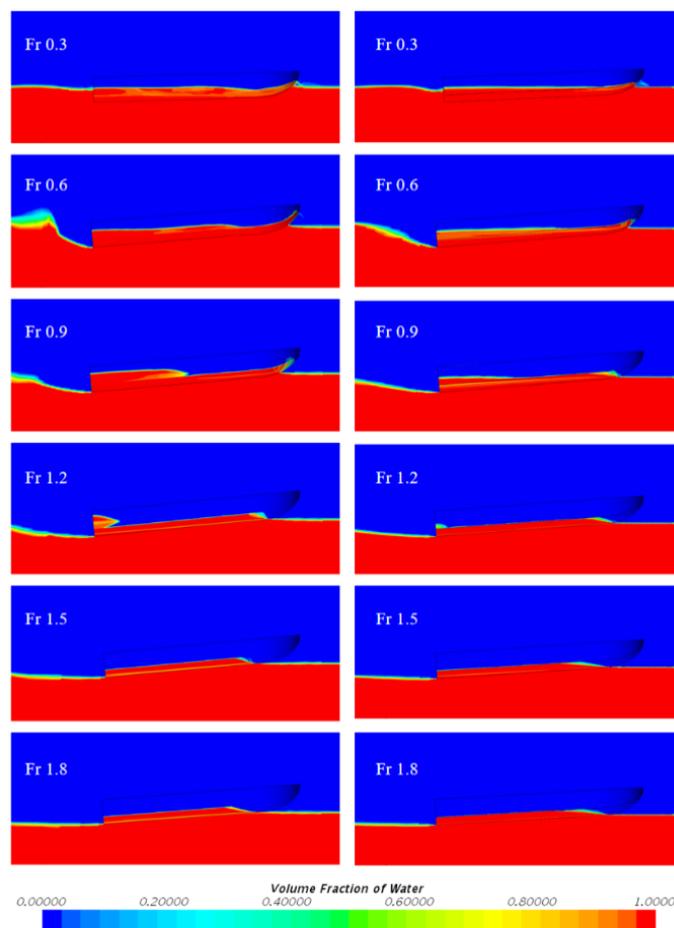
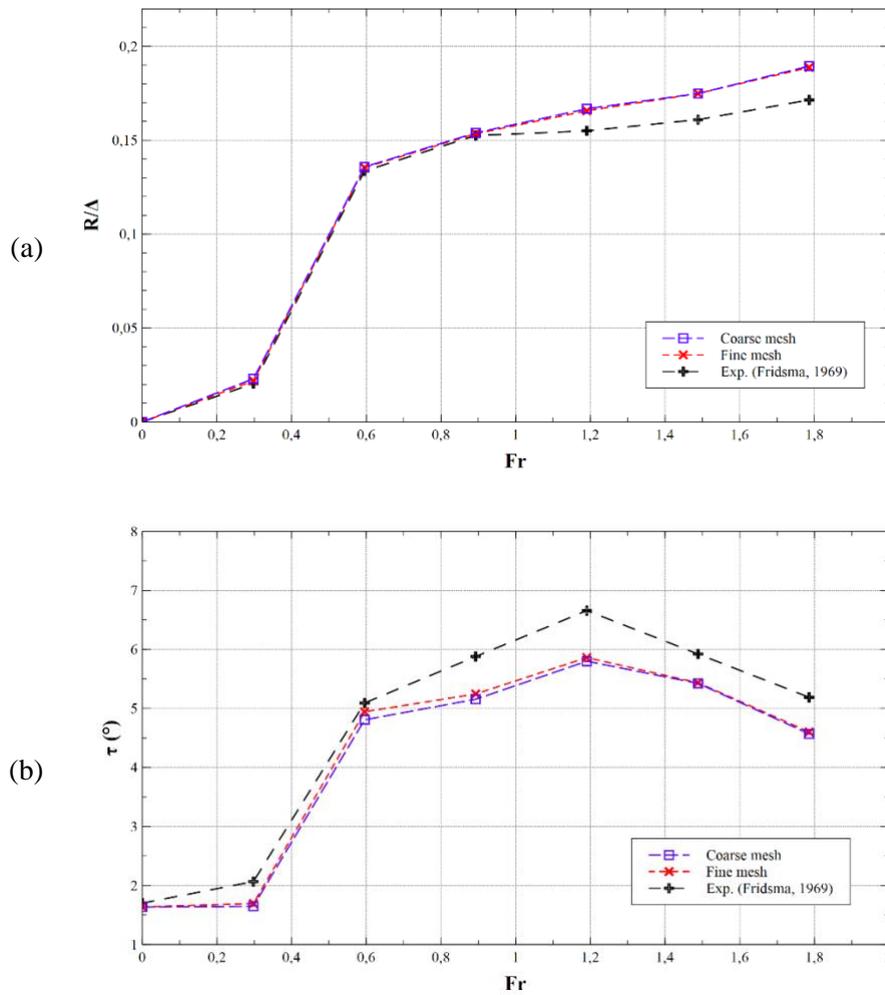


Figure 11. Comparison free surface elevation between mesh refinement 1 (left) and mesh refinement 2 (right)

4.4. Mesh Comparison

Fine mesh and coarse mesh results are compared to each other. The mesh qualities are used to show the effect of mesh number to resistance, heave motions, and trim performances. Coarse mesh has a total of 700,000 cells and fine mesh used 1,450,000 cells. The number of mesh differences does not have a significant effect on resistance, heave motions, and trim performance as shown in Figure 12.



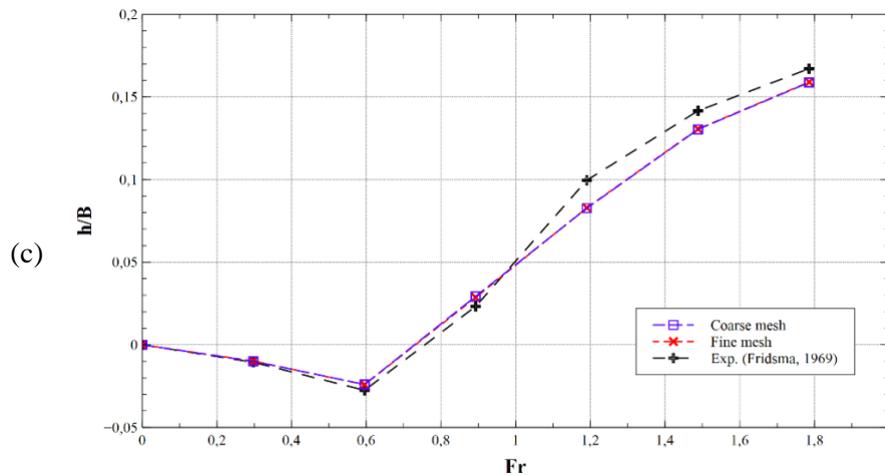


Figure 12. Result comparison of (a) Resistance, (b) Heave, and (c) Trim

5. Conclusion

Numerical Ventilation Problem often occurs during high-speed boat with high Froude number (Fr) using Volume of fluid. The methods to solve the problem indicates each own's weaknesses and advantages. Mesh refinement method is shows good performance in solving NVP. On the other hand, it minimizes errors in resistance predictions. Increasing the number of mesh does not have a major effect on performance results. Results show that the numerical results agree well with experimental data at $Fr < 1$, however at $Fr > 1$ the gap show between 7-9.5% of total resistance. This research indicates that a numerical approach at high speed will reduce the level of accuracy.

Acknowledgment

The authors wished to thank Diponegoro University for the funding support of research ID 145/UN7.5.3.2/HK/2020, March 2, 2020.

References

- [1] Faltinsen O M 2005 *Hydrodynamics of High-Speed Marine Vehicles* (New York: Cambridge University Press)
- [2] Avci A G and Barlas B 2018 An experimental and numerical study of a high speed planing craft with full-scale validation *J. Mar. Sci. Technol.* **26** 617–28
- [3] Wheeler M P, Matveev K I and Xing T 2018 Validation study of compact planing hulls at pre-planing speeds *ASME 2018 5th Joint US-European Fluids Engineering Summer Conference (Montreal)* (New York: ASME) p 1–8
- [4] Gray-Stephens A, Tezdogan T and Day S 2019 Strategies to minimise numerical ventilation in CFD simulations of high-speed planing hulls *ASME 2019 38th International Conference on Ocean, Offshore and Arctic Engineering (Glasgow)* (New York: ASME)
- [5] Fridsma G 1969 *A Systematic Study of The Rough-water Performance of Planning Boat* (New Jersey: Stevens Institute of Technology)
- [6] Samuel, Iqbal M and Utama I K A P 2015 An investigation into the resistance components of converting a traditional monohull fishing vessel into catamaran form *Int. J. Technol.* **6** 432–41
- [7] Samuel, Kim D J, Iqbal M, Bahatmaka A and Rio Prabowo A 2018 Bulbous bow applications on a catamaran fishing vessel for improving performance *MATEC Web Conf. (Bali)* vol 159 (French: EDP Sciences) p 1–6

- [8] Samuel, Trimulyono A and Santosa A W B 2019 Simulasi CFD pada kapal planing hull *KAPAL Jurnal Ilmu Pengetahuan & Teknologi Kelautan* **16** 123–28
- [9] Yousefi R, Shafaghat R and Shakeri M 2013 Hydrodynamic analysis techniques for high-speed planing hulls *Appl. Ocean Res.* **42** p 105–13
- [10] CD-Adapco 2017 *User guide STAR-CCM 13*
- [11] ITTC 2011 *Recommended Procedures and Guidelines Practical Guidelines for Ship CFD* p 1–18
- [12] De Luca F, Mancini S, Miranda S and Pensa C 2016 An extended verification and validation study of CFD simulations for planing hulls *J. Sh. Res.* **60** 101–18
- [13] Mancini S, De Luca F and Ramolini A 2017 Towards CFD guidelines for planing hull simulations based on the Naples Systematic Series *7th Int. Conf. Comput. Methods Mar. Eng. Mar. 2017(Nantes)* (Barcelona: CIMNE Congress Bureau) p 1071–85
- [14] Federici A 2014 *Design and analysis of non-conventional hybrid high-speed hulls with hydrofoils by CFD methods* (Liguria: University of Genoa)