

RESISTANCE PREDICTION OF SOLAR BOAT IN CALM WATER USING COMPUTATIONAL FLUID DYNAMIC APPROACH

Nur Ain, A.R¹, Muhamad Daniel Haiqal¹, Mohd Azahari, J.¹, Khairuddin, N.M.¹, Norhanifah, A.R¹ and N. Manap¹

¹School of Mechanical Engineering, College of Engineering, Universiti Teknologi MARA, Cawangan Terengganu, Kampus Bukit Besi, 23200 Bukit Besi, Dungun, Terengganu

ainabdrahman@uitm.edu.my

ABSTRACT

This project presents a CFD simulation on a swath solar boat. Swath is a Small Waterplane Area Twin Hull. It is a hull design that has a small waterplane area with pontoons. Researching the hull for the resistance and pressure distribution analysis takes too much time and money. One way to solve this is by implementing CFD in Boat development. Computer Fluid Dynamics, CFD is a simulation to replicate real-world data. It starts by developing the boat's model and setting up the boundary conditions around it. The boundary will be mesh and transferred into CFD, followed by setting up the code solver and running the simulation. Next, the resistance and hull distribution can be generated. In this project, it is found that the higher the speed, the higher the resistance. Pressure distribution at a slow speed is concentrated below the pontoons, but when the speed increases, the pressure is more concentrated in front of the pontoons and struts. This project is expected to give a significant improvement in ship development.

Keywords: SWATH, Resistance, CFD

1.0 INTRODUCTION

The solar boat is a different type of boat than a typical standard boat. It is because the boat's engine uses different types of energy. The solar boat will use electricity rather than the famous diesel engine choice. Solar boat is more eco-friendly to the environment than conventional boat since the power source is cleaner and more renewable than diesel, which is obtained from petroleum. Diesel engines will also produce carbon dioxide (CO₂) that will heat our earth. Our planet faces a global warming issue that melts the ice south pole that keeps the sea level rising yearly. The solar boat is tough to apply in many boats. The solar boat is heavier than a typical electric boat because it carries a solar panel. One of the boats suitable for using a solar boat is the recreational boat. [1]

A recreational solar boat is a perfect implementation because a solar boat will be a big one since the boat needs to put solar panels on to get the power. The solar panel will be put on the ship's roof, making the space below the ship more space than a typical boat. Also, an eco-friendly boat like this will encourage people to care about the environment. More people who are conscious about the environment would like to use or ride solar boats than old diesel boats for recreation.

2.0 SWATH HULL

Small Waterplane Area Twin Hull (SWATH) is the acronym for Small Waterplane Area Twin Hull (SWATH). Naval architects investigated the seakeeping motions of a catamaran. The large deck area was appealing, but the ship constantly bounced because of all the waves. The waves' forces proved to be proportional to the width of the hulls, as the designers anticipated. The waterplane area, more specifically [2]

The waterplane area was shrunk to the size of a small strut, just big for a crew member to sit down. All the buoyancy was concentrated in submerged hulls that resembled submarine hulls. The SWATH was born, a ship with exceptional seakeeping because waves have a low impact.

Swath provides superior seakeeping capability. The deck area of SWATH ships is comparable to that of a catamaran, but they have significantly stronger seakeeping qualities. The idea of SWATH derives from semi-submersible offshore rigs, which are designed to provide a working platform with minimized motions in the open sea [3]

The crew will benefit significantly from the reduced ship movements. The crews of research ships may not be experienced mariners. Reduced ship vibrations can enhance crew productivity because they feel seasick quickly. Even seasoned mariners appreciate the smooth gliding motion of a SWATH vessel. There will be no more rocking. There are only gentle surges as the ship gradually rises and slides along the waves.[4]

2.2 SHIP RESISTANCE

There are five types of resistance: wave-making, frictional, eddy, and air resistance. All this resistance will be combined to create a simple force called total resistance, R_t . Wave-making resistance is essential in ship-making because the wave movement has an energy that could repeal the ship's movement. This wave resistance can be ignored in submarine development [5]. The wave-making resistance happens due to the energy the ship needs to continuously supply to the wave system created on the water's surface. The frictional resistance is due to the motion of the hull through a viscous fluid [6].

Eddy resistance is due to the energy that eddies shed from the hull or appendages carry away. Local eddying will occur behind appendages such as boss-ins, shafts, and shaft struts and from stern frames and rudders if these items need to be appropriately streamlined and aligned with the flow. Also, if the after-end of the ship is too blunt, the water may not follow the curvature and will break away from the hull, again giving rise to eddies and separation resistance. Eddy resistance and wave-making resistance have a connection. The resistances under eddy resistance and Wave making are commonly taken together under residuary resistance[7]

2.3 COMPUTATIONAL FLUID DYNAMIC (CFD)

Computational fluid dynamic "CFD" is a method widely used in the engineering field since the improvement in the software keeps getting better and more precise each year. Wei yufeng stated that CFD involves many advantages, such as a shorter duration, more straightforward operation, lower cost, and higher repeatability [8]. Overall, CFD has been widely utilized to analyze various transmission processes, such as fluid flow, heat transfer, and mass transfer, and has received widespread recognition from researchers and users. As a result, it is crucial in hydraulics scientific study and technical design.[9]

In computational methods, the computational domain refers to the region of space or the physical system that is discretized and represented by a computational mesh or grid. This domain is typically defined by the boundaries that enclose it. On the other hand, boundary conditions are specific constraints or conditions applied to the boundaries of the computational domain. These conditions are used to model the system's behavior at its boundaries and to ensure that the numerical solution accurately represents the simulated physical phenomena.

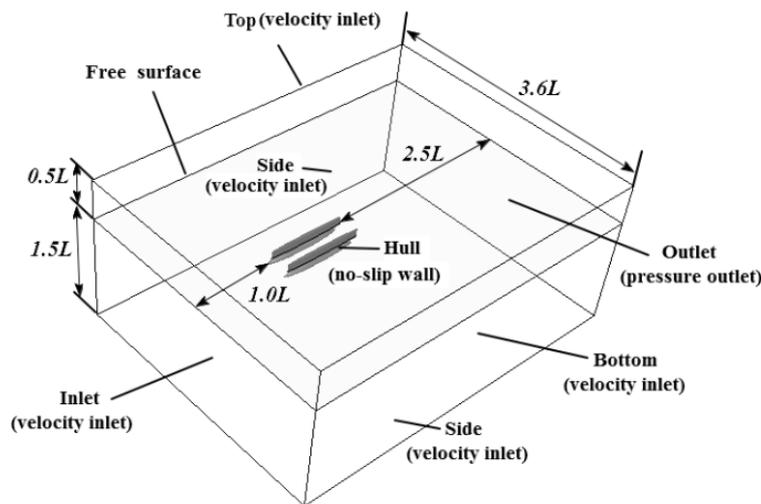


Figure 1: Boundary Setup [10]

The Volume of Fluid (VOF) method evolves a free surface. The computational domain for this study's boundary setup reference is in Figure 1. The dimensions of the cuboid domain are $1.0L$ in front of the bow, $2.5L$ behind the stern, $1.8L$ in the lateral direction, and $0.5L$ above and $1.5L$ below the calm sea surface. The hull body is subjected to the no-slip wall boundary condition. For the outlet, the pressure outlet condition is employed. The remaining limits are considered velocity inlets. In addition, numerical wave dampening is applied to side boundaries to prevent unwanted wave reflection.

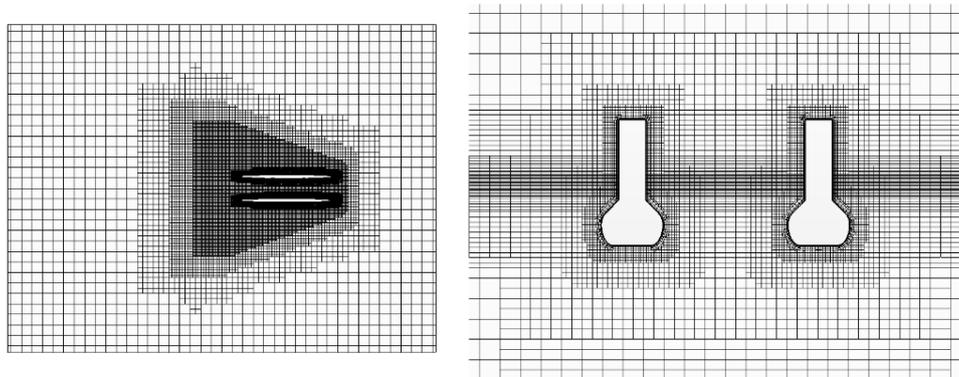


Figure 2: Grid of hull [10]

A grid of hulls sometimes called a grid of cells or just a grid, can be used to discretize the computer domain. This grid will divide The computer domain into smaller regions, enabling one to conduct computations or data analysis within those sections. Figure 2 shows that the grid of the hull consists of Six layers of prismatic cells with a growth ratio of 1.2 are developed to resolve near-wall flow better. A thickness of the first grid layer is chosen at 0.0011 m, reflecting the suggested nondimensional distance y^+ range of 30 to 100 for the wall functions. Volume refinement blocks are built around the hull, free surface, and Kelvin wave area to resolve the flow field further [10].

3.0 RESEARCH METHODOLOGY

3.1 Flow Chart

The methodology of engineering research follows a systematic approach to ensure thorough and ordered investigation. as in Figure 3.1.

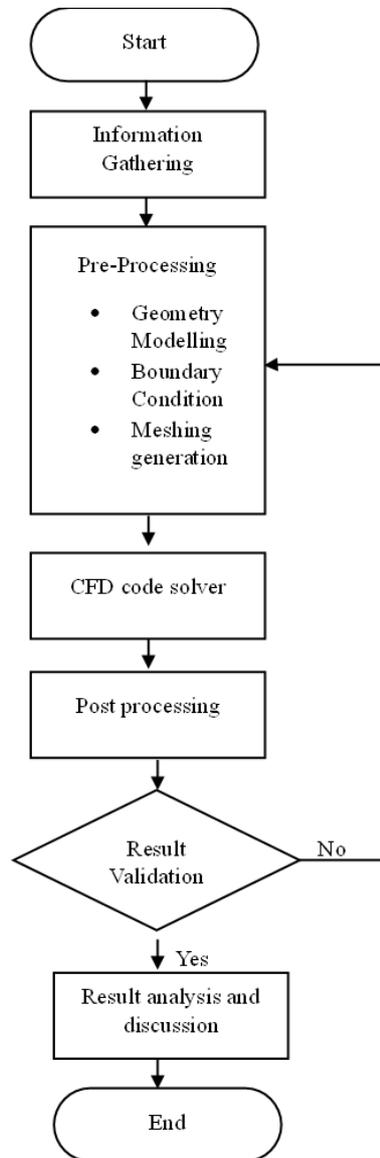


Figure 3: Flow Chart

The first step is understanding and searching for any literature related to the project. This requirement is essential to find a critical review of previous work related to the project's theoretical background and methods used in similar problems or requirements. All this information can be utilized to form the basis for this project methodology. Then it is continued by the CFD simulation part using Ansys Fluent, starting with Pre-Processing, next is CFD code solver, and Post-processing. Then, the result obtained is validated and analyzed.

3.2 BOAT MODELING

This study describes a CFD simulation of a solar-powered swath boat with the model dimension as in Table 3.2. Swath is a Twin Hull with a Small Waterplane Area. It is a vessel design with pontoons and a small waterplane area. Figure 4 is the solar-powered swath boat modeling developed using Polycad software.

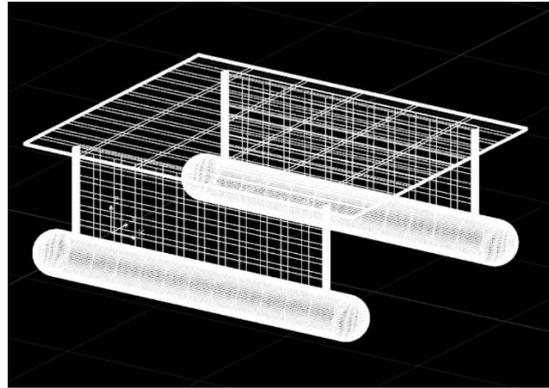


Figure 4: Swath solar boat modelling

Table 1 shows the dimension of the hull for this project. The draft of the hull is set at 0.5 meters. The length overall is 2.5 meters. The torpedo and platform have the same length. The hull breadth is 2 meters. The pontoon spacing with each other center is 1.7 meters. This dimension will be the same as the analysis model because no real-life experimental will be conducted.

Table 1: Model Dimension

Dimension	Full Scale
Length Over All (m)	2.5
Length of waterline (m)	1.94
Breadth Over All(m)	2
Hull Spacing Between Waterline (m)	1.7
Draft (m)	0.5
Length Pontoons/Torpedos (m)	2.5
Length of the platform (m)	2.5

3.3 CODE SOLVER

Table 2 is the code solver employed to execute the simulation in CFD Fluent. In this instance, y is the gravity plane, and the gravity value will be -9.81 according to the Multiphase Model. Fluid Volume is selected, and open channel flow and implicit body force are activated. Laminar is the model's viscous model, and pressure inlet and pressure outlet are enabled, and a ship's wall is defined as not a slippery surface. The initialization is hybrid and flat, indicating that the water is already within the boundary and

begins at the inlet. The plan is coupled, and the pressure is instantaneous by using the second-order upwind as the momentum for the solution, and the volume fraction is a result of compression. This computation will use the automatic time step method with a time scale factor of 0.01 seconds. This investigation repeated at various speeds, including 0.5 m/s, 1 m/s, 1.5 m/s, 2 m/s, 3 m/s, 4 m/s, and 5 m/s.

Table 2: Code Solver

Gravity	-9.81m/s
Multiphase model	Volume of fluid
Open channel flow	enable
Implicit body force	enable
Viscous model	Laminar
Pressure inlet	enable
Pressure outlet	enable
Ship wall	No slip
Initialisation	Hybrid, flat, inlet
Scheme	coupled
Pressure	Presto
Momentum	Second order upwind
Volume fraction	Compressive
Time step method	automatic
Time scale factor	0.01

4.0 RESULTS AND DISCUSSION

The results are divided into two parts. The first part is the CFD resistance result; the second is the Pressure distribution on the hull.

4.1 RESISTANCE

Table 3:Resistance Data

Speed m/s	Resistance swath, N	Total Resistance, N	Friction Resistance, N	Wave Making Resistance, N
0.5	4.1164608	8.2329216	0.170348498	8.062573102
1	16.299976	32.599952	0.170348498	32.4296035
1.5	40.710044	81.420088	0.170348498	81.2497395
2	68.060948	136.121896	0.170348498	135.9515475
3	124.10302	248.20604	0.170348498	248.0356915
4	195.21458	390.42916	0.170348498	390.2588115
5	275.38678	550.77356	0.170348498	550.6032115

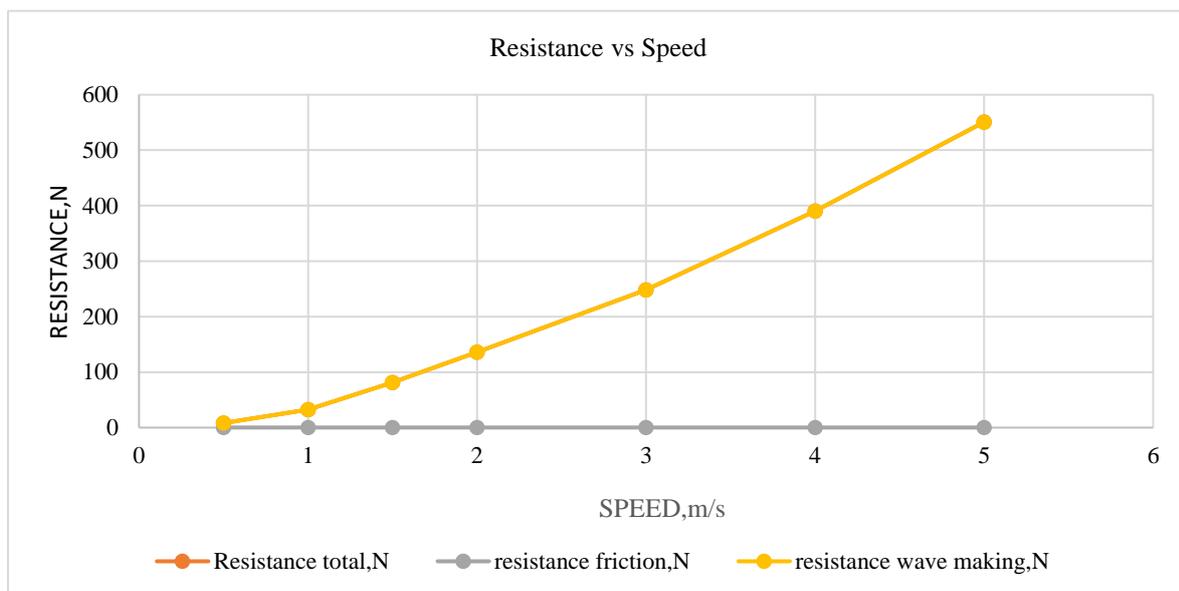


Figure 5: Graph Resistance vs Speed

Figure 5 shows a component of resistance: total resistance, friction resistance, and wave-making resistance. The graph and table show that the resistance of friction is constant. It is also shown that total resistance and wave-making resistance increase when the speed increases. The figure shows the highest resistance point in this project analysis of each resistance in 5m/s. The total resistance is 550.77356N, and the wave-making resistance is 550.6032115N. Meanwhile, at 5m/s, the resistance of friction is constant at 0.170348498N.

4.2 PRESSURE DISTRIBUTION

Figure 6, 7, and 8 shows the pressure distribution acting on the hull. The pressure distribution of the hull changes at different speeds. In 0.5m/s, table 3 show that the highest point of distribution is at the bottom of the pontoon due to the water depth as the simulation was carried out in calm water.

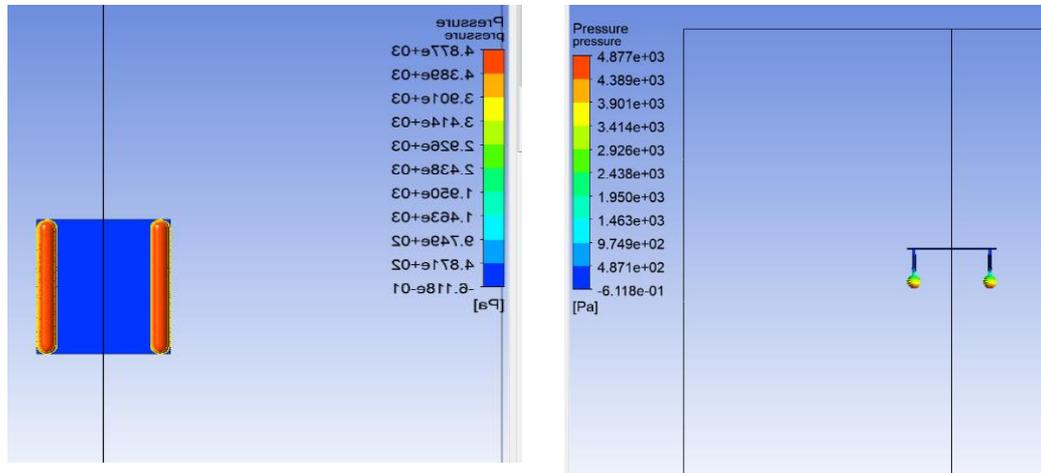


Figure 6: Pressure distribution on 0.5m/s

The pressure distribution is noticeable to start shifting at 2m/s, and the pressure's focus point changes from the bottom of the pontoons to the front of the pontoons as in Figure 7.

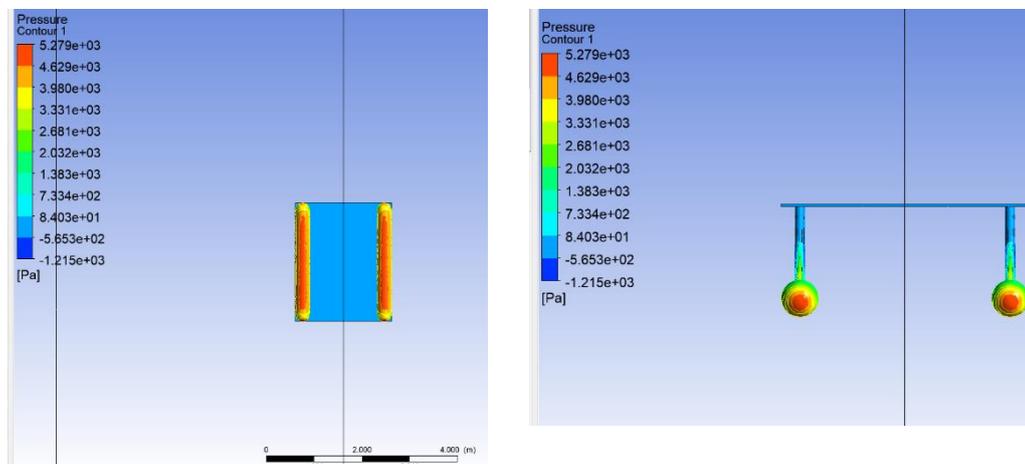


Figure 7: Pressure distribution on 2m/s

The most visible change of focus area of pressure distribution happens in 5m/s, as in Figure 8. The highest-pressure distribution point changes from below the pontoons into the front of the pontoons and the strut. Pressure distribution changes when the speed increases due to the increment of swath boat resistance.

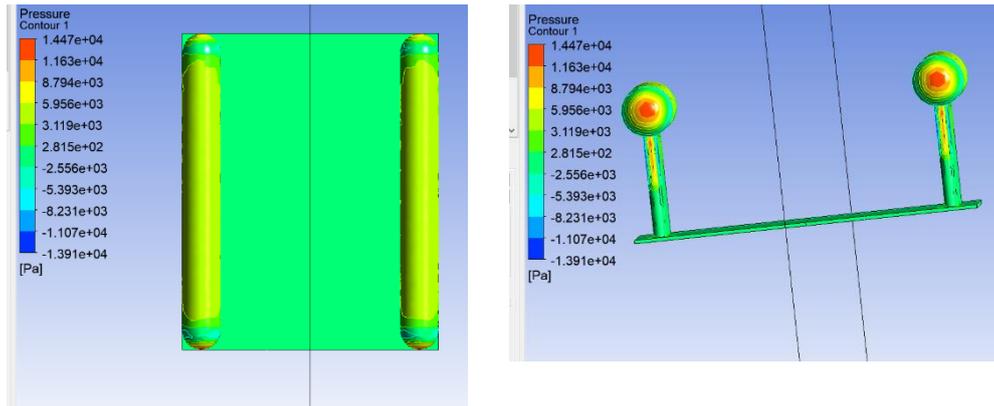


Figure 8: Pressure distribution on 5m/s

4.3 Resistance Validation

Table 4: Resistance Validation

Speed, m/s	Rt hull 1 Swath, N	RT full, N	Rt full reference, N [3]	Mismatch, N	Percentage error%
0.976	1.428326	2.856652	2.748	-0.10865	3.953857
1.301	1.7116237	3.4232474	3.132	-0.29125	9.299087
1.952	3.4759031	6.9518062	5.918	-1.03381	17.46884
2.603	5.1922978	10.3845956	9.4	-0.9846	10.47442

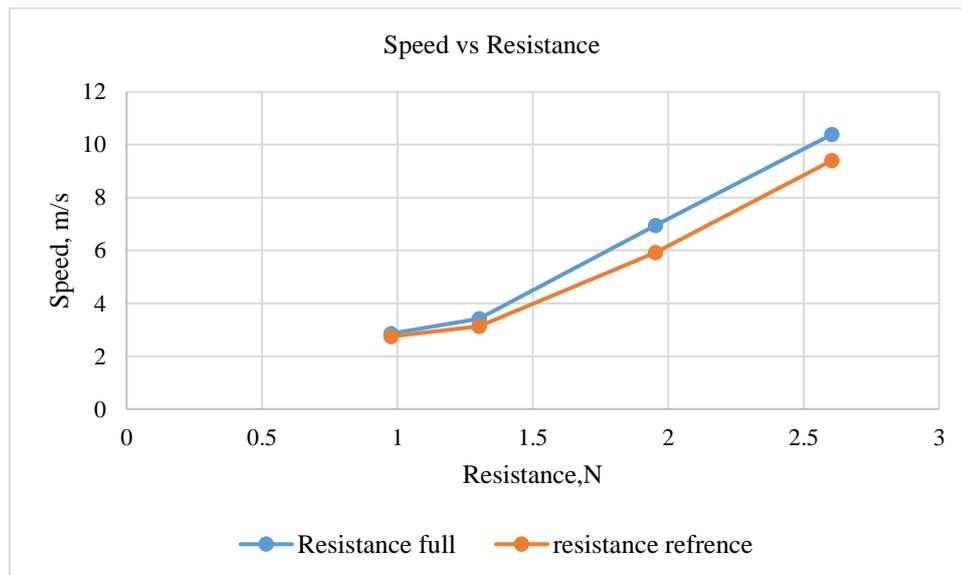


Figure 9: Graph Speed vs Resistance

Table 4 and Figure 9 above show two data sets: reference data and replicate data. Replicate data is run by a model that has been replicated using the Reference data [3] model. The Two data are run with four speeds: 0.967,1.301,1.952, and 2.603 m/s. These two data show some agreement with an error percentage under 20%. These error percentage is mismatched under 1N of resistance. The highest error is 18%, and the lowest is 4 %.

5.0 CONCLUSION AND RECOMMENDATION

Water flow around the boat is simulated using the Volume of the fluid method using Ansys Fluent. Ansys Fluent can get the resistance on hull data in Newton and the hull's Pressure distribution. The data summarize that the faster the boat goes, the more resistance will increase. The method is validated using other research papers about CFD analysis on the Swath hull to know the validation for the pressure distribution on the hull changes throughout the speed change. The highest-pressure point is in the front of the pontoons and struts at final speed.

Meanwhile initial speed, the highest-pressure point is the bottom of the hull. This project achieves its objective: find resistance on the boat and pressure distribution on the hull. The method is validated by another research paper CFD ANSYS Fluent analysis with under 20% error.

The way to improve this analysis is to use a high-specification device because it could make the analysis faster. Smaller meshing sizing will make the data more accurate, but the analysis time will become slower. Experiments can be done to further validated the data.

REFERENCES

- [1] M. Mittendorf and A. D. Papanikolaou, "Hydrodynamic hull form optimization of fast catamarans using surrogate models," *Ship Technology Research*, vol. 68, no. 1, pp. 14–26, Jan. 2021, doi: 10.1080/09377255.2020.1802165.
- [2] A. Zare, H. Sayyadi, and M. H. Karimi, "Numerical Modeling of an Advanced Semi-SWATH Hull in Calm Water and Regular Head Wave," *Journal of Marine Science and Application*, vol. 20, no. 4, pp. 646–659, Dec. 2021, doi: 10.1007/s11804-021-00239-6.
- [3] E. Begovic, C. Bertorello, A. Bove, and F. De Luca, "Experimental study on hydrodynamic performance of SWATH vessels in calm water and in head waves," *Applied Ocean Research*, vol. 85, pp. 88–106, Apr. 2019, doi: 10.1016/j.apor.2018.10.012.
- [4] E. Begovic, S. Mancini, and E. Begovic Carlo Bertorello Simone Mancini, "Hydrodynamic performances of small size swath craft CFD Self propulsion simulation on Surface Combatant Ship View project Mathematical modelling of planing hulls maneuvering in 6DOF View project HYDRODYNAMIC PERFORMANCES OF SMALL SIZE SWATH CRAFT," vol. 66, 2015, [Online]. Available: <https://www.researchgate.net/publication/289129179>
- [5] A. Ali, A. Maimun, and Y. M. Ahmed, "Analysis of Resistance and Generated Wave around Semi SWATH Hull at Deep and Shallow Water," *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences Journal homepage*, vol. 58, pp. 247–260, 2019, [Online]. Available: www.akademiabaru.com/arfmts.html
- [6] van M. J.D and O. P. van, *Principles of Naval Architecture Volume II: Resistance, Propulsion and Vibration by Hardcove, Arbutus Books*, vol. 2. 1988.

- [7] C. M. Lee, J. W. Yu, J. E. Choi, and I. Lee, "Effect of bow hull forms on the resistance performance in calm water and waves for 66k DWT bulk carrier," *International Journal of Naval Architecture and Ocean Engineering*, vol. 11, no. 2, pp. 723–735, Jul. 2019, doi: 10.1016/j.ijnaoe.2019.02.007.
- [8] Y. Wei, "The development and application of CFD technology in mechanical engineering," *IOP Conf Ser Mater Sci Eng*, vol. 274, p. 012012, Dec. 2017, doi: 10.1088/1757-899X/274/1/012012.
- [9] D.-J. Kim, D.-J. Kim, M. Iqbal, A. Bahatmaka, and A. Prabowo, "Modification of Traditional Catamaran to Reduce Total Resistance: Configuration of Centerbulb," *Maritime Transportation and Harvesting of Sea Resources*, pp. 541–546, 2019.
- [10] K. Dai and Y. Li, "Experimental and numerical investigation on maneuvering performance of small waterplane area twin hull," *Brodogradnja*, vol. 72, no. 2, pp. 93–114, Apr. 2021, doi: 10.21278/brod72206.