INTERNATIONAL JOURNAL OF METACENTRE
MARITIME APPLICATION, TECHNOLOGY, ENGINEERING, AND TRANSPORTATION
http://ijm-nasp.unhas.ac.id. eISSN : 2809-8307

Study of Ship Resistance due to Trim using Computational Fluid Dynamics

Muhammad Anjas Syam[1*], Suandar Baso[2], Andi Ardianti[3]

[1]*Department of Naval Architecture, Batulicin Polytechnic, Indonesia*
[2,3]*Department of Naval Architecture, Faculty of Engineering, Hasanuddin University, Indonesia*

Received: 10/10/2023
Revised: 27/11/2023
Accepted: 10/12/2023
Published:31/12/2023

* Corresponding author:
anjasmuhammad77@gmail.com

Abstract

When a ship experiences trim, the geometric parameters of the underwater ship change compared with even keel condition, such as the ship's draft, length of waterline, and wetted surface area. All these factors affect the ship resistance. The purpose of this research is to understand the influence of trim on ship resistance, and to explain the comparison of ship resistance in even keel, trim by bow, and trim by stern conditions based on Computational Fluid Dynamics analysis.. The method for calculating resistance in this research applies software Autodesk CFD. The type of ship in this study is a container ship. In this study, the hydrodynamic analysis of the ship has been successfully conducted, dynamic parameters such as velocity magnitude, static pressure distribution and resistance have been obtained. The ship model's resistance trim by bow and trim by stern conditions shows a similar tendency, increasing with the trim value. However, when comparing the ship model resistance from each trim condition with the resistance in the even keel condition, there is a difference. The trim by bow condition experiences an increase with an average percentage value of 3.32%. Meanwhile, the trim by stern condition experiences a decrease with an average percentage value of 0.06%. Then, overall, trim by bow condition of -1.441 m shows the highest resistance at 6.044 N and trim by stern condition at 0.5 m has the smallest resistance, which is 5.674 N.

Keywords: Ship resistance; Trim by bow; Trim by stern; autodesk CFD

1. Introduction

Mobilizing goods using maritime transportation has environmental implications. Maritime transportation, especially ships using diesel engines, is one of the sources of air pollution. Large engine ships using diesel or marine gas oil produce emissions such as Nitrogen oxides (NOx), carbon monoxide (CO), hydrocarbons (HC), and sulfur oxides (SOx), which are known to cause health and environmental problems. In an effort to address this, the International Maritime Organization (IMO) has been promoting energy efficiency improvements and developing strategic measures to reduce ship emissions. Since 2013, IMO has established two standards for ship fuel efficiency. These are known as the Energy Efficiency Design Index (EEDI) and the Energy Efficiency Operational Indicator (EEOI) [1].

As a result, optimization of ship performance during the ship design process to achieve good hull form and propulsion system, so as to reduce resistance and increase the propulsive efficiency have attracted more attention. Nevertheless, for the existing vessel, there are not many possibilities to change the hull form or improve the propulsion system. Of course, it might be possible to do that but the costs would be unacceptable in most cases [2]. To reduce emissions, ships need to consume less fuel or operate in a fuel-efficient manner MEPC70 guides to decrease the ships' fuel consumption, starting from ship handling, voyage planning, improved fleet management, etc [3]. The ship's handling could be from utilizing ship Turnaround Time (TRT) in port to the trimming of the ship [4]. Specifically, about ship trimming, it could decrease the total exhaust gasses from ships by reducing the Wetted Surface Area (WSA), thus decreasing its resistance, fuel oil consumption, and in the end, decreasing the exhaust gasses emitted by the ship [5].

Trim optimisation is one of the easiest and cheapest methods for ship performance optimisation and fuel consumption reduction. It does not require any hull shape modification or engine upgrade. The optimisation can be done by proper ballasting or choosing of proper loading plan [6]. FORCE Technology is a leading consultant in the trim optimisation, where trim tests have been performed for almost 300 vessels including tankers, container vessels, LNG carriers, Ro-Ro vessels, ferries with the majority being however container vessels. Testing made so far shows possible fuel savings of up to 15% at specific conditions compared to even keel. In overall fleet operations, typical savings can be as high as 2 to 3% [6]. The methodology for studying trim optimization measure is based on the fact that when a vessel is trimmed, the following parameters of ship geometry will change compared to even keel condition: submerged hull form, especially at bow and stern; wetted surface area; length of waterline. All of these factors have effects on ship resistance at a specific speed and loading condition. Thus, by studying influence of trim on resistance of the vessel, ship designers will be able to provide the captain with the best configuration for trim at a specific draft and speed from the point of view minimum resistance. The key for trim optimization measure is to predict the resistance accurately and efficiently [2].

Generally, ship resistance is calculated using model experiments in a towing tank. However, the required costs are substantial. Numerical methods using ship design software can be one of the solutions for calculating ship resistance[7]. Computational Fluid Dynamics (CFD) is a scientific discipline of numerical fluid dynamics solution, in addition to pure theoretical and experimental approaches. The fundamental concept of using CFD-based software is the numerical solution of fluid equations, namely the Navier-Stokes Equations, based on the principles of mass conservation, momentum conservation, and energy conservation [8]. In recent years, some tools of CFD have been widely developed and involved in the application of ship design and ship research. Then, these CFD already have been used in the practical ship design for predicting flow around the hull, flow separations, wave contour, water resistance, wake field, hull-wave interaction, etc. Although CFD has been developed progressively in the past sixty years and widened available, it shall more be progressed in the future [9]. Concurrent with CFD development and enhancement by some researchers and ship design consultants, CFD workshop in ship hydrodynamics had been conducted by International Towing Tank Conference (ITTC) since 1980. ITTC [10] [11] provides a good practice guideline which can be applied to the most ship hydrodynamic application. Regardless, the validation of the numerical simulations is an essential way for the interpretation of the results, and the identification of those aspects of simulation is an effort that has to be improved continuously.

Based on those explanations previously, it must be noted that the progress of CFD for the investigation of ship hydrodynamics has been well conducted, nevertheless, these will be always developed and enhanced in future work to obtain a satisfied result. Furthermore, OpenFOAM, as open source, has been used rapidly by researchers and industries. On the other side, the open source, namely Autodesk CFD has a discretization method, Finite Element Method (FEM), and this seems different compared with OpenFOAM. The free Autodesk CFD is for students and educator’s version [12]. Therefore, in this research, an investigation is carried out on the effect of trim on ship resistance, fluid velocity around the ship, flow patterns, and pressure distribution acting on the model surface using Autodesk CFD [13].

2. Methods

2.1. Ship Data and Model Scale

Data used in this research is secondary data obtained indirectly or from existing sources, such as the main dimensions of the ship and body plan. The body plan of the actual ship is shown in Figure 1.

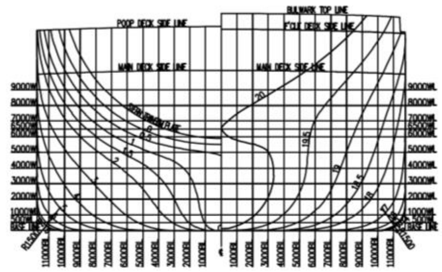


Figure 1. Body Plan of the Actual Ship

The actual ship which was employed in this simulation is conatiner type. The principal particulars of the actual ship and the scaled models are provided in Table 1. The ship model scale is 1:50.

Table 1. Main dimensions of the Ship

Description	Actual Ship	Ship Model
Length Over All / LOA (m)	98.90	1.98
Length Between Perpendicular / LBP (m)	92.00	1.84
Length Water Line / LWL (m)	94.50	1.89
Breadht / B (m)	23.50	0.47
Depth / H (m)	10.00	0.20
Draught / T (m)	6.50	0.13

2.2. Ship Trim

Ship trim could be defined through the following convenient formula Eq. (1):

$$Trim = \text{Ta - Tf} \quad (1)$$

If the trim value is positive (+) it indicates the ship is trim by stern, whereas if the value is negative (-) it indicates the ship is trim by bow.

The Numerical Simulation in this study was conducted for several variations of ship trim conditions while maintaining the same displacement and velocity values. Before determining the trim variations, the actual trim that occurs during operation was first determined. After obtaining the actual trim value when the ship is in operation, according to the scope of this research, the trim variations were divided into the even keel condition, three bow trim conditions, and three stern trim conditions. The results can be seen in Table 2.

Table 2. Determination of Trim Variations

Displacement (12048 T)							
	Trim by bow			Even Keel	Trim by Stern		
Condition	**1**	**2**	**3**	**4**	**5**	**6**	**7**
Trim (m)	-1.441	-1	-0.5	0	0.5	1	1,441
Trim (°)	-0.8974	-0.6228	-0.3114	0	0.3114	0.6228	0.8974
Tf (m)	7.218	7.001	6.752	6.5	6.250	5.997	5.771
Ta (m)	5.777	6.001	6.252	6.5	6.750	6.997	7.212

2.3. Domain and Boundary Condition

The domain and boundaries used in the hydrodynamic profile simulation follow the regulations provided by ITTC regarding geometry. These regulations include the distance between the boundary and the model used. ITTC specifies that the distance between the boundary inlet and the model should be 1-2 L, the distance between the boundary outlet and the model should be 3-5 L, the distance between the boundary wall or symmetry located beside the model should be 1-2 L, and the distance between the bottom and the model should be 1-2 L. [10] [11]. Below are the images and dimensions resulting from the creation of the domain and boundaries based on the regulations provided by ITTC.

The boundary condition phase is the process of inputting velocity, pressure, and slip/symmetry conditions to the walls of the testing tank. Velocity is defined as the fluid velocity with the same value as the ship's velocity placed on the front wall and opposite the direction of the ship model (inlet), while pressure is placed on the back wall (outlet), and on the side, top, and bottom walls of the ship model are conditioned as slip/symmetry, meaning that on these wall surfaces, the fluid is allowed to flow freely. The visualization of the boundary conditions can be seen in Figure 2.

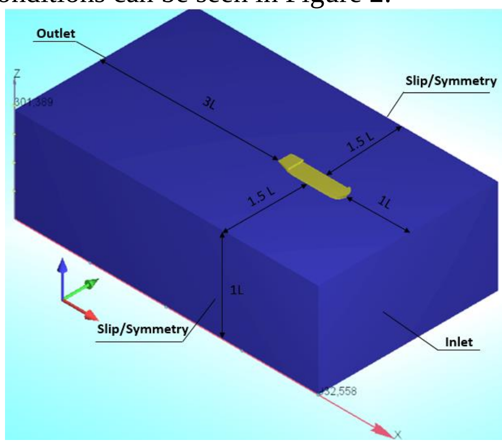


Figure 2. Domain Dimension and Boundary Condition

2.4. Overview of Autodesk CFD

Autodesk CFD software perfoms Computational Fluid Dynamics (CFD) simulations, enabling engineers and anaylyst to make informade prediction regarding the behavior of liquids and gases [12]. This software uses the Finite Element Method (FEM). The FEM is a particular numerical method for solving partial differential equations in two or three space variables and to predict the behaviour of each element. Autodesk CFD uses FEM primarily because of its flexibility in modeling any geometric shape such as linear for 3D tetrahedral elements (unstructured grids) wherein Galerkin's method of weighted residuals is generally used. Hence, the geometric flexibility inherent in finite elements has been maintained in Autodesk CFD [9].

The governing equations for fluid flow are the external incompressible flow of the Navier-Stokes or momentum equations [14]. The governing PDES for continuity equation can be written as:

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho v}{\partial x} + \frac{\partial \rho v}{\partial y} + \frac{\partial \rho w}{\partial z} = 0 \tag{2}$$

Where, ρ is the density, t is the time, u is the velocity component in x-direction, v is the velocity component in y-direction, and w is the velocity component in z-direction. Then, X-Momentum, Y-Momentum, and Z-Momentum equations are derived from the continuity equation as follows. X-Momentum is given:

$$\frac{\partial u}{\partial t} + \rho u \frac{\partial u}{\partial x} + \rho v \frac{\partial u}{\partial y} + \rho w \frac{\partial u}{\partial z} = \rho g_x - \frac{\partial u}{\partial t} + \frac{\partial u}{\partial x}\left[2\mu \frac{\partial u}{\partial x}\right] + \frac{\partial}{\partial y}\left[\mu\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)\right] + \frac{\partial}{\partial z}\left[\mu\left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right)\right] + S_\omega + S_{DR} \tag{3}$$

Y-Momentum equation is given:

$$\rho \frac{\partial v}{\partial t} + \rho u \frac{\partial v}{\partial x} + \rho v \frac{\partial v}{\partial y} + \rho w \frac{\partial v}{\partial z} = \rho g_y - \frac{\partial p}{\partial y} + \frac{\partial}{\partial x}\left[\mu\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)\right] + \frac{\partial}{\partial y}\left[2\mu \frac{\partial v}{\partial y}\right] + \frac{\partial}{\partial z}\left[\mu \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right] + S_\omega + S_{DR} \tag{4}$$

Z-Momentum equation is given:

$$\frac{\partial w}{\partial t}+\rho u\frac{\partial w}{\partial x}+\rho v\frac{\partial w}{\partial y}+\rho w\frac{\partial w}{\partial z}=\rho g_z-\frac{\partial p}{\partial z}+\frac{\partial}{\partial x}\left[\mu\left(\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x}\right)\right]+\frac{\partial}{\partial y}\left[\mu\left(\frac{\partial v}{\partial z}+\frac{\partial w}{\partial y}\right)\right]+\frac{\partial}{\partial z}\left[2\mu\frac{\partial w}{\partial z}\right]+S_\omega+S_{DR} \quad (5)$$

Where, gx, gy, gz are the gravitational acceleration in x, y, z directions, μ is the viscosity, Sω rotating flow, and SDR is the distributed resistance term.

The two source terms in the momentum equations are Sω for rotating coordinates and SDR distributed resistances, respectively. The distributed resistance term can be written in general as:

$$S_{DR}=-\left(K_I+\frac{f}{D_H}\right)\frac{\rho V_i^2}{2}-C\mu V_i \quad (6)$$

Where, V is the velocity, i refers to the global coordinate direction (u, v, w momentum equation), f is the friction factor, d is the hydraulic diameter, C is the permeability. K-factor term can operate on a single momentum equation at a time because each direction has its own unique K-factor. The other two resistance types operate equally on each momentum equation.

The other source term is for rotating flow. This term can be written in general as:

$$S\omega=-2\partial\omega i\times Vi-\partial\omega i\times\omega i\times ri \quad (7)$$

Where, ω is the rotational speed and r is the distance from the axis of rotation.

For turbulence models, this study uses the wall function or k-epsilon (k-ε), k is the kinetic energy per unit mass and ε is the turbulent dissipation wherein it is suitable for the interactions of the external incompressible flow with complex geometry.

2.5. Mesh Strategy

Mesh sizing is the process of dividing the geometry of the model into small parts called elements. The method used is the Finite Element Method (FEM), in CFD fluid analysis, meshing is performed on the fluid and ship model, where the fluid area in contact with the ship model has a high mesh density. The mesh density in the area that contacts the fluid is increased to obtain the appropriate Friction resistance value with the correct Convergence.

Mesh independence study is a method of investigating, whether the simulation results are independent of the underlying mesh or not. A Mesh independence study consists of running the same simulation using grids with different resolutions and analyzing how much the converged solution changes with each mesh.

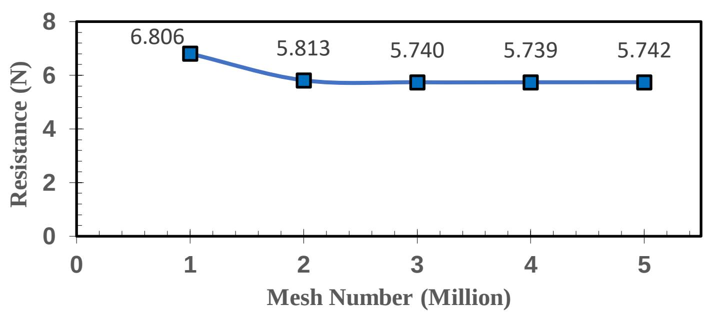


Figure 3. Compared Resistance with Mesh Count.

Based on Figure 3, it can be seen that for Element counts above 3 million, the resistance becomes more linear with increasing element count. And it can be concluded that using a mesh with a count of 3 million will result in the same resistance as a mesh with a count of 4 million, 5 million, or higher; which requires significantly more computational effort compared to the 3 million mesh.

3. Result and Discussion

3.1. Values of Residuals In and Residuals Out

Based on the simulation results that have been conducted, validation results were obtained based on the residual in and residual out values, where the residual out values are in the range of 10^{-8} and the residual in values are larger than the residual out values. This indicates that the calculations have converged. The values of residual in and residual out depend on the mesh density of the model and the boundary layer. The results of residual in and residual out from the conducted simulations can be seen in Table 3.

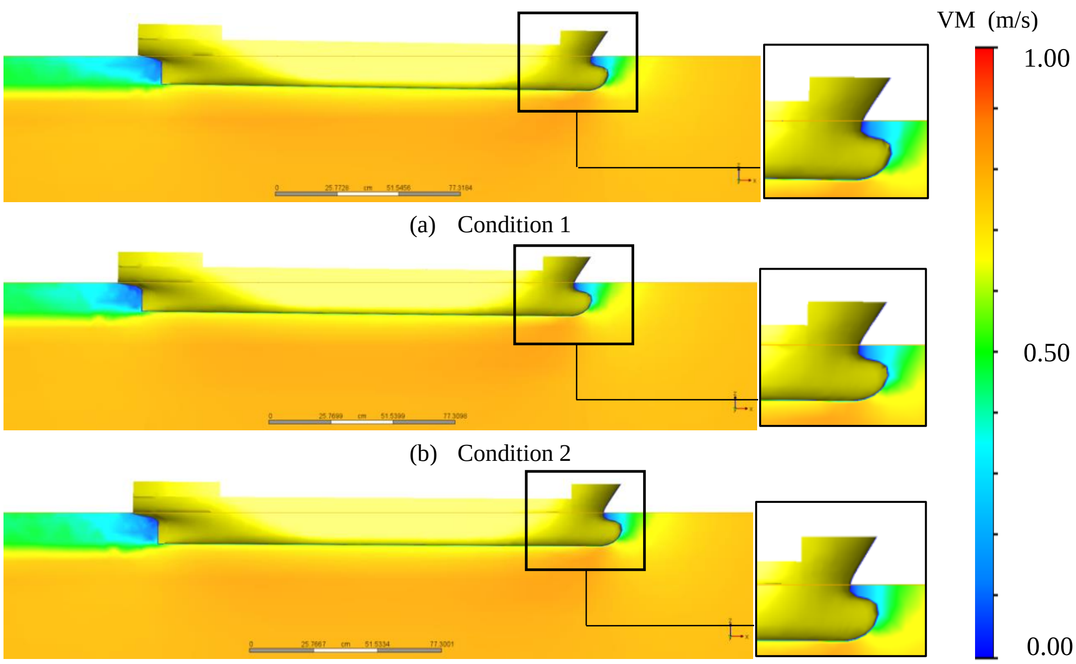
Table 3. The Residual In and Residual Out Values of the Ship Model under Various Trim Conditions

Condition	Trim (m)	Residual In	Residual Out
1	-1.441	1.7414e+00	1.7587e-08
2	-1	1.9243e+00	1.9538e-08
3	-0.5	1.7230e+00	1.8179e-08
4	0	1.7000e+00	1.7002e-08
5	0.5	1.8292e+00	1.9295e-08
6	1	1.7913e+00	1.8883e-08
7	1.441	1.8247e+00	1.9598e-08

Based on Table 3, it can be concluded that the simulation process meets the success criteria for Autodesk CFD analysis.

3.2. Visualization of Velocity Magnitude

The visualization of velocity magnitude shows the fluid flow velocity occurring on the ship model hull. This needs to be understood to see its effect on resistance due to fluid flow.



(a) Condition 1

(b) Condition 2

(c) Condition 3

Figure 4. Visualization of Velocity Magnitude of Trim by Bow Condition

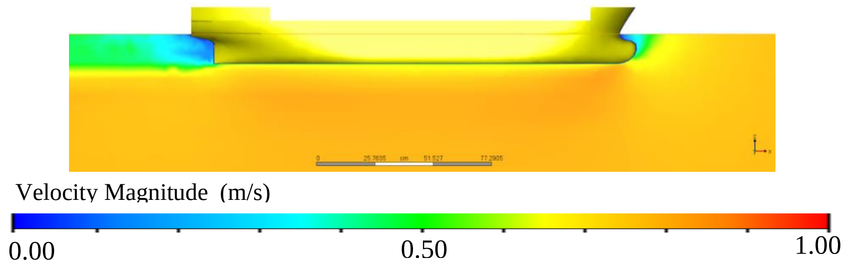
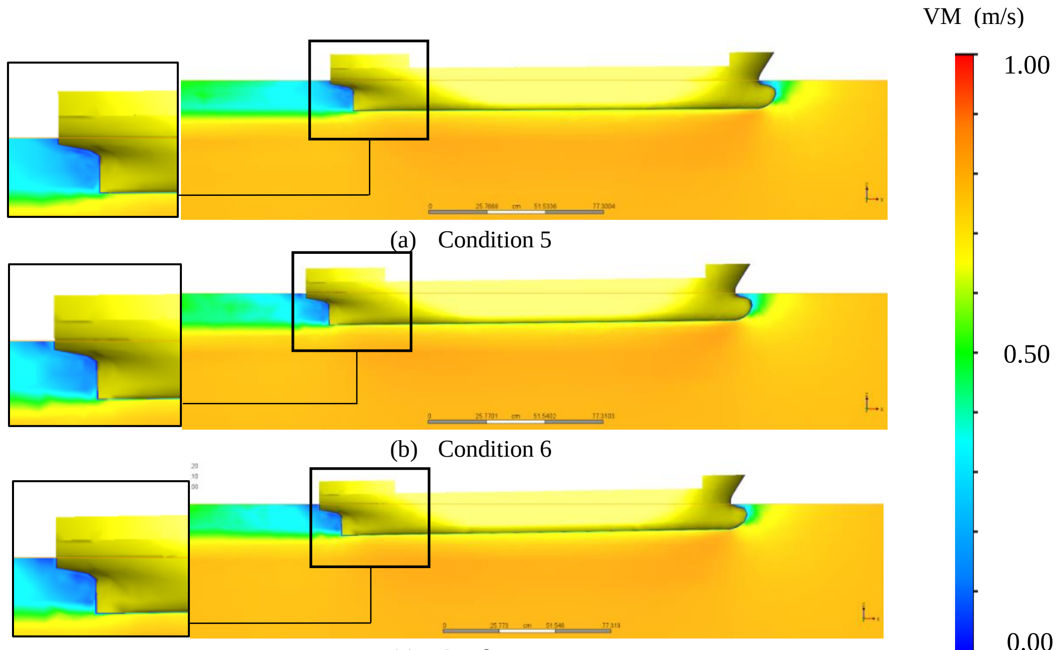


Figure 5. Visualization of Velocity Magnitude of Condition 4 (Even Keel)



(a) Condition 5

(b) Condition 6

(c) Condition 7

Figure 6. Visualization of Velocity Magnitude of Trim by Stern Condition

Based on Figure 4, 5 and 6, the simulation results in the trim by bow, even keel and trim by stern conditions, it can be observed that there are similarities at the front part of the ship model, the fluid velocity is low to moderate, indicated by light blue and green colors with a range of values from 0.2 m/s to 0.55 m/s. After that, the velocity increases on the side of the ship's hull to approximately 0.65 m/s to 0.8 m/s, marked by the yellow color. Meanwhile, at the stern part of the ship model, the fluid velocity decreases again, marked by green and light blue colors with a range of values from 0.1 m/s to 0.55 m/s. In a small portion of the stern tip of the ship model, the fluid velocity approaches zero, indicated by the dark blue color.

The streamline flow pattern around the ship hull will result in variations in flow velocity. This is caused by the boundary layer, which forms when the flow is attached to or very close to the ship model. In this region, the velocity tends to decrease or even stop completely. The halt in flow around the ship model is due to the friction between the fluid and the ship model. Then, due to the law of mass conservation which mandates that the average velocity remains constant throughout the flow, as one moves away from the ship model, the fluid velocity must increase to compensate for the near-zero velocities around the ship model. This velocity continues to increase until it returns to a uniform state under certain conditions. Flow velocity is also closely related to pressure distribution. According to potential flow theory, the ship's cross-section experiences high pressure and low velocity at the front. In the middle section, velocity increases and pressure decreases, while at the rear, pressure increases again and fluid velocity decreases. This affects ship resistance.

3.3. Visualization of Static Pressure

The fluid flow moving at a certain velocity along the bow to the stern of the ship causes an increase in static pressure distribution in certain areas along the submerged hull of the ship. The simulation results showing the visualization of static pressure can be observed in Figure 7, 8 and 9.

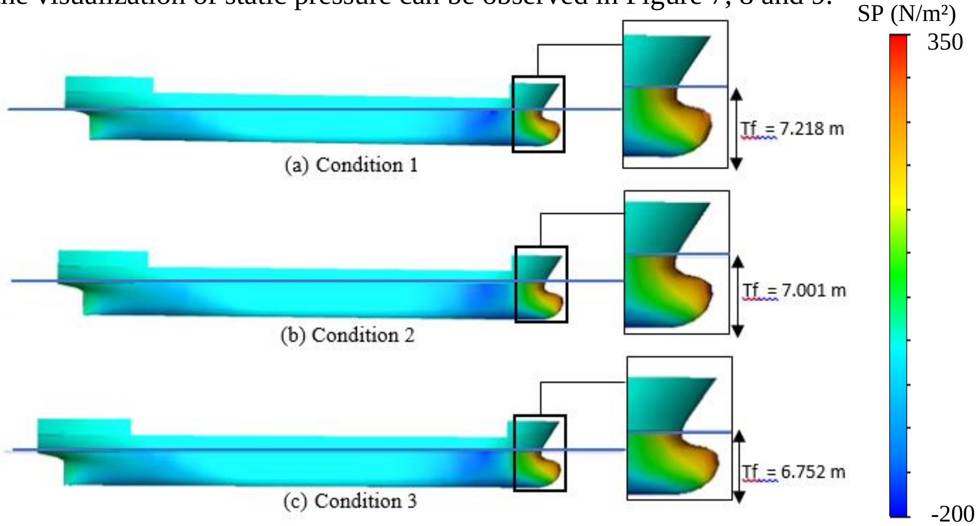


Figure 7. Visualization of Static Pressure of Trim by Bow Condition

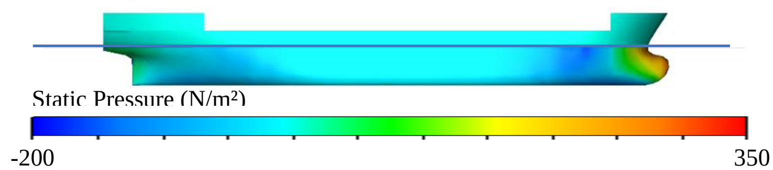


Figure 8. Visualization of Static Pressure of Condition 4 (Even Keel)

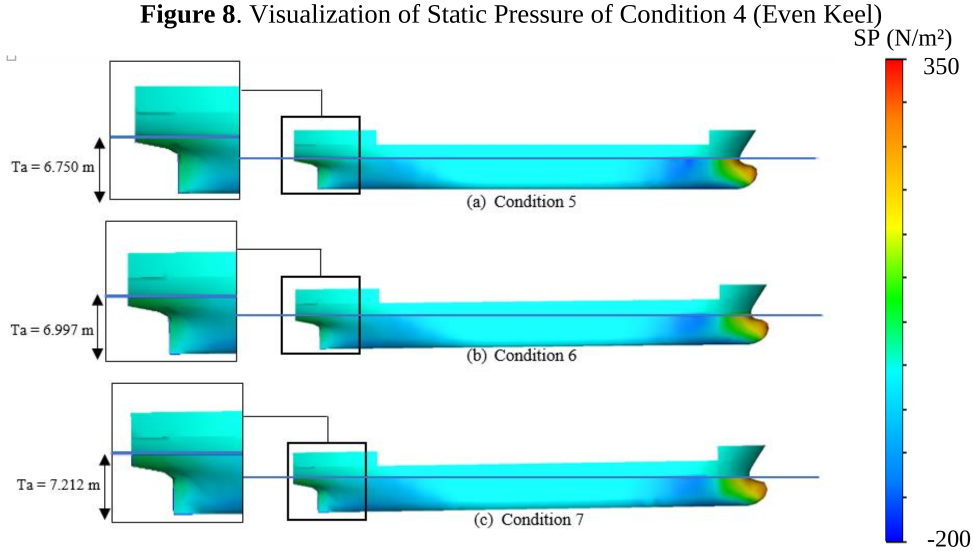


Figure 9. Visualization of Static Pressure of Trim by Bow Condition

Based on Figure 7, 8 and 9, it can be seen that there is a distribution of static pressure along the ship model. At the bow of the ship model, the static pressure distribution has a relatively high level, ranging from 150 N/m² to 300 N/m². After that, the static pressure starts to decrease and stabilize along the side of the ship model, with a range of approximately -50 N/m² to -150 N/m². At the stern of the ship model, the static pressure increases again, marked by a range of values between 100 N/m² to 150 N/m². This is also in accordance with Bernoulli's principle, which states that an increase in the velocity of fluid flow will cause a simultaneous decrease in static pressure.

Similar to the velocity magnitude, the difference between the three conditions trim by bow, even keel, and trim by stern is the area indicating the level of static pressure distribution along the hull of the ship model. In the bow area, the trim by bow condition has a wider area with relatively high static pressure compared to the even keel and stern trim conditions and has the highest static pressure value, which is 302.29 N/m² in the trim by bow condition of -1.441 m. At the stern of the ship model, the trim by stern condition has a wider area with a medium level of static pressure compared to the trim by bow and even keel conditions. This is due to the presence of the submerged transom area when in the stern trim condition. The difference in static pressure distribution will impact the magnitude of the pressure drag and the coefficient of drag along the hull of the ship model.

3.4. Ship Model Resistance for each Trim Variation

The results of the ship model resistance in all values conditions from the conducted CFD simulations can be seen in Table 4.

Table 4. Comparison of Ship Model Resistance for each Trim Variation

Condition	Trim (m)	Velocity (m/s)	Resistance (N)
1	-1.441	0.8	6.044
2	-1	0.8	5.926
3	-0.5	0.8	5.822
4	0	0.8	5.740
5	0.5	0.8	5.674
6	1	0.8	5.690
7	1.441	0.8	5.846

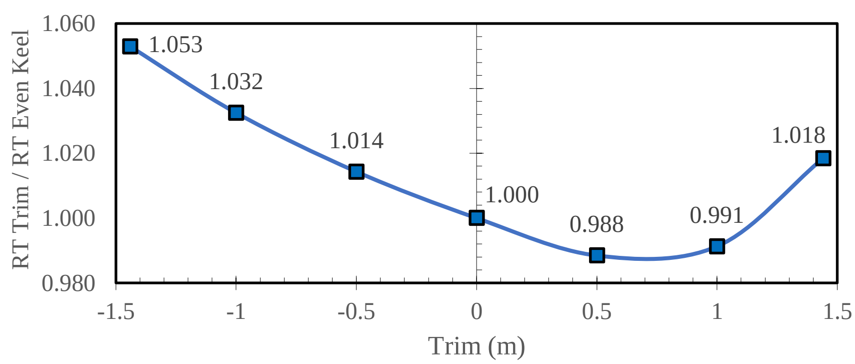


Figure 10. Relationship of Trim Conditions and RT Trim/RT Even Keel

Based on Figure 10, it can be seen that the ship model resistance in the trim by bow conditions of -1.441 m has the highest resistance with a value of 6.044 N. Meanwhile, the trim by stern condition of -0.5 m has the lowest resistance with a value of 5.674 N. For the trim by bow conditions, it can be observed that the resistance in the -0.5 m trim by bow condition increases by 1.4% compared to the even keel condition. Furthermore, the -1 m trim by bow condition increases by 3.2%. Similarly, the -1.441 m trim by bow condition increases by 5.29%. Therefore, the average increase in resistance for the trim by bow condition compared to the even keel condition is 3.32%. For the trim by stern conditions, it can be observed that the resistance in the -0.5 m trim by stern condition decreases by 1.16% compared to the even keel condition. Similarly, the -1 m stern trim condition decreases by 0.87%. However, the -1.441 m trim by stern condition increases by 1.84% compared to the even keel condition. Therefor, the average change in resistance for the trim by stern condition compared to the even keel condition is a decrease of 0.06%.

4. Conclusions

The hydrodynamic analysis of the ship using the Autodesk CFD application has been successfully conducted, with dynamic parameters such as velocity magnitude, static pressure distribution on the hull and resistance obtained. The resistance of the ship model in both trim by bow and trim by stern conditions shows the same tendency to increase with the increase in trim value. However, when comparing the resistance of the ship model in each trim condition with the resistance of the ship model in even keel condition, there is a difference. The resistance in the trim by bow condition increased by an average of 3.32% compared to the even keel condition, whereas the resistance in the trim by stern condition decreased by an average of 0.06% compared to the even keel condition.

References

[1] D.-W. Seo, H.-S. Park, and K.-M. Han, "Analysis of resistance performance for various trim conditions on container ship using CFD," *J. Ocean Eng. Technol*, vol. 29(3), pp. 224–230, 2015.

[2] T.-H. Le *et al.*, "Numerical investigation on the effect of trim on ship resistance by RANSE method," *Appl. Ocean Res.*, vol. 111(2), pp. 1–12, 2021.

[3] Marine Environment Protection Committee (MEPC), Resolution MEPC.282(70) - 2016 Guidelines for the Development of a Ship Energy Efficiency Management Plan (SEEMP), in International Maritime Organization (IMO) Resolutions, 2016.

[4] A. Fadillah, S. Pusaka, Y. Manullang, A. Dewanto, and D. Faturachman, "Strategy for reducing pollutant emissions from ship activities at the port of Tanjung Perak," *WSEAS Trans. Environ. Dev.*, vol. 2(5), p. 155 162, 2015.

[5] Undap, Robin, and A. Fadillah, "The Effect of Trim on Tanker, Container and Bulk Carrier Ship Toward the Reduction of Ship's Exhaust Gas Emission," *J. Ilmu Pengetah. dan Teknol. Kelaut.*, vol. 18(2), pp. 58–68, 2021.

[6] Reichel, A. Maciej, Minchev, and N. L. Larsen, "Trim optimisation-theory and practice," *TransNav Int. J. Mar. Navig. Saf. Sea Transp.*, vol. 8(3), pp. 1–9, 2014.

[7] Erik Sugianto and A. Winarno, "Computational model tahanan kapal untuk menentukan kebutuhan daya kapal bulk carrier 8664 DWT," *J. Kelaut.*, vol. 10(2), pp. 21–29, 2917.

[8] W. Haribowo, S. Samuel, and E. S. Hadi, "Analisa Hambatan Pada Variasi Bentuk Lambung Kapal Ikan Tradisional Catamaran Dengan Metode Cfd," *J. Tek. Perkapalan*, vol. 4(1), pp. 21–29, 2016.

[9] S. Baso, A. Ardianti, and A. A. Rosmani, "An Extended Validation Of Free Cfd Application To Ship Resistance Prediction For Using In Preliminary Design Stage," *J. Eng. Sci. Technol.*, vol. 16(3), pp. 2544–2561, 2021.

[10] ITTC, *procedures and guidelines: practical guidelines for ship CFD applications, 7.5*. USA: Boulder, 2011.

[11] ITTC, "Practical guidelines for ship CFD applications (No. 7.5-03-0203)," *ittc.info*, 2014. https://ittc.info/media/4198/75-03-02-04.pdf.

[12] https://www.autodesk.com/product/cfd/overview

[13] ITTC, "Practical guidelines for ship resistance CFD (No. 7.5-03-0204)," *ittc*, 2019. https://ittc.info/media/4198/75-03-02-04.pdf

[14] Autodesk CFD, "Learning guide: General fluid flow and heat transfer equations," *Autodesk*, 2019.