eISSN: 2721-5717; pISSN: 2747-2124 https://journal.unhas.ac.id/index.php/zonalaut

ZONA LAUT

JOURNAL OF OCEAN SCIENCE AND TECHNOLOGY INNOVATION

Comparison of Resistance Values for Various Patrol Vessel Step Hull Geometry Variations Using Computational Fluid Dynamics (CFD)

*Fitri Hardiyanti, Agung Prasetyo Utomo, Athallah Bintang Rahagi, I Putu Arta Wibawa, Desi Tri Cahyaningati, Aang Wahidin, Miftachuddin
Departement of Shipbuilding Engineering. Politeknik Perkapalan Negeri Surabaya, Indonesia
*fitrihardiyanti@ppns.ac.id

Abstract

This study investigates the effect of stepped hull geometry variations on the hydrodynamic resistance of a patrol vessel operating in Morowali waters. Computational Fluid Dynamics (CFD) simulations were conducted to evaluate the resistance performance of an existing hull and eight stepped hull configurations at an operating speed of 30 knots. The parameters investigated include step spacing, spray strip angle, and spray strip configuration. The results demonstrate that the application of stepped hull geometry generally reduces total resistance compared to the baseline hull. The optimal configuration, Model S3 with a 1.0 m step spacing and a 60° spray angle, achieved the lowest resistance of 28,384.1 N, corresponding to a reduction of 7.34%. From an engineering perspective, this reduction is significant, as lower resistance directly contributes to improved fuel efficiency, extended operational range, and enhanced endurance of patrol vessels during maritime surveillance operations.

Keywords: *CFD, hydrodynamic resistance, patrol vessel, stepped hull, resistance reduction*

1. INTRODUCTION

Indonesia is an archipelagic country with extensive maritime areas that require effective and continuous surveillance. One of the regions experiencing increasing maritime activity is the waters of Morowali, Central Sulawesi, driven by industrial development and shipping traffic growth [11], [21]. These conditions require patrol vessels capable of operating at high speeds while maintaining fuel efficiency and operational endurance.

Patrol vessels are commonly designed with planing hull forms due to their capability to achieve high service speeds. However, conventional planing hulls are often associated with relatively high hydrodynamic resistance, particularly at operational speeds where fuel consumption becomes a critical concern. Increased resistance directly affects propulsion power requirements, fuel usage, and operational range, which are essential performance parameters for patrol vessels conducting long-duration surveillance missions [7], [20].

One effective approach to reducing hydrodynamic resistance in high-speed vessels is the application of stepped hull geometry. The introduction of steps on the hull bottom can reduce the wetted surface area and alter flow separation characteristics, resulting in improved lift generation and reduced total resistance. Previous studies have reported resistance reductions in the range of approximately 5–10%, depending on step spacing, step position, and spray strip geometry [5], [16], [22]. Nevertheless, the hydrodynamic performance of stepped hulls is highly sensitive to geometric parameters and operating conditions, requiring careful evaluation during the design stage.

Despite the growing number of studies on stepped planing hulls, most existing research focuses on generalized hull forms or experimental-scale models, with limited emphasis on patrol vessel applications. In addition, CFD-based comparative investigations addressing the resistance performance of stepped hull patrol vessels operating in Morowali waters remain scarce. Therefore, there is a lack of numerical studies that systematically evaluate the influence of step spacing and spray strip configuration on patrol vessel resistance under representative operational conditions.

Therefore, this study aims to numerically investigate the hydrodynamic resistance characteristics of a patrol vessel equipped with various stepped hull configurations using Computational Fluid Dynamics (CFD). The effects of step spacing and spray strip angle on total resistance are analyzed at an operational speed of 30 knots. The objective of this research is to identify an optimal stepped hull configuration that provides resistance reduction relative to the existing hull, thereby contributing to improved efficiency and performance of patrol vessels operating in Morowali waters.

2. METHODS

2.1. Research Object

The object of this study is a patrol vessel operating in the waters of Morowali, Central Sulawesi. The vessel has the following principal dimensions:

Length Over All (LOA)	: 13.00 m
Beam Moulded (B)	: 3.50 m
Depth Moulded (H)	: 1.40 m
Draft (T)	: 0.6 m
Coefficient Block (Cb)	: 0.40
Displacement	: 9.74 ton
Service Speed (Vs)	: 25.00 knots
Maximum Speed (Vmax)	: 30.00 knots

The data used in this research are actual and valid for the purposes of analysis and simulation. Data collection was carried out through documentation of the technical design and vessel specifications. The general plan of the patrol vessel is shown in Fig. 1.

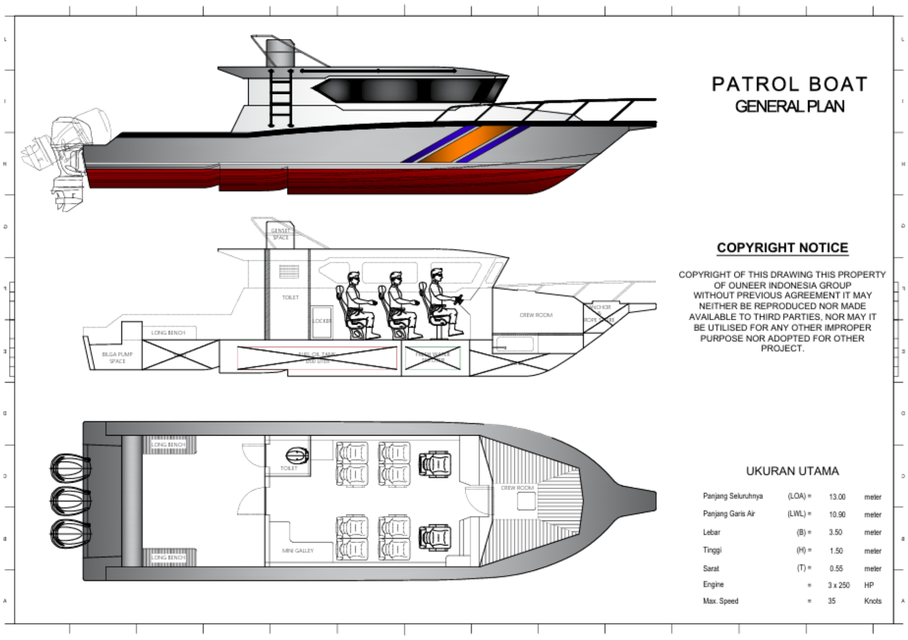


Fig. 1 General Plan Patrol Boat

2.2. Design Variation

The focus of this research is to examine the performance of step hulls under various configurations. The step hull design offers the advantage of reducing the wetted surface area, which directly contributes to lowering the total resistance. The design variations in this study are based on findings from previous research that demonstrated significant effects on the hydrodynamic performance of vessels. The variations considered are as follows:

1. Step spacing: 1 m and 1.5 m. The 1.5 m spacing represents the original vessel configuration, while the 1 m spacing was selected to evaluate the influence of reduced wetted surface area and improved stability, as reported by Niazmand Bilandi et al. (2023).

2. Spray strip angle: 45°, 60°, and 90°, following the profile of Clement (1968) and the design parameters of Savitsky (2007). This variation affects the direction of water spray and the magnitude of drag produced.
3. Number of spray strips: without spray and with upper spray, to assess the effect of adding spray elements on sailing efficiency and total resistance reduction.
4. Step position: shifted from frame 12 to frame 11 (closer to midship) to optimize lift distribution and performance at higher speeds, as suggested by the study of Eka Febrian et al. (2018).

In total, these variations produced eight different step hull models, with the baseline form derived from the existing vessel. The objective of these variations is to identify the step hull configuration that yields the lowest resistance.

2.3. Simulation Method

The hydrodynamic performance of the patrol vessel and its stepped hull variations was evaluated using a Computational Fluid Dynamics (CFD) approach. This method was selected due to its capability to predict flow behavior and resistance characteristics of high-speed vessels with complex hull geometries without the need for physical model testing. All simulations were conducted under identical operating conditions to ensure a consistent comparison between the existing hull and the stepped hull configurations. The simulation flow diagram is shown in Fig. 2.

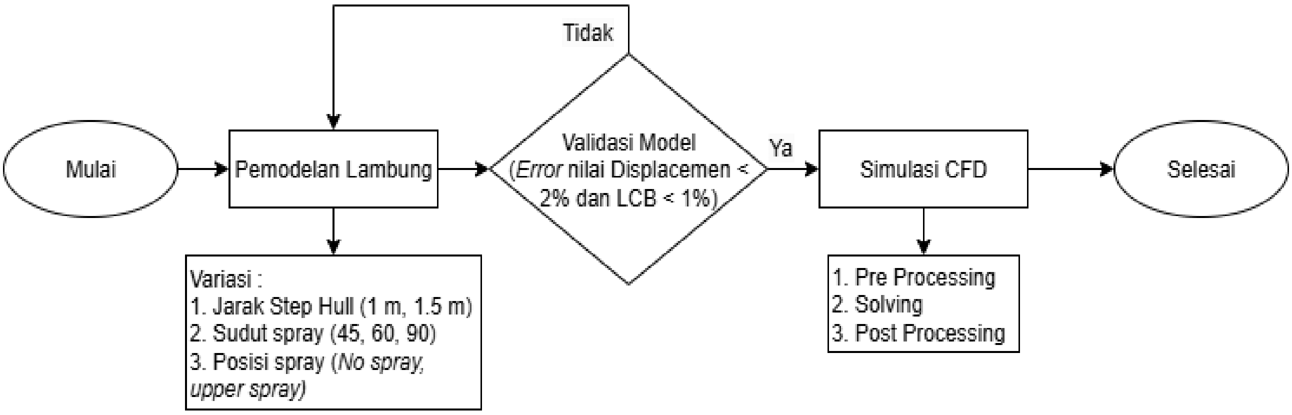


Fig. 2 Flow Diagram

1. Hull Modeling
   3D hull models were developed for the existing vessel and all stepped hull variations based on the principal dimensions of the reference patrol vessel. Prior to the CFD simulations, geometric verification was performed by comparing displacement and longitudinal center of buoyancy (LCB) values to those of the existing hull. This step ensured that any differences in resistance results were attributable to design modifications rather than inconsistencies in the baseline geometry.
2. Computational Domain and Boundary Conditions
   The computational domain was constructed to represent calm water conditions and to minimize boundary effects on the flow field. Domain dimensions were defined following ITTC recommendations, with sufficient clearance upstream, downstream, laterally, and vertically relative to the vessel length. To reduce computational cost while maintaining accuracy, a half-body model was adopted by exploiting the geometric symmetry of the hull.
   Boundary conditions were applied as follows: a uniform velocity inlet corresponding to an operating speed of 30 knots was prescribed at the upstream boundary, while a pressure outlet condition was assigned at the downstream boundary. The hull surface was defined as a no-slip wall, and the remaining domain boundaries were treated as symmetry or far-field conditions. A two-phase flow model was employed to capture the interaction between water and air.
3. Solver Settings
   The simulations were performed using a pressure-based solver with a steady-state formulation, which is commonly adopted for resistance prediction of high-speed vessels operating at constant speed. The Volume of Fluid (VOF) method was applied to model the free-surface flow between water and air.
   Turbulence effects were modeled using the k–ω turbulence model, which is well suited for predicting near-wall flow behavior and free-surface interactions. This model has been widely applied in CFD

studies of planing and stepped hull vessels and provides reliable convergence characteristics for high Reynolds number marine flows.

4. Solution
After meshing, the simulations were executed using a turbulence model commonly applied for high-speed free-surface flows. The simulations were conducted in steady-state and multiphase modes to represent the interaction between air and water. Iterations continued until convergence was achieved, with residuals below 10^{-5}. The simulations were carried out for all step hull variations at a speed of 30 knots, allowing direct comparison of designs under realistic operating conditions.
5. Post-processing
Upon completion, the simulation data were extracted for further analysis. Visualizations such as streamlines, pressure contours, and volume fractions were used to observe flow patterns, pressure distribution, as well as wave and spray formations. The numerical output in the form of total resistance (RT) was compared across all variations to determine the most efficient design in terms of resistance.
6. Validation and Mesh Testing
To ensure simulation accuracy, geometry validation was performed by comparing the displacement and LCB values from the models with the original vessel design data. The acceptable tolerance was a maximum difference of ±2% for displacement and ±1% for LCB. In addition, a mesh independency test was conducted by comparing simulation results across several mesh densities. The simulations were considered valid if the resistance differences between mesh levels did not exceed 5%, ensuring that the results were not affected by grid size variations.

2.4. Materials and Data

This research was conducted computationally without the use of physical models. The materials and data utilized in this study are as follows:

- Vessel geometry data
- Reference literature

3. RESULTS AND DISCUSSION

3.1. Hull Modeling

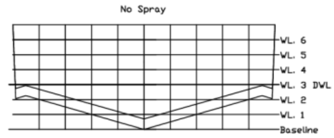
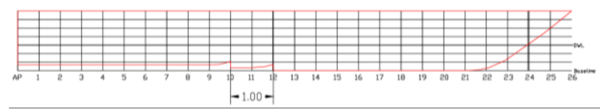
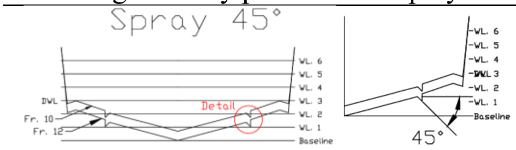
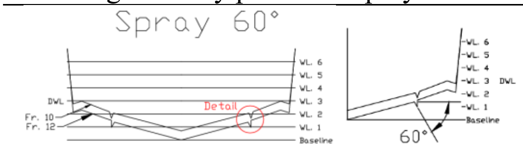
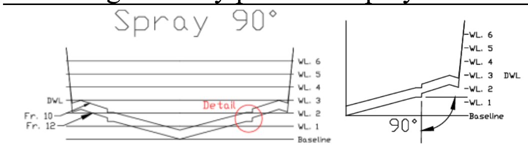
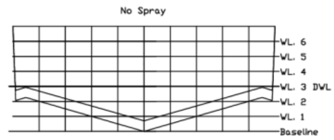
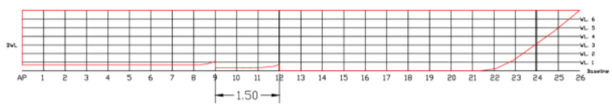
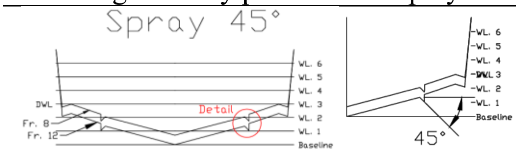
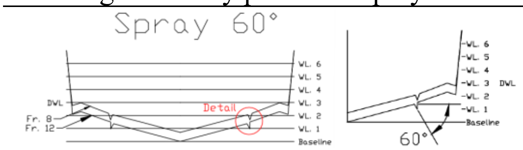
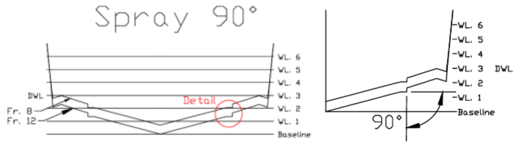
In the modeling process, several variations were applied as described in Section 2.2, resulting in a total of nine vessel model configurations. The details of each variation along with the parameter combinations are shown in Table 1.

Table 1. Model Variations

No	Model	Variation Type		
		Step Distance	Spray Type	Spray Degree
1	Existing Model	1.5 m	Double Spray	90°
2	Model S1	1 m	-	-
3	Model S2	1 m	Upper Spray	45°
4	Model S3	1 m	Upper Spray	60°
5	Model S4	1 m	Upper Spray	90°
6	Model S5	1.5 m	-	-
7	Model S6	1.5 m	Upper Spray	45°
8	Model S7	1.5 m	Upper Spray	60°
9	Model S8	1.5 m	Upper Spray	90°

After the variations of each step hull model were determined, hull modeling was carried out. This process was intended to accurately represent the hull form of each design variation based on the dimensions of the existing vessel. The generated models were then used as the basis for the CFD simulations. The details of each hull model are shown in Table 2.

Table 2. Hull Model

Model	Side View	Body Plan
S1		 Fig. 4 Body plan without spray
S2	 Fig. 3 Side view with 1 meter step distance	 Fig. 5 Body plan with spray 45°
S3		 Fig. 6 Body plan with spray 60°
S4		 Fig. 7 Body plan with spray 90°
S5		 Fig. 9 Body plan without spray
S6	 Fig. 8 Side view with 1.5 meter step distance	 Fig. 10 Body plan with spray 45°
S7		 Fig. 11 Body plan with spray 60°
S8		 Fig. 12 Body plan with spray 90°

3.2. Model Validation

Af ter the design variations were determined, vessel modeling was carried out. During this process, validation was conducted by comparing the results of each variation with the data of the existing step hull vessel. The purpose of this validation was to ensure that the developed models remained consistent with the characteristics of the reference vessel. Complete information regarding the specifications of the existing step hull vessel is shown in Table 3.

Table 3. Data Existing Model

No	Description	Value	Unit
1	Displacement	9,74	t
2	LCB	5,67	from zero pt. (+ve fwd) m

The values in Table 3 were used as references to validate the developed step hull models. Validation focused on displacement and longitudinal center of buoyancy (LCB) to ensure consistency with the reference vessel. This step ensured that differences in simulation results were caused by design variations rather than changes in the baseline hull form. The validation tolerances followed IACS (2021) standards, as summarized in Table 4.

Table 4. Table Tolerance

Parameter	Tolerance
Displacement	+/- 2%
Longitudinal center of buoyancy, from AP	+/- 1% / 50 cm

This tolerance table serves as a reference for correcting the differences between numerical calculations and actual data, ensuring that the hull design continues to comply with operational safety standards. The validation results of the hull modeling for the various design variations are shown in Table 5.

Table 5. Value Validation

Model	Hydrostatic Data		Percentage Correction	
	Disp (ton)	Lcb (m)	Disp (ton)	Lcb (m)
Existing Model	9.74	5.67	-	-
Model S1	9.84	5.694	1.03%	0.42%
Model S2	9.876	5.64	1.40%	0.53%
Model S3	9.89	5.633	1.54%	0.65%
Model S4	9.828	5.644	0.90%	0.46%
Model S5	9.797	5.679	0.59%	0.16%
Model S6	9.825	5.65	0.87%	0.35%
Model S7	9.877	5.642	1.41%	0.49%
Model S8	9.625	5.68	1.18%	0.18%

Based on Table 5, all tested step hull models satisfy the geometric validation criteria against the existing vessel, with displacement values within the maximum tolerance of ±2% and longitudinal center of buoyancy (LCB) deviations below ±1%.

3.3. Resistance Analysis

After the ship models were completed, the next step was to evaluate the resistance values. The results of this analysis were then compared with the data from the existing ship model to identify the design variations that are effective in reducing resistance. The analysis process for each model can be illustrated as follows.

1. Pre Processor
   a. Geometry
   After the modeling process was completed, the next stage was resistance analysis. The first step in the geometry setup involved importing the ship model in the required format. The geometry used as input is shown in Fig. 13.

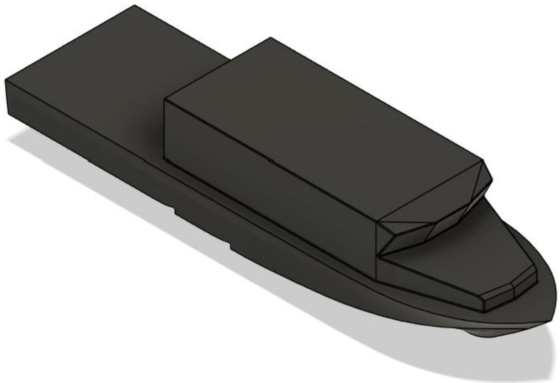


Fig. 13 Model for Geometry Input

b. Domain

After the ship model was successfully imported, the next step was to construct the fluid domain. The size of the domain was determined based on specific boundary standards to ensure that the simulation results closely represented real conditions. Following the recommendations of the ITTC, the distance between the model and the domain boundaries was defined as presented in Table 6.

Table 6. Domain Size

Domain Size		
Description	Size	Input (m)
Front	2 x LOA	26
Back	3 x LOA	39
Side	2 x LOA	26
Top	0,5 x LOA	6.5
Bottom	1.5 x LOA	19.5

The ship model was then modified into a half-body configuration. This approach was selected because ships generally exhibit geometric symmetry between the port and starboard sides.

c. Meshing

To determine the optimal mesh size, a series of tests was conducted using mesh values ranging from the smallest to the largest. In this study, the mesh size was varied from 0.01 m to 0.1 m. The results of the drag force obtained from these tests are presented in Table 7 and Fig. 14.

Table 7. Size Mesh

Size Mesh (m)	Total Drag Force (N)
0.01	172384.3
0.02	56842.91
0.03	143219.1
0.04	10502.94
0.05	11143.88
0.06	11453.23
0.07	7547.366
0.08	11453.23
0.09	20987.66
0.1	198740.6

Meanwhile, Fig. 14 presents the graphical results obtained from the previous mesh size test.

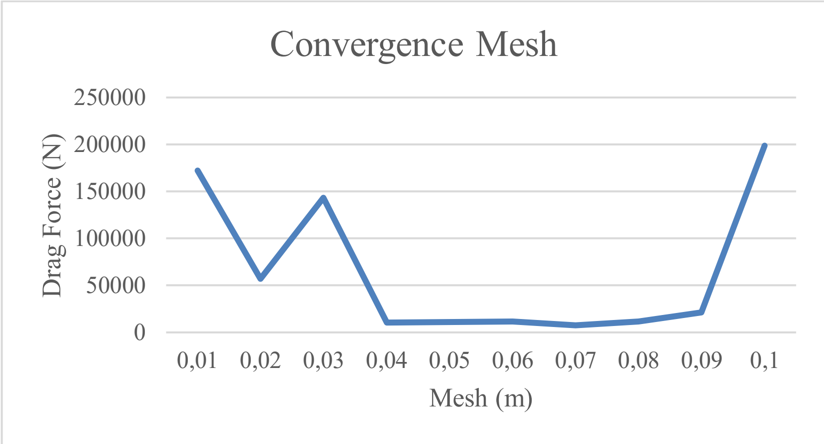


Fig. 14 Mesh Size Convergence Test Graph

Based on Fig. 14, it can be observed that the simulation achieved convergence within the mesh size range of 0.04 m to 0.08 m. The next step is to conduct the analysis using a finer mesh size in order to obtain more accurate simulation results. This process begins with the smallest element size, namely 0.04 m, as presented in the simulation results shown in Table 8 and Fig. 15.

Table 8. Size Mesh

Size Mesh (m)	Total Drag Force (N)
0.041	17,321.62
0.042	8,178.288
0.043	19,041.91
0.044	18,758.51
0.045	11,832.37
0.046	13,881
0.047	10,744.04
0.048	10,589.68
0.049	11,438.27
0.05	11,143.88
0.051	11,062.61
0.052	11,647.4
0.053	18,424.88

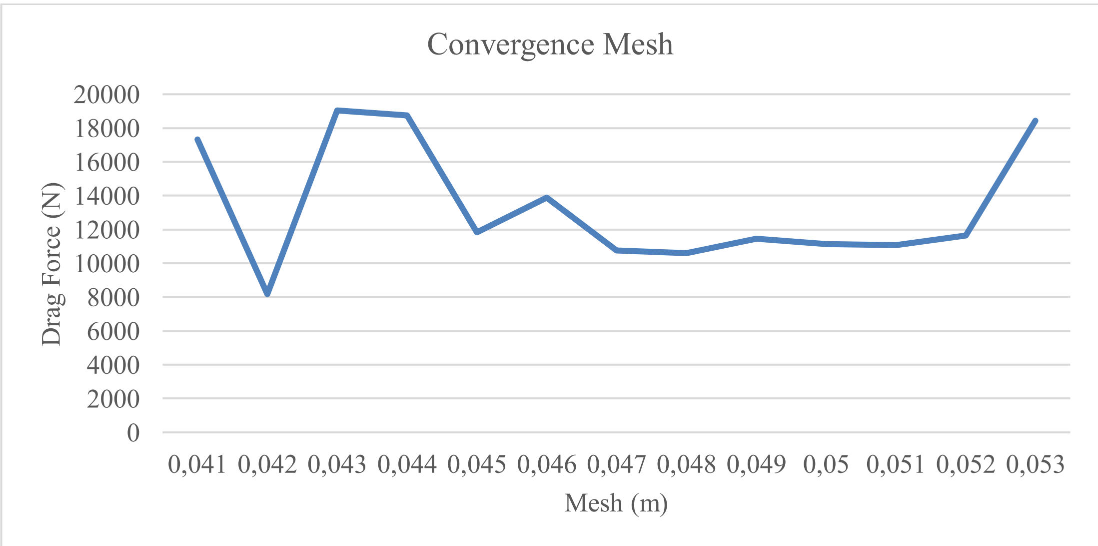


Fig. 15 Mesh Size Convergence Test Graph

Based on Fig. 15, it can be seen that the mesh values have reached convergence within the size range of 0.49 m to 0.51 m. Therefore, this range will be used as a reference for determining the mesh size in the subsequent modeling process.

d. Boundary Condition

The boundary condition configuration in this study includes defining the ship model surface as a ship-wall, while the outer domain boundaries are simulated as calm water to represent sea conditions during the simulation. The vessel speed used is 30 knots, which corresponds to 15.43 m/s after conversion. The outlet serves as the region where the fluid exits the domain after interacting with the ship model. The velocity and water depth at the outlet are set equal to the inlet conditions to maintain flow stability within the simulation domain. The input values for the boundary conditions are shown in Table 9.

Table 9. Boundary Condition

No	Parameter	*Input*
1	Material Type 1	Water
2	Material Type 2	Air
3	Inlet Velocity	15,43 m/s

2. Solver

Table 10 shown the detailed configuration applied during the setup stage.

Table 10. Setup

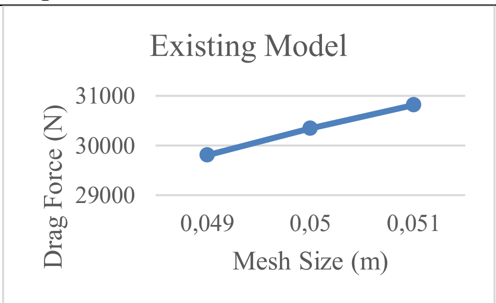
No	Parameter	Pengaturan
1	Solver type	*Pressure-based*
2	Velocity Formulation	*Absolute*
3	Time	*Steady*
4	Models	*Multiphase*: (Volume of Fluid) *Viscous*: K-omega

3.4. Results

1. Resistance Validation

All resistance reductions reported in this study are referenced to the existing hull model, which serves as the baseline configuration under identical operating conditions at a speed of 30 knots. The drag force results must achieve convergent meshing values as validation to ensure that the simulation runs were successfully performed. The resistance values for each model are presented in Table 11.

Table 11. Summary of Resistance Results

Model	Graphic	Description
Existing Model	 Fig. 16 Existing Model Graph	The Existing Model shows a resistance trend that tends to increase with larger mesh sizes, with values ranging between 29,803 N and 30,552 N.

Model S1

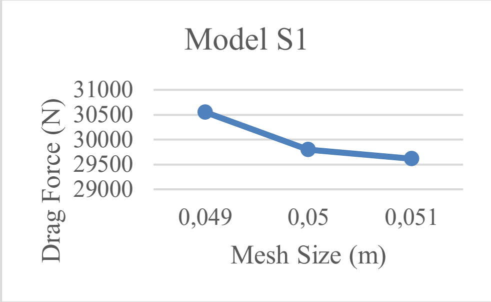


Fig. 17 Model S1 Graph

Model S1 demonstrates a consistently decreasing resistance trend as the mesh size increases, with resistance values ranging from 28,798 N to 29,793 N.

Model S2

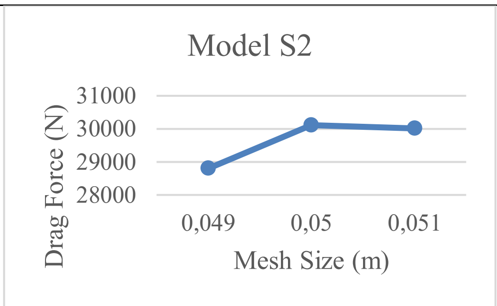


Fig. 18 Model S2 Graph

Model S2 exhibits a resistance trend that increases with mesh size before stabilizing, with values ranging between 28,798 N and 30,112 N.

Model S3

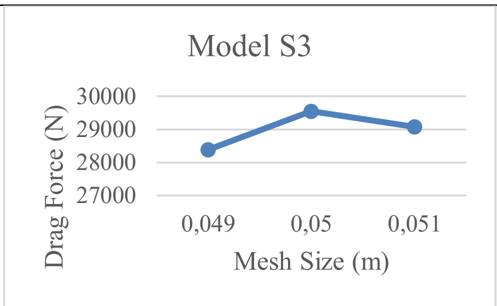


Fig. 19 Model S3 Graph

Model S3 shows a fluctuating resistance trend, increasing at a mesh size of 0.05 and slightly decreasing at 0.051. The resistance values range from 28,384 N to 29,608 N.

Model S4

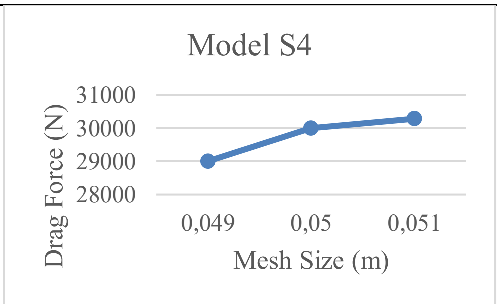


Fig. 20 Model S4 Graph

Model S4 presents a consistently increasing resistance trend with larger mesh sizes, with resistance values ranging between 29,006 N and 30,289 N.

Model S5

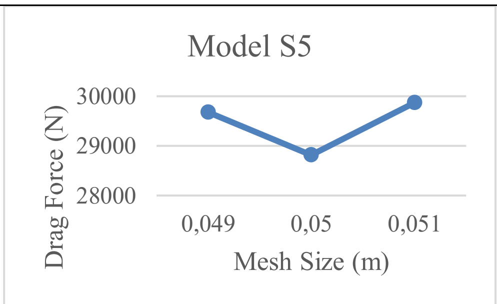


Fig. 21 Model S5 Graph

Model S5 demonstrates a fluctuating resistance trend, decreasing at a mesh size of 0.05 before sharply increasing at 0.051. The resistance values range from 28,815 N to 30,513 N.

Model S6

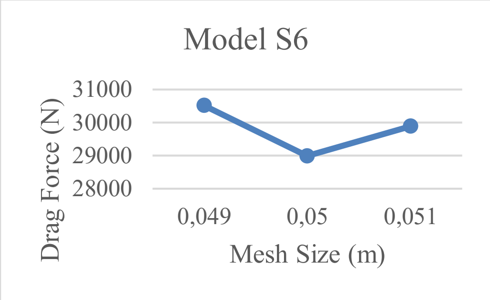


Fig. 22 Graph Model S6

Model S6 shows a fluctuating resistance pattern, decreasing at a mesh size of 0.05 and increasing again at 0.051. The resistance values range from 28,990 N to 30,513 N.

Model S7

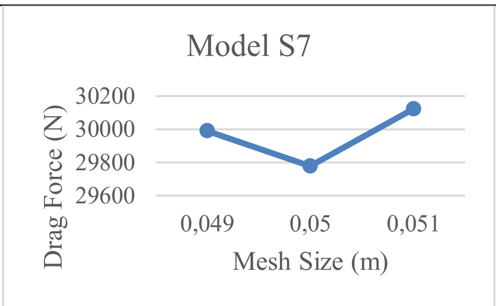


Fig. 23 Graph Model S7

Model S7 exhibits a relatively stable resistance trend with minor fluctuations, decreasing at a mesh size of 0.05 before increasing again at 0.051. The resistance values range between 29,000 N and 30,123 N.

Model S8

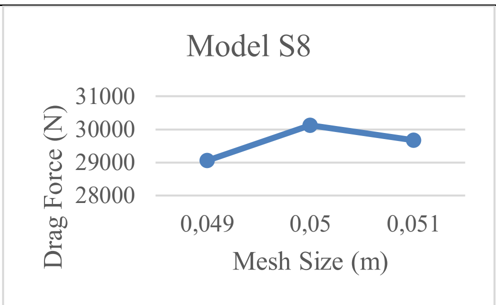


Fig. 24 Graph Model S8

Model S8 presents a fluctuating resistance trend, increasing at a mesh size of 0.05 and decreasing at 0.051. The resistance values range from 29,055 N to 30,121 N.

Table 11 indicates that all models fulfill the validation criteria, with simulation deviations remaining within the 5 percent threshold. The summarized resistance validation results for each model are provided in Table 12, ensuring the reliability and consistency of the numerical analysis.

Table 12. Summary of resistance value

Model	Mesh Size (m)	Drag Force (N)	Percentage
Existing Model	0.049	29,803.3126	0.00%
	0.05	30,339.2835	1.80%
	0.051	30,813.4285	3.39%
Model S1	0.049	30,552.1862	0.00%
	0.05	29,793.32	2.48%
	0.051	29,615.008	3.07%
Model S2	0.049	28,798.63	0.00%
	0.05	30,110.9904	4.56%
	0.051	30,012.76	4.22%
Model S3	0.049	28,384.1072	0.00%
	0.05	29,547.32	4.10%
	0.051	29,067.891	2.41%
Model S4	0.049	29,006.1396	0.00%

	0.05	29,998.71	3.42%
	0.051	30,289.002	4.42%
Model S5	0.049	29,676.6	0.00%
	0.05	28,814.94	2.90%
	0.051	29,870.02	0.65%
Model S6	0.049	30,513.0983	0.00%
	0.05	28,990.44	4.99%
	0.051	29,887.55	2.05%
Model S7	0.049	29,990.441	0.00%
	0.05	29,777.65	0.71%
	0.051	30,123.1798	0.44%
Model S8	0.049	29,054.199	0.00%
	0.05	30,121.7	3.67%
	0.051	29,678.3912	2.15%

Based on Table 12, the simulation results indicate that the process has achieved convergence, as evidenced by the stability of residual values at each iteration. Therefore, the results can be considered valid.

2. Summary of Resstance Result

The following is a summary of the resistance analysis of the Step hull models, as presented in Table 13.

Table 13. Summary Drag Force

Model	Drag Force (N)
Existing Model	29,803.3
Model S1	30,552.2
Model S2	28,798.6
Model S3	28,384.1
Model S4	29,006.1
Model S5	29,676.6
Model S6	30,513.1
Model S7	29,990.4
Model S8	29,054.2

Table 13 indicates that the existing hull model, which serves as the baseline reference, generates a total resistance of 29,803.3 N at an operating speed of 30 knots. Using this baseline, Model S3 achieves the lowest resistance of 28,384.1 N, corresponding to a reduction of 7.34%. This level of reduction is consistent with previous stepped hull investigations, where resistance reductions of approximately 5–8% were reported by Eka Febrian et al. and 6–10% by Niazmand Bilandi et al., depending on step geometry and operating conditions. The agreement with these studies confirms the technical validity of the present results.

The superior performance of Model S3 is primarily attributed to a reduction in wetted surface area and a more favorable lift distribution along the hull bottom. The 60° spray strip angle effectively redirects spray flow away from the hull surface, thereby reducing pressure concentration and spray-induced drag. In addition, the selected step spacing promotes smoother flow separation at the step location, leading to improved flow stability and reduced turbulence intensity in the downstream region.

4. CONCLUSION

Table 13 shows that the drag force of the Existing Model is 29,803.3 N at a speed of 30 knots. To determine the model with the best performance, the selection criterion was based on the resistance values at 30 knots.

According to this criterion, Model S3 recorded the lowest resistance of 28,384.1 N at 30 knots, representing a reduction of 7.34 percent compared to the Existing Model.

This study is limited to steady state CFD simulations conducted at a single operating speed. Dynamic effects such as trim variation, heave motion, and unsteady wave interactions were not considered. Future studies should incorporate transient CFD simulations and experimental validation to further assess the hydrodynamic performance of stepped hull configurations under varying operational conditions.

ACKNOWLEDGMENTS

The author would like to express sincere gratitude to the Shipbuilding Polytechnic of Surabaya for providing the opportunity and facilities to carry out this research. Special thanks are also extended to the lecturers of the Naval Architecture and Ship Design Study Program for their guidance, direction, and moral and material support throughout the preparation of this article. The author also wishes to acknowledge the support of family, fellow students, and all others who cannot be mentioned individually for their prayers, motivation, and assistance, which have contributed to the successful completion of this research. It is hoped that this study will be beneficial and serve as a reference for further development in the field of naval architecture.

BIBLIOGRAPHY

[1] Anderson D., & White. (2018). Planning Hulls In Modern Craft. International Journal Of Marine Engineering.
[2] Baker L., & Smith. (2019). Advanced Boat Design. Nautical Publishing.
[3] Bakhtiari, M., & Ghassemi, H. (2017). Numerical Study Of Step Forward Swept Angle Effects On The Hydrodynamic Performance Of A Planing Hull. Https://Doi.Org/10.17402/228
[4] Caesar Wiratama. Tipe Modul Hull. Scribd. Accessed On July 22, 2025 Dari Https://Www.Scribd.Com/Document/683883543/Tipe-Modul-Hull
[5] Eka Febrian, C., Chrismianto, D., Rindo, G., & Teknik Perkapalan, J. (2018). Jurnal Teknik Perkapalan Analisis Hambatan Dan Gaya Angkat Dari Modifikasi Stephull Dengan Variasi Sudut Pada Kapal Pilot Boat 15 Meter Alu Menggunakan Metode Cfd. Jurnal Teknik Perkapalan, 6(1). Http://Ejournal3.Undip.Ac.Id/Index.Php/Naval
[6] Eko, F., Nugroho, A., Budiarto, U., Wibawa, A., & Santosa, B. (2024). Analisis Pengaruh Penambahan Spray Strips Pada Stepped Planing Hull Terhadap Hambatan Total Menggunakan Metode Computational Fluid Dynamics Spasi-Times New Roman 11 Italic. Jurnal Teknik Perkapalan, 12(4). Https://Ejournal3.Undip.Ac.Id/Index.Php/Naval
[7] Graham, J. (2020). Marine Hull Design. Maritime Press.
[8] Hidayat D., Prasetyo E., & Sari. (2020). Strategi Pengelolaan Keamanan Maritim Di Indonesia. Jurnal Ilmu Kelautan.
[9] Iacs. (2021). Common Structural Rules: Tolerances And Criteria. International Association Of Classification Societies.
[10] Johnson, M. (2021). Ship Stability And Design. Oceanic Books.
[11] Kementerian Lingkungan Hidup. (2020). Laporan Kualitas Lingkungan Hidup.
[12] Kusumastanto, T. (2023). *Potensi Keunggulan Kompetitif Sumber Daya Kelautan Indonesia*. Jurnal Ilmu Kelautan Dan Perikanan, 28(1), 1–10.
[13] Mardiyanto R., Rahman A. A., & Yulianto. (2021). Peran Kapal Patroli Dalam Penegakan Hukum Di Laut. Jurnal Pertahanan.
[14] Miller P. T., & Johnson. (2020). Step Hulls And Their Applications. Journal Of Naval Architecture.
[15] Muckle, W. (1983). Naval Architecture. Longman.
[16] Niazmand Bilandi, R., Dashtimanesh, A., Mancini, S., & Vitiello, L. (2023). Comparative Study Of Experimental And Cfd Results For Stepped Planing Hulls. Ocean Engineering, 280. Https://Doi.Org/10.1016/J.Oceaneng.2023.114887
[17] Purnomo, H. (2012). Rencana Garis Dan Bentuk Lambung Kapal. Politeknik Ilmu Pelayaran.
[18] Radiarta D., Prasetyo E., & Sari. (2015a). Kondisi Kualitas Perairan Di Kabupaten Morowali Provinsi Sulawesi Tengah: Pendekatan Spasial Dan Statistik Multivariat. Jurnal Akuakultur.

[19] Radiarta D., Prasetyo E., & Sari. (2015b). Pengaruh Pencemaran Laut Terhadap Ekosistem Perairan Di Indonesia. Jurnal Lingkungan Dan Pembangunan Berkelanjutan.
[20] Rawson E. C., & Tupper. (2001). Basic Ship Theory. Reed Educational And Professional Publishing.
[21] Syarifuddin, N. (2022). *Pengaruh Industri Pertambangan Nikel Terhadap Kondisi Lingkungan Maritim Di Kabupaten Morowali*. Jurnal Riset Dan Teknologi Terapan Kemaritiman, 3(1), 12–22.
[22] Zubaer, H., Budiarto, U., Iqbal, M., & Hydrodnamika, L. (2018). Jurnal Teknik Perkapalan Analisa Variasi Twin Step Hull Pada Kapal Pilot Boat 15 Meter Alu Dengan Menggunakan Metode Cfd. Jurnal Teknik Perkapalan, 6(1), 295. Http://Ejournal3.Undip.Ac.Id/Index.Php/Navals