Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage:
https://semarakilmu.com.my/journals/index.php/fluid_mechanics_thermal_sciences/index
ISSN: 2289-7879

Journal of Advanced Research Fluid Mechanics and Thermal Sciences
ISSN 2289-7879

Numerical Investigation of Contra Rotating Propeller Efficiency on Ferries through CFD Simulation

Muhammad Iqbal Nikmatullah[1,*], Andi Haris Muhammad[1], Baharuddin[1], Ridha Sulistyawati Rasyid[1], Andi Hirawaty Subhan Putri[1]

[1] Department of Marine Engineering, Faculty of Engineering, Hasanuddin University, Makassar, Indonesia

ARTICLE INFO

Article history:
Received 9 June 2024
Received in revised form 15 September 2024
Accepted 22 September 2024
Available online 10 October 2024

Keywords:
Contra rotating propeller; computational fluid dynamic; thrust; torque; propulsion efficiency

ABSTRACT

This research focuses on the pressing issues of global warming and fossil fuel depletion. It discusses the International Maritime Organization's (IMO) proactive approach to these challenges by introducing the Energy Efficiency Design Index (EEDI). The EEDI is a pioneering initiative that emphasizes the development of propulsion systems incorporating Energy Saving Devices (ESD). A vital feature of this research is the exploration of the contra-rotating propeller, a propulsor configuration that promises higher efficiency than conventional single-propeller propulsion systems. The unique advantage of the contra-rotating propeller lies in its ability to recover rotational energy in the propeller slipstream. This energy recovery produces higher thrust with torque equivalent to conventional systems, thereby enhancing efficiency. The research employed Computational Fluid Dynamics (CFD) software to determine the impact of contra-rotating propellers on propeller efficiency in ferry propulsion systems. The study aimed to identify an optimal propeller placement and diameter combination to maximize efficiency. The findings of this research are promising. The use of contra-rotating propellers on ferry boats resulted in a significant increase in propeller efficiency. Specifically, efficiency increased by 6.498% on model 2 with a diameter ratio of 1:0.87. Similarly, in model 1 with a diameter ratio of 1:1, efficiency increased by 3.374%. These results highlight the potential of contra-rotating propellers in enhancing the energy efficiency of maritime propulsion systems. Therefore, this research provides valuable insights for designing future energy-efficient maritime vessels.

1. Introduction

The efficiency of a propeller, a critical part in ship propulsion, experiences a significant increase of 7.591% when a contra-rotating propeller is employed, as compared to the efficiency achieved by a traditional single screw propeller [1-3]. This enhancement in efficiency is not a trivial matter but rather a substantial improvement that holds considerable implications for the field of marine engineering [4,5].

---

[*] *Corresponding author.*
E-mail address: lakibbal@unhas.ac.id

https://doi.org/10.37934/arfmts.122.1.107117

When the efficiency of various types of ship propulsion equipment is plotted on a graph, the optimal efficiency curve reveals that the results of this research are not merely satisfactory but indeed commendable [6]. This is a testament to the potential of the contra-rotating propeller in revolutionizing the industry.

The Contra Rotating Propellers, a unique type of propeller system, features dual-coaxial propellers mounted on a single axis [7]. These propellers are arranged sequentially, with one positioned in front of the other, and rotate in opposite directions. This innovative design allows for a power saving of approximately 13%, addressing the issue of rotational energy that might otherwise be lost when using a conventional single-screw propeller [2,7].

Furthermore, the versatility of this type of propeller extends to its application on small outboard units, proving its wide-ranging usability and potential for further exploration in various maritime contexts [8].

The Contra Rotating Propeller (CRP) type is a propulsor configuration that offers superior efficiency compared to conventional single propellers [7-9]. The main benefit is its capability to reclaim the rotational energy in the propeller's slipstream [8,9]. This concept allows the ship to move more efficiently, reducing the energy loss that typically occurs in the propeller slipstream [8].

This propeller type has been applied to modern ships that use electric propulsion systems [8]. As such, the ship can move faster and smoother ande fuel [8,10]. In an environmental context that increasingly values energy efficiency and emission reduction, using Contra Rotating Propellers becomes increasingly relevant and important [9,10].

In other words, the Contra Rotating Propeller is an innovative solution that allows ships to run more efficiently and environmentally friendly [8,11]. It is a significant step forward in maritime technology, paving the way towards a more sustainable and efficient future [10,11].

This research aims to evaluate the efficacy of contra-rotating propellers on ferry vessels, specifically in relation to enhancing propulsion efficiency. This involves studying the unique dynamics of contra-rotating propellers and how they interact with the fluid flow at different speeds, aiming to optimize maritime transportation by improving fuel efficiency and performance.

2. Methodology

The focal point of this research is the KMP Bontoharu ferry, a meticulously designed ship model with dimensions detailed explicitly in Table 1. The contra-rotating propeller, a pivotal part of this model, is presented in two distinct design variations, each with unique characteristics and implications.

Table 1
Ship particulars

Dimension	Value
Lengths of All	54.000 m
Lengths Between Perpendiculars	47.450 m
Breadth	14.000 m
Height	3.400 m
Draft	2.450 m
Speed	6.618 m/s
Displacement	1148 tons

The first design variation keeps an equal ratio of 1:1 (as visually depicted in Figure 1) between the front and rear propellers. This design ensures a balanced distribution of force, thereby improving the propulsion dynamics of the ship model.

The second design variation, however, deviates slightly from the first. It features a unique ratio of 1:0.87 (illustrated in Figure 2), separating 0.1 meters between each propeller. This particular design choice allows for a nuanced adjustment of the propulsion dynamics, potentially leading to improved efficiency and performance [11-13].

These variations in design provide a comprehensive understanding of the potential advantages and implications of different contra-rotating propeller configurations and contribute significantly to the field of maritime engineering. By exploring these design variations, this research paves the way for future innovations in ship propulsion systems, thereby enhancing the efficiency and sustainability of maritime transportation.

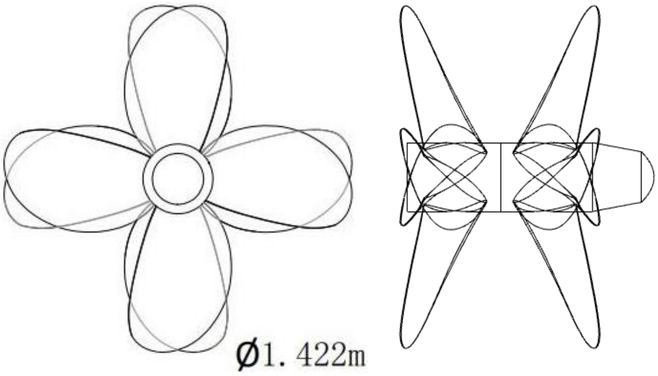

Fig. 1. CRP 1 design with a ratio of 1:1

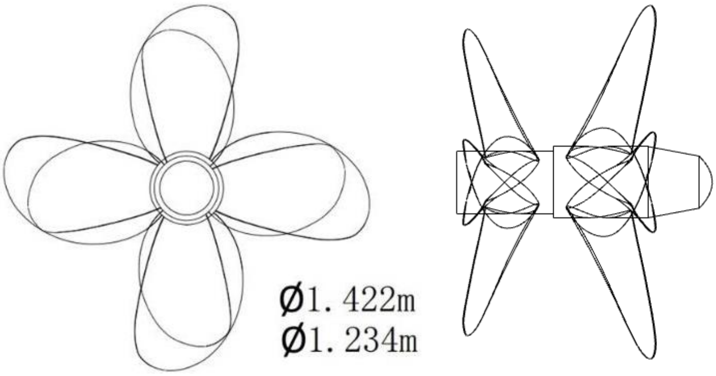

Fig. 2. CRP 2 design with a ratio of 1:0.87

We delve into computational fluid dynamics (CFD) to simulate and analyze the behavior of these propeller configurations under various conditions. The CFD modeling process is visually represented in Figure 3.

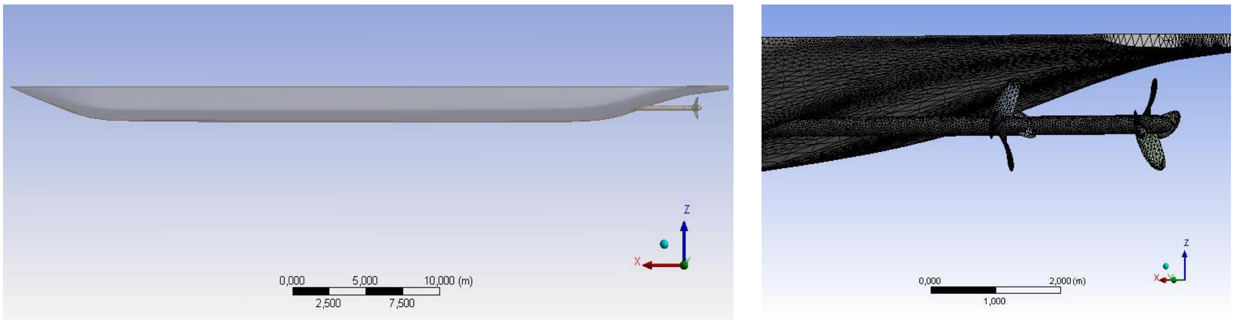

Fig. 3. Modeling and mesh in CFD

Meshing is an integral and crucial aspect of Computational Fluid Dynamics (CFD) modeling [14-17]. In the context of our research, we use three distinct classifications of mesh resolution: coarse, medium, and fine. These classifications are not arbitrary; they each represent varying degrees of detail captured in the simulation. The coarse mesh provides a broader overview, while the medium mesh offers a balance between detail and computational efficiency. The fine mesh, on the other hand, offers the most detailed view, capturing the minutest features of the simulation, thereby offering the highest resolution [15,18]. This method enables us to understand how the resolution of the mesh influences the precision and dependability of our Computational Fluid Dynamics simulations. It is important to note that the choice of mesh resolution can significantly influence the results, and hence, it is a critical factor to consider in CFD modeling [16].

We embarked on a comprehensive comparative analysis in our quest to validate the Computational Fluid Dynamics (CFD) simulations that we conducted. This analysis juxtaposed the results derived from our simulations with those obtained from empirical formulas, providing a robust framework for validation.

Intriguingly, our findings unveiled a fascinating aspect of CFD simulations. Despite its increased computational demand, we discovered that the use of a fine-resolution mesh yielded markedly more exact results. This finding underscores the importance of resolution in CFD simulations, highlighting its significant role in enhancing the results' precision.

While the use of high-resolution meshes does indeed place a greater demand on computational resources, the benefits they offer in terms of precision are considerable. Their enhanced accuracy far outweighs the increased computational demand, making them a valuable tool in CFD simulations. This insight has profound implications for future research, potentially paving the way for more exact and reliable simulations in the field of fluid dynamics.

The non-dimensional parameters used in analyzing the results about torque and thrust are outlined as follows, as referenced in previous studies [19,20]

$$J = \frac{V_A}{nD} \tag{1}$$

$$K_T = \frac{T}{\rho n^2 D^4} \tag{2}$$

$$K_Q = \frac{T}{\rho n^2 D^5} \tag{3}$$

These insights are valuable for our current research and contribute to the broader field of maritime engineering. By understanding the implications of different contra-rotating propeller configurations and the impact of mesh resolution on CFD accuracy, we can continue to innovate and improve in the field [11,21].

3. Results and Discussion

3.1 Resistance

Ensuring the precision of simulation results is of utmost importance. A practical approach to this involves calculating ship resistance, a reliable measure of the simulation's accuracy. This technique employs the Holtrop method, a well-established and widely recognized method in naval architecture for predicting ship resistance [22].

The Holtrop method offers a comprehensive analysis, making it a preferred choice for such calculations [22]. The comparison of ship resistance derived from CFD simulations and empirical calculations using the Holtrop method is visually represented in Figure 4. This comparative curve offers a clear view of the data, allowing for an easy understanding of the correlation and deviation between the two sets of results.

Upon thoroughly analyzing the data, it becomes clear that the results obtained from the Computational Fluid Dynamics (CFD) simulations and those derived from the Holtrop method show a close alignment. The mean discrepancy between these two sets of outcomes is roughly 7%.

At first glance, this discrepancy might appear to be relatively insignificant. However, even such a seemingly minor discrepancy in CFD simulations assumes a critical role. Precision is paramount in CFD simulations; even a slight deviation can have significant implications.

Therefore, while the average difference of 7% might seem trivial in a different context, it is a major factor that must be considered in CFD simulations. This underlines the importance of meticulousness and precision in CFD simulations and serves as a reminder of the need for rigorous validation and verification processes in computational modeling.

Using the Holtrop method in conjunction with CFD simulations provides a robust approach to evaluating ship resistance, ensuring the accuracy and validity of the simulation results [23,24].

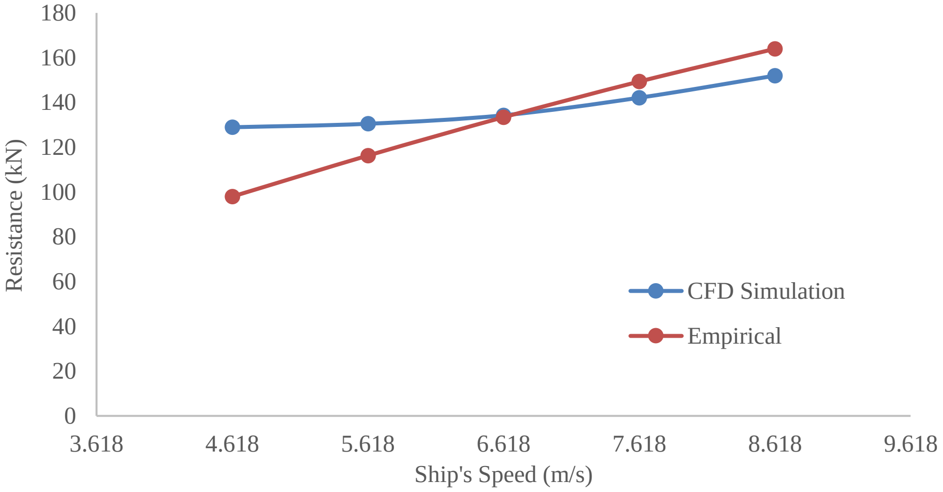

Fig. 4. Resistance calculation curve resulting from an approach using the Holtrop method and CFD simulation

The difference in resistance values obtained using the Holtrop approach and CFD simulation is caused by differences in the calculation methods. In CFD simulations, resistance calculations are based on the value of flow obstruction on the object, while the Holtrop approach calculates resistance not only on the ship's hull below the waterline but also focuses on additional components of the ship, wind resistance and others.

3.2 Thrust and Turque

In the field of marine engineering, the performance of a propeller is a critical aspect. The thrust generated by the propeller is one of the key parameters that decide the performance of a ship. In this context, we have two types of propellers under consideration: a conventional propeller and a contra-rotating propeller.

The conventional propeller has a traditional design and has been used for many years in various types of vessels. The empirical calculations obtained with the ship and propeller dimensions suggest that a conventional propeller can generate a thrust of 144.525 kN.

The simulation results for a conventional open-water propeller show a slightly higher thrust value of 145.251 kN. This suggests that the actual performance of the propeller might be slightly better than the empirical calculations suggest.

Moving on to the contra-rotating propeller models, these are a more recent innovation. In a contra-rotating setup, two propellers are placed in series but rotate in opposite directions. This design is believed to be more efficient, as it can use more energy in the water flow.

The CFD simulations were conducted for three different variations of the contra-rotating propeller. As shown in Table 2, the results show that the model CRP 2, which has a propeller ratio of 1:0.87, generates the highest thrust among all the models evaluated. This suggests that the contra-rotating design, specifically the CRP 2 model, could offer superior performance compared to a conventional propeller. While the conventional propeller offers reliable performance, the contra-rotating propeller, particularly the CRP 2 model, shows promise for even higher thrust values. These findings could have significant implications for the design and performance of future marine vessels.

Table 2
Propeller thrust by CFD simulation in velocity 6.618 m/s

Model	Thrust (kN)	%gain
Conventional	144.525	0.00
CRP 1	149.402	3.374
CRP 2	153.916	6.497

In the Computational Fluid Dynamics (CFD) simulation results, we can see an intriguing correlation between the speed and the propeller thrust, as depicted in Figure 5 (for CRP 1) and Figure 6 (for CRP 2). Both figures present a somewhat analogous pattern across the two propeller variations simulated. This suggests that the behavior of the propellers under different speeds is consistent, regardless of their specific design or configuration.

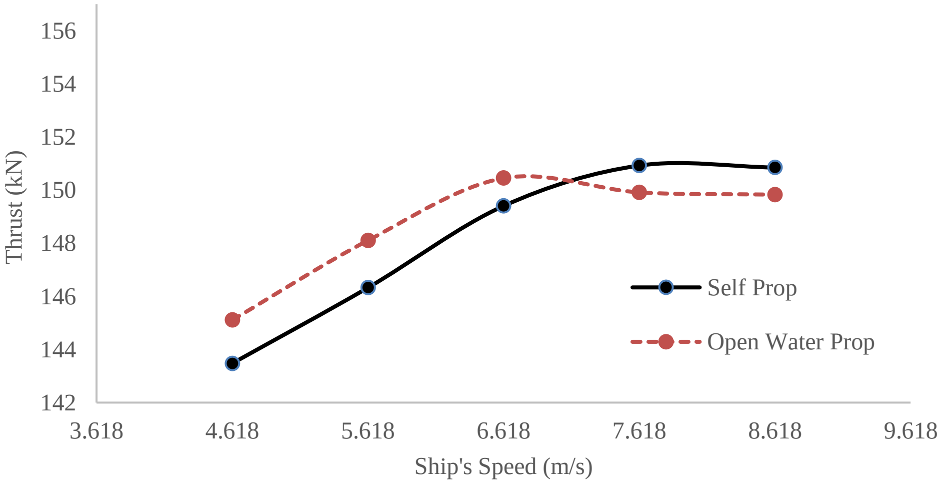

Fig. 5. Thrust values of several ship speeds using model CRP 1

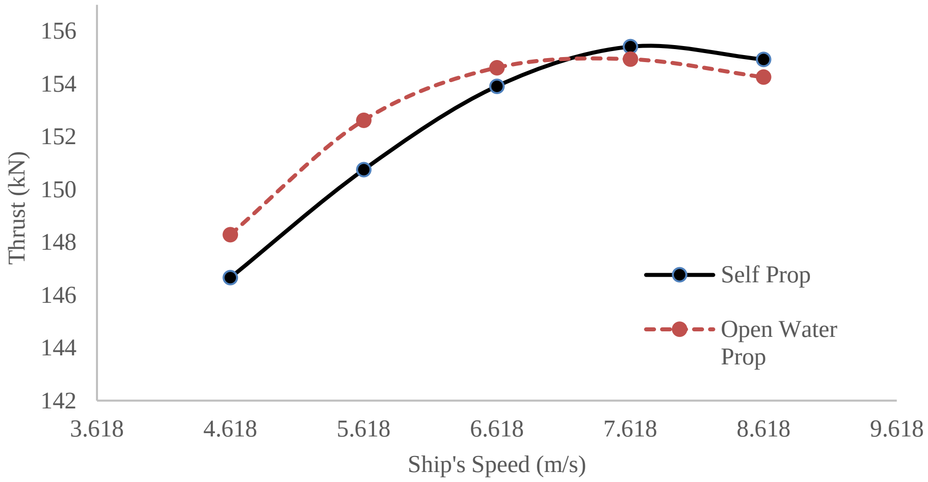

Fig. 6. Thrust values of several ship speeds using model CRP 2

Interestingly, the apex of the thrust value was recorded at a speed of 7.618 m/s. This shows that the propellers could generate the maximum thrust at this particular speed. However, there was a noticeable decline in the thrust value beyond this speed.

The observed decline in performance could be attributed to many factors. These factors could range from an increase in air resistance, which is a common phenomenon at higher speeds, to a shift in the efficiency of the propellers, which various operational and environmental conditions could influence.

Despite the decline, these findings offer valuable insights into the intricate dynamics of propeller performance. They shed light on how speed, a critical parameter in maritime operations, influences the performance of the propellers in this simulation. The simulation results unravel a complex interplay between speed and propeller thrust, two key factors that significantly affect the propulsion efficiency of a vessel.

This relationship between speed and propeller thrust is not merely a linear one but rather a complex interaction that can have profound implications for the design and operation of propellers. A deeper understanding of this relationship, as eased by these simulation results, can serve as a valuable resource in propellers' future design and optimization.

From the same Computational Fluid Dynamics (CFD) simulation, we obtained the torque values for CRP 1 and CRP 2, as depicted in Figure 7 and Figure 8, respectively. In Figure 5 through Figure 8, tests were conducted under two distinct conditions: open water conditions (represented by the red line) and behind the hull propeller (represented by the black line). It is important to note that there is a significant disparity in the torque results derived from these two testing conditions.

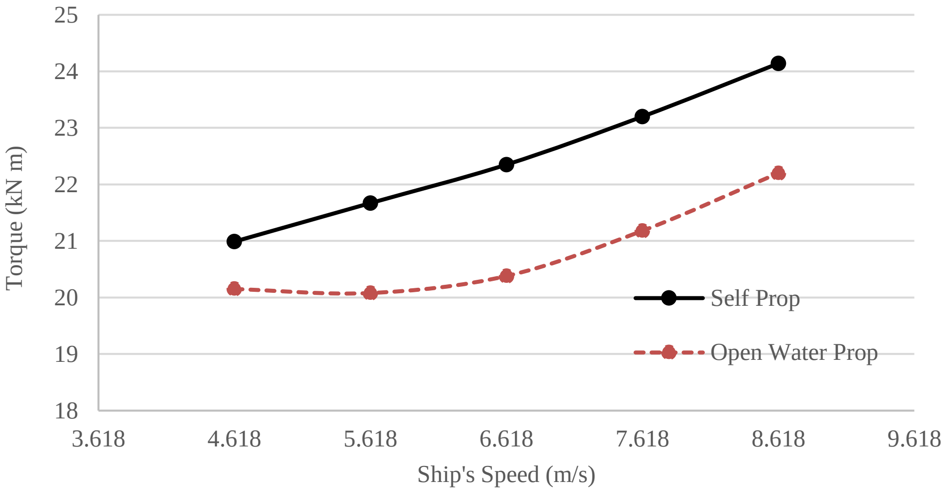

Fig. 7. Torque obtained from CFD simulation in model CRP 1

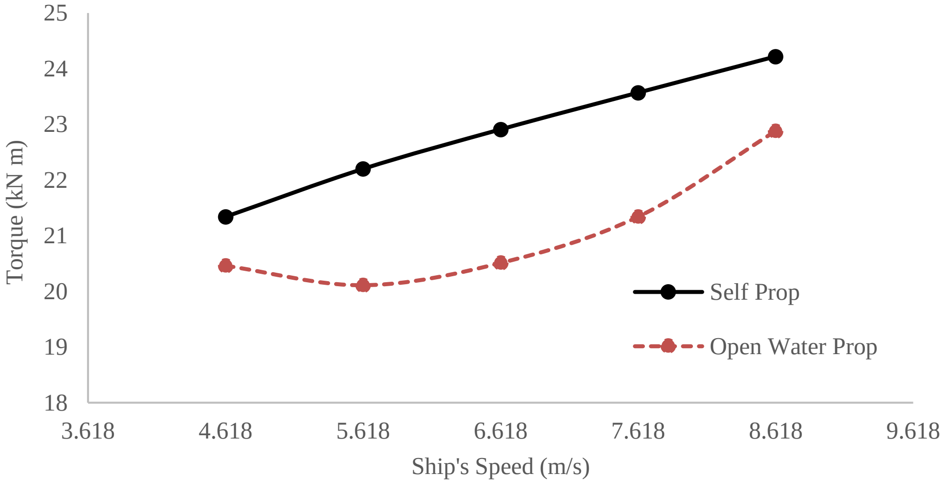

Fig. 8. Torque obtained from CFD simulation in model CRP 2

This disparity can be ascribed to the diverse flow circumstances that the propeller encounters in each situation. In the open water test, the flow conditions approaching the propeller are comparatively more steady and unchanging. This is because no external structures influence the water flow, allowing the propeller to run in a consistent environment. On the other hand, when the test is conducted with the hull, the flow pattern entering the propeller is greatly influenced by the shape of the hull and its speed. The hull can disrupt the water flow, creating a more turbulent and variable environment for the propeller. This can significantly affect the propeller's performance, as shown in Figure 9. The simulation results emphasize the importance of considering the operational environment when evaluating the performance of a propeller, as it can significantly affect critical factors such as torque.

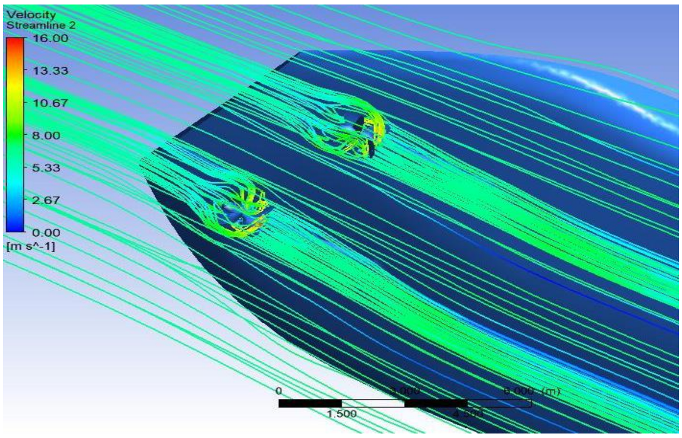

Fig. 9. Flow pattern in CRP

Figure 9 is the simulation results of fluid flow interacting with contra-rotating propellers. The simulation visualizes the velocity streamline patterns around the propellers, which are shown in two distinct swirling motions indicative of each propeller's rotation direction.

The two distinct swirling motions comprehensively visualize how the contra-rotating propellers influence the fluid flow. The front propeller accelerates the fluid and then interacts with the rear propeller, moving in the opposite direction. As the simulation depicts, this interaction between the fluid flow and the propellers can lead to complex flow patterns and velocity distributions.

4. Conclusion

Appearing from the research is a clear and compelling argument for the use of a contra-rotating propeller in marine propulsion systems. The study provides robust evidence that this technology can significantly enhance performance metrics. More specifically, the research saw a substantial increase in both thrust and torque, with the former seeing an amplification of 6.498% and the latter experiencing a rise of 4.899%. These enhancements are not just hypothetical; they exert a direct and quantifiable influence on the total efficiency of the propulsion system.

The results were even more striking when this contra-rotating propeller technology was integrated into ferry systems. The first variation of the system saw an efficiency boost of 1.265%, while the second variation recorded a more substantial increase of 3.797%.

But the research did not stop there. It also explored the impact of using propellers of different diameters on a single axis. The findings from this part of the study could have far-reaching implications for the design of future propulsion systems. The evidence suggests that using propellers of differing diameters can lead to superior efficiency outcomes compared to using propellers of identical diameters. This insight challenges conventional design principles and opens up new possibilities for innovation in marine propulsion technology.

In conclusion, the research provides a strong case for the adoption of contra-rotating propellers in marine propulsion systems. The increases in thrust, torque, and overall efficiency could revolutionize the way we design and run these systems. Furthermore, the potential for even more significant efficiency gains through the use of propellers of different diameters offers an exciting avenue for future research and development in this field.

Acknowledgments

The research team would like to thank the funding aid from the LBE Faculty of Engineering, Hasanuddin University.

References

[1] Kim, M. C., H. H. Chun, and Y. D. Kang. "Design and experimental study on a new concept of preswirl stator as an efficient energy-saving device for slow speed full body ship." *Transactions-Society of Naval Architects and Marine Engineers* 112 (2004): 111-121.

[2] Khan, Md Riad, Raju Kundu, Waliur Rahman, Md Mobinul Haque, and Md Refayeth Ullah. "Efficiency study: Contra-rotating propeller system." In *AIP Conference Proceedings*, vol. 1980, no. 1. AIP Publishing, 2018. https://doi.org/10.1063/1.5044369

[3] Holtrop, Jan. "A statistical re-analysis of resistance and propulsion data." *International Shipbuilding Progress* 31, no. 363 (1984): 272-276.

[4] Ali, Munazid, Ariana I Made, and I Ketut Aria Pria Utama. "CFD Analysis on the Development of Pre-Duct Shape to Improve Propeller Performance." *CFD Letters* 16, no. 2 (2024): 118-132. https://doi.org/10.37934/cfdl.16.2.118132

[5] Adietya, Berlian Arswendo, I Ketut Aria Pria Utama, Wasis Dwi Aryawan, and Sutiyo Sutiyo. "CFD analysis into the effect of using propeller boss cap fins (PBCF) on open and ducted propellers, case study with propeller B-series and kaplan-series." *CFD Letters* 14, no. 4 (2022): 32-42. https://doi.org/10.37934/cfdl.14.4.3242

[6] Yu, Jin-Won, Seong-geun Kim, Wooyoung Jeong, Yang Ik Kim, Jung-Eun Choi, and Inwon Lee. "Prediction of resistance and propulsion performances using calm-water and resistance tests in regular head waves of a 1,800 TEU container ship." *International Journal of Naval Architecture and Ocean Engineering* 14 (2022): 100458. https://doi.org/10.1016/j.ijnaoe.2022.100458

[7] Wenhui, Yan, and Zhang Kun. "Effects of Stage Spacing on Contra-Rotating Propeller Aerodynamic Interactions." In *Journal of Physics: Conference Series*, vol. 2478, no. 12, p. 122013. IOP Publishing, 2023. https://doi.org/10.1088/1742-6596/2478/12/122013

[8] Filippone, A. "Historical development of the coaxial contra-rotating propeller." *The Aeronautical Journal* 127, no. 1311 (2023): 699-736. https://doi.org/10.1017/aer.2022.92

[9] Gur, Ohad. "Extending blade-element model to contra-rotating configuration." In *IOP Conference Series: Materials Science and Engineering*, vol. 638, no. 1, p. 012001. IOP Publishing, 2019. https://doi.org/10.1088/1757-899X/638/1/012001

[10] Tran, Tien-Anh, and Thi-Uyen-Uyen Nguyen. "Performance Investigation of the Propulsion System Using Contra Rotating Propeller." In *International Conference on Material, Machines and Methods for Sustainable Development*, pp. 449-456. Cham: Springer Nature Switzerland, 2022. https://doi.org/10.1007/978-3-031-39090-6_50

[11] Zhu, Wencai, Feng Zhou, Huijie Shu, and Guoliang Hu. "Effect of contra-rotating propeller boss cap fins (CRPBCF) on the performance of marine propellers." *Ocean Engineering* 266 (2022): 112932. https://doi.org/10.1016/j.oceaneng.2022.112932

[12] Zhang, Yu-xin, Xiao-ping Wu, Zhi-yong Zhou, Xuan-kai Cheng, and Yu-long Li. "A numerical study on the interaction between forward and aft propellers of hybrid CRP pod propulsion systems." *Ocean Engineering* 186 (2019): 106084. https://doi.org/10.1016/j.oceaneng.2019.05.066

[13] Tadros, Mina, Manuel Ventura, and C. Guedes Soares. "Towards fuel consumption reduction based on the optimum contra-rotating propeller." *Journal of Marine Science and Engineering* 10, no. 11 (2022): 1657. https://doi.org/10.3390/jmse10111657

[14] Seifollahi Moghadam, Zahra, François Guibault, and André Garon. "On the evaluation of mesh resolution for large-eddy simulation of internal flows using OpenFOAM." *Fluids* 6, no. 1 (2021): 24. https://doi.org/10.3390/fluids6010024

[15] Vázquez, Carla, María del Mar Micó, and Jesús Colprim. "Computational Fluid Dynamics (CFD) Modelling of a Full-Scale Oxidation Ditch: Suitable Mesh Selection Method and Hydrodynamic Perfomance Analysis." In *International Conference on Wider-Uptake of Water Resource Recovery from Wastewater Treatment*, pp. 394-399. Cham: Springer Nature Switzerland, 2024. https://doi.org/10.1007/978-3-031-63353-9_67

[16] Lintermann, Andreas. "Computational meshing for CFD simulations." *Clinical and Biomedical Engineering in the Human Nose: A Computational Fluid Dynamics Approach* (2021): 85-115. https://doi.org/10.1007/978-981-15-6716-2_6

[17] Khairuddin, Khairuddin, Andi Haris Muhammad, Muhammad Iqbal Nikmatullah, and Faisal Mahmuddin. "Water Depth Effect to Ferry Ship Resistance with Computational Fluid Dynamic Method." *CFD Letters* 14, no. 5 (2022): 98-105. https://doi.org/10.37934/cfdl.14.5.98105

[18] Doustdar, M. Mehdi, and Hamid Kazemi. "Effects of fixed and dynamic mesh methods on simulation of stepped planing craft." *Journal of Ocean Engineering and Science* 4, no. 1 (2019): 33-48. https://doi.org/10.1016/j.joes.2018.12.005

[19] Johannessen, Jahn Terje, Lucia Sileo, Kourosh Koushan, Vladimir Krasilnikov, Simon Froitzheim, and Reinhard Schulze. "Numerical and experimental study of a series of contrarotating propellers." In *Eighth International Symposium on Marine Propulsors, SMP'24*, pp. 625-637, Berlin, Germany. 2024.

[20] Propulsion Committee. *ITTC - Recommended Procedures and Guidelines. Hybrid Contra-Rotating Shaft Pod Propulsors Model Test. 7.5-02-03-01.6*. International Towing Tank Conference, 2017.

[21] Adjailia, Fouzia, and Michal Takáč. "Computational Methods in Computational Fluid Dynamics." In *International Conference on Sustainable and Innovative Solutions for Current Challenges in Engineering & Technology*, pp. 83-91. Singapore: Springer Nature Singapore, 2022. https://doi.org/10.1007/978-981-99-1431-9_7

[22] Cannon, Sophie, Stephen Boyd, and Julian Whitewright. "Development of a quantitative method for the assessment of historic ship performance." In *Practical Design of Ships and Other Floating Structures: Proceedings of the 14th International Symposium, PRADS 2019*, September 22-26, 2019, Yokohama, Japan-Volume I 14, pp. 269-289. Springer Singapore, 2021. https://doi.org/10.1007/978-981-15-4624-2_16

[23] Kamil, Md Salim, Mohamad Amir Azfar Roslan, and Muhammad Fauzan Misran. "Comparative Study of Ship Wave Resistance by Various Methods of Solution." In *Advanced Maritime Technologies and Applications: Papers from the ICMAT 2021*, pp. 121-127. Springer International Publishing, 2022. https://doi.org/10.1007/978-3-030-89992-9_11

[24] Rudan, Smiljko, Šimun Sviličić, Ivan Munić, Antonio Luca Cantilena, Irena Radić Rossi, and Alice Lucchini. "Comparison of Different Methods for Ancient Ship Calm Water Resistance Estimation." *Journal of Marine Science and Engineering* 12, no. 4 (2024): 658. https://doi.org/10.3390/jmse12040658