Optimization of Propeller Tunnel Design for the Polbeng II Ship

Jupri[,a)], Siswandi B[1,a)], Afriantoni[1,b)], Efrizal[1,c)], Fadzi hanif[2)], Muhammad Arika Hadri[3)]

[a,] Dept. of Naval Architecture, Bengkalis State Polytechnic, Indonesia 28712
[b,] Dept. of Naval Architecture, Bengkalis State Polytechnic, Indonesia 28712
[c,] Dept. of Naval Architecture, Bengkalis State Polytechnic, Indonesia 28712

[a)]Corresponding author: jupri@polbeng.ac.id, siswandi@polbeng.ac.id, afriantoni@polbeng.ac.id, efrizal@polbeng.ac.id
[b)] fadzihanif4@gmail.com
[c)] muhammadarikahadri@gmail.com

Abstract. The Polbeng II ship is one of the crew vessels owned by Bengkalis State Polytechnic. This ship plays a vital role in supporting various research and exploration activities in the waters of Bengkalis. To enhance the ship's performance in terms of speed, the addition of a propeller tunnel component has significant potential to improve the vessel's speed and efficiency. The tunnel is designed based on the propeller diameter (D) used by the Polbeng II ship, and this study compares several design model variations of 20%D, 30%D, 40%D, and 50%D in relation to the distance from the rudder. The objective is to achieve the highest flow pressure value among the four design model variations using CFD evaluation based on an input fluid flow velocity of 1 m/s. The evaluation results indicate that the model with 40%D yields the highest pressure value of 103.73 pascals, along with a flow velocity of 2.018 m/s.

Keywords: Tunnel, Speed, Pressure

INTRODUCTION

Ships are a primary mode of transportation at sea, playing a crucial role in the global economy. The performance of a ship encompasses not only safety and reliability but also operational efficiency, including speed and fuel consumption. In efforts to enhance the efficiency and performance of vessels, the development of propulsion technology has become a key focus. One promising technology is the use of propeller tunnels. [1, 2].

A propeller tunnel is a type of ship propulsion system where the blades of the propeller are housed within a tunnel located beneath the hull of the vessel. This design can reduce hydrodynamic drag and enhance propulsion efficiency, potentially increasing the ship's speed. The use of propeller tunnels has been studied and implemented on various types of vessels, demonstrating promising results. [3, 4, 5].

The Polbeng II ship is one of the crew vessels owned by Bengkalis State Polytechnic (Polbeng). This ship plays a crucial role in supporting various research and exploration activities in the waters of Bengkalis. It is typically used for marine surveys and other research activities in both shallow and deep waters. In terms of propulsion, the Polbeng II uses a propeller with a diameter of 50 cm. Given the need to improve the speed and operational efficiency of the vessel, along with the potential of tunnel propeller technology, this study aims to determine the optimal model and size of the tunnel. The goal is to use this tunnel to generate greater thrust, thereby increasing the ship's speed. The thrust value is influenced by the fluid flow patterns passing through the propeller and the rudder, meaning that the distance between the rudder and the propeller can affect the vessel's thrust capability. According to Sembiring et al. (2016), optimal thrust is achieved at a distance of 55.1% D. If the distance between the propeller and the rudder exceeds 55.1% D, the resulting thrust decreases. Meanwhile, Nikmatullah et al. (2022) indicate that the optimal distance between the propeller and the rudder is 20% D. If this distance is less than 20% D, the thrust decreases; similarly, if the distance exceeds 20% D, thrust value also declines. Based on this data, the study designs tunnel models considering the distance from the tunnel's end to the rudder. The variations in distance considered are 20% D, 30% D, 40% D, and 50% D, where "D" represents the diameter of the propeller used on the ship. The dimensions of the tunnel are designed to match the shape of the ship's stern. The planned models and shapes of the tunnel are illustrated in Figure 1.

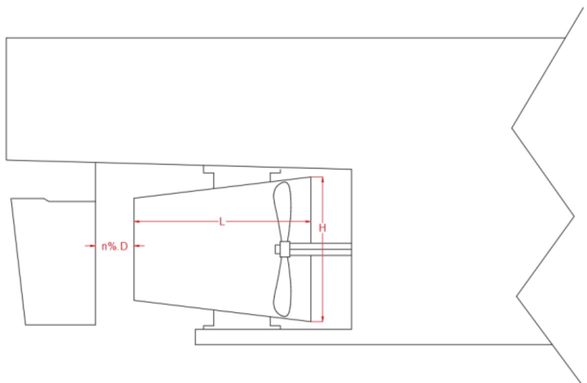


Figure 1. Tunnel Model

METHODS

The propeller tunnel is a type of propeller installed within a tunnel or channel created in the lower part of the ship, typically near the stern. This design aims to enhance thrust efficiency and reduce hydrodynamic drag around the propeller. The tunnel is designed to facilitate water flow to the propeller, improving thrust efficiency while also protecting the propeller from damage caused by collisions with objects in the water. In this study, the designed tunnel can be illustrated as a pipe through which water flows. When fluid flows through a pipe with varying diameters, differences in velocity and pressure occur. According to the law of conservation of energy, a change in one form of energy will be accompanied by a change in another form of energy. The relationship between changes in pressure and velocity is expressed in Bernoulli's equation as follows:

$$\frac{p_1}{\rho g} + \frac{v_1^2}{2g} + Z_1 = \frac{p_2}{\rho g} + \frac{v_2^2}{2g} + Z_2 \quad (1)$$

Theoretically, in a flow that experiences acceleration, the pressure will decrease. Conversely, in a flow that experiences low velocity (expansion), the pressure will increase. This principle is a fundamental aspect of fluid dynamics and is crucial for understanding how propeller tunnels function. By optimizing the design of the tunnel to create regions of accelerated flow, we can achieve lower pressure near the propeller, enhancing thrust and improving overall efficiency.

RESULTS AND DISCUSSION

Tunnel Design

The tunnel is designed in the shape of a pipe with different inlet and outlet diameters. The fluid is directed through the inlet with a specific pressure and flow rate and exits through the outlet, resulting in variations in pressure and flow rate between the inlet and outlet. To determine the pressure and flow rate values within the tunnel, evaluations are conducted using CFD (Computational Fluid Dynamics) software. Based on the specified dimensions and the contour of the Polbeng II ship's stern, a tunnel model is created, and the 3D model of the tunnel can be seen in Figure 2.

Tabel 1. Geometry tunnel

No	Unit	Tunnel 20%D (cm)	Tunnel 30%D (cm)	Tunnel 40%D (cm)	Tunnel 50%D (cm)
1	Inlet Diameter	56,5	56,5	56,5	56,5
2	Outlet Diameter	39,9	39,9	39,9	39,9
3	Tunnel Length	74,1	69,1	64,1	59,1
4	Distance from Tunnel to Rudder	10	15	20	25
5	Propeller Diameter (D)	50	50	50	50

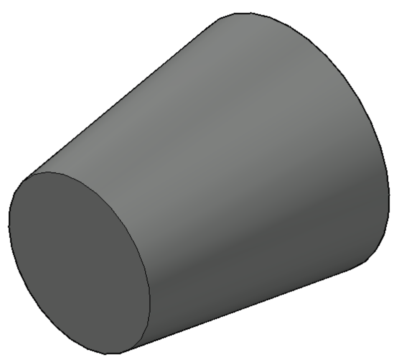


Figure 2. 3D Tunnel Model

EVALUATION OF THE TUNNEL

To evaluate the pressure and flow velocity of the fluid using CFD, the mesh setup on the model must be carefully configured, as it is a critical factor in determining the quality of the CFD simulation results. The mesh size used in this study is set to 0.05 m. The results of the meshing process can be seen in Figure 3.

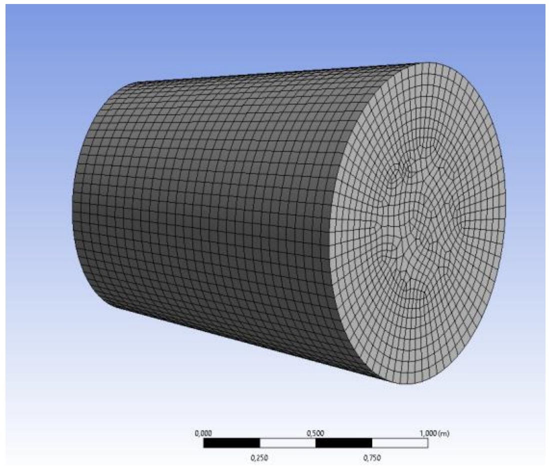


Figure 3. Mesh Model

The mesh configuration on the model illustrates the distribution of elements used in the CFD simulation to analyze the performance of the tunnel. The meshing process aims to represent the elements on the surface of the model to be evaluated; smaller elements generally yield more accurate results. After meshing, the next step is to determine the flow direction within the tunnel model that will be assessed.

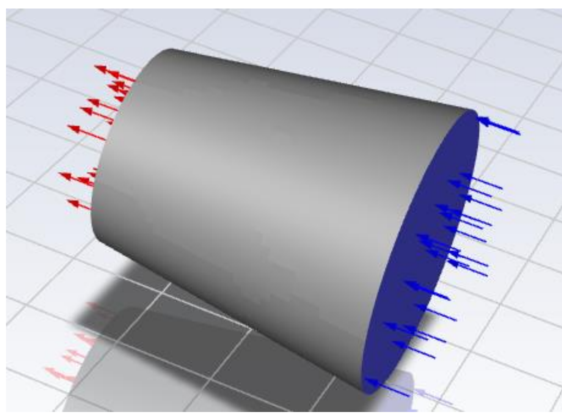
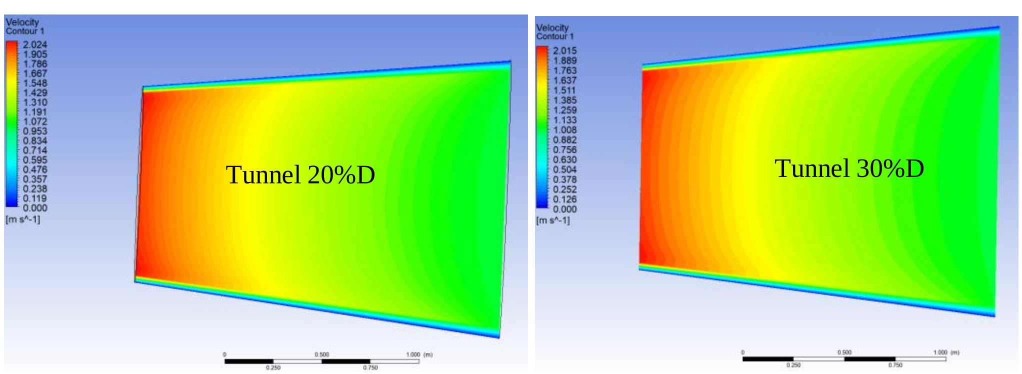


Figure 4. Flow Direction

The configuration of the fluid flow direction within the tunnel is conducted to determine how the flow is directed through the tunnel system. This step is essential for defining the inlet and outlet. In this evaluation, the fluid flow velocity is set at 1 m/s. Based on this flow velocity, the results of the CFD simulations regarding flow rate and pressure can be seen in Figures 5 and 6.



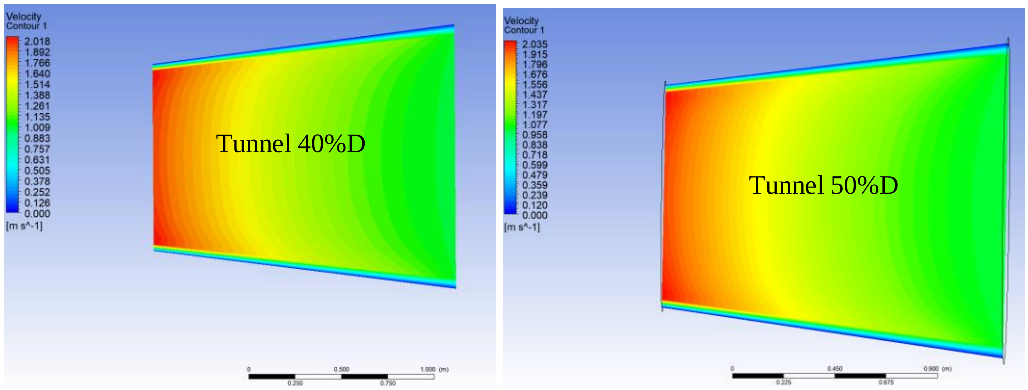


Figure 5. Flow Velocity of Tunnel

From Figure 5, the flow velocity within the tunnel is illustrated, showing how the fluid flow velocity varies across different tunnel designs. It can be observed that the fluid flow velocity at the outlet of the 20%D tunnel is 2.024 m/s, at the outlet of the 30%D tunnel is 2.015 m/s, at the outlet of the 40%D tunnel is 2.180 m/s, and at the outlet of the 50%D tunnel is 2.035 m/s.

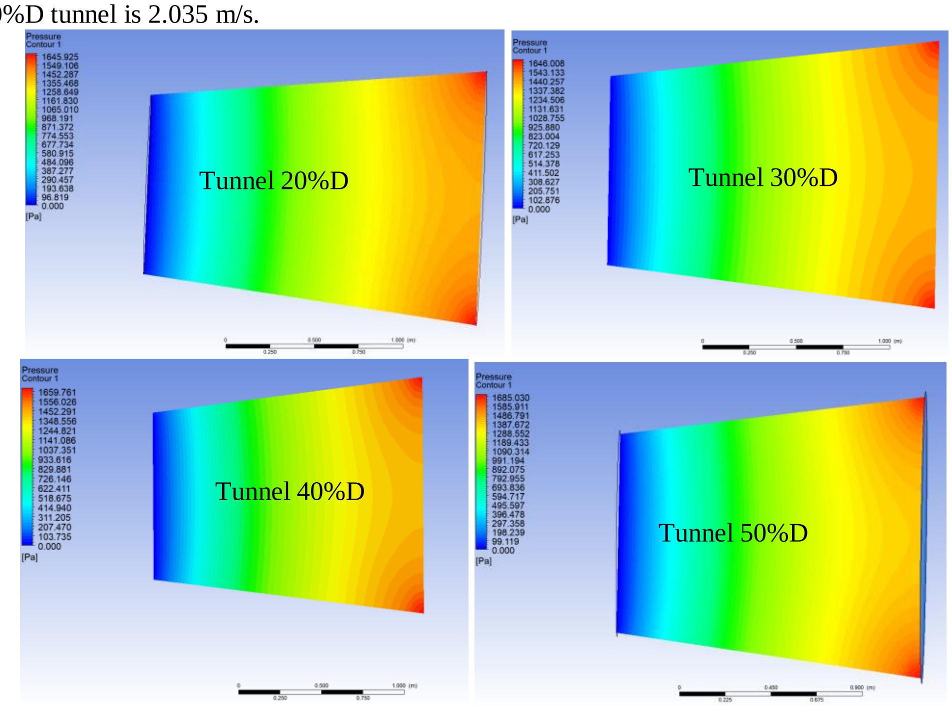


Figure 6. Flow Pressure in the Tunnel

Figure 6 illustrates the flow pressure within the tunnel, showing the pressure values across different tunnel design variations. From Figure 6, it can be observed that the flow pressure produced by each tunnel is as follows: at the outlet of the 20%D tunnel, the pressure is 96.819 pascal; at the outlet of the 30%D tunnel, it is 102.87 pascal; at the outlet of the 40%D tunnel, it is 103.73 pascal; and at the outlet of the 50%D tunnel, it is 99.119 pascal. Overall, a comparison of the flow velocities and pressures can be seen in Table 2.

Tabel 2. Hasil evaluasi aliran fluida

No	Model	Flow Velocity (m/s)	Pressure Velocity (pascal)
1	Tunnel 20%D	2,024	96,819
2	Tunnel 30%D	2,015	102,87

3	Tunnel 40%D	2,018	103,73
4	Tunnel 50%D	2,035	99,119

From Table 2, it can be identified that the 40%D tunnel model is the most optimal design for the Polbeng II ship, as it exhibits the highest flow pressure of 103.73 pascal.

CONCLUSIONS

The tunnel is designed with an inlet diameter of 56.5 cm and an outlet diameter of 39.9 cm, with model variations of 20% D, 30% D, 40% D, and 50% D. The evaluation results show that the 40% D tunnel model exhibits the best characteristics, achieving a flow velocity of 2.018 m/s and a generated flow pressure of 103.73 pascal.

ACKNOWLEDGMENTS

Friends from the Department of Ship Engineering have played a significant role in providing motivation and overcoming challenges during the shipbuilding process. We would like to express our heartfelt gratitude for the support you have provided. Additionally, P3M has played a crucial role in this project by providing vital financial support. Without the financial support from P3M, this project would not have been possible. We would also like to extend our gratitude to Bengkalis State Polytechnic for giving us the opportunity and trust to build the Polbeng II ship.

REFERENCES

[1] Carlton, J. S. Marine Propellers and Propulsion. Butterworth-Heinemann. 2007.

[2] Wu, Y., & Liu, Z. Numerical investigation on the influence of propeller configurations on a ducted propeller. Ocean Engineering, 107, 172-183.2015.

[3] Nguyen, T. T., & Kim, M. H. Effect of propeller-induced flow on the performance of tunnel thruster using numerical simulations. Ocean Engineering, 137, 94-108.2017.

[4] Sajjadi, S. M., Rostami, A., & Jahanbakhshi, A. (2017). Performance and cavitation characteristics of different bollard pull propellers: Experimental and numerical analysis. Ocean Engineering, 142, 190-200.2017.

[5] Schneekluth, H., & Bertram, V. Ship Design for Efficiency and Economy (Second Edition). Butterworth-Heinemann.2017.

[6] Sembiring, H. D. E., Chrismianto, Deddy., & Manik, Perlindungan. (2016). Pengaruh jarak rudder dan propeller terhadap kemampuan thrust menggunakan metode CFD (Studi kasus kapal kriso container ship). Jurnal Teknik Perkapalan, Vol. 4, No. 1, 229 - 236.

[7] Nikmatullah, M. I., Muhammad, A. H., Haryanto., & Zulkifli. (2022). Pengaruh jarak rudder dan propeller terhadap kemampuan thrust kapal perikanan 30 GT. Jurnal riset & Teknologi terapan kemaritiman, Vol. 1, No. 1, 22-25.

[8] Rahayu, Puji., Putri, D.K., Rosalina., Indriyani, Nita. (2021). Pengaruh diameter pipa pada aliran fluida terhadap nilai head loss. Jurnal Agitasi, Vol. 2, No. 1. hal.

[9] Sarjito., Subroto., Kurniawan, arif. (2016). Studi distribusi tekanan aliran melalui pengecilan saluran secara mendadak dengan belokan pada penapang segi empat. Jurnal ilmiah teknik mesin, Vol 17, No. 1. hal.

[10] Abidin, Kurniati., Wagiani, Sri. (2013). Studi analisis perbandingan kecepatan aliran air melalui pipa venturi dengan perbedaan diameter pipa. Jurnal Dinamika, Vol. 04, No. 1. hal 62-78.