International Journal of Marine Engineering Innovation and Research, Vol. 8(2), Jun. 2023. 251-260
(pISSN: 2541-5972, eISSN: 2548-1479)

Numerical Study of Kaplan Propeller by Using CFD: Effect of Angle and Blade Diameter Variations

Mohammad Danil Arifin[1], Frengki Mohamad Felayati[2], Muswar Muslim[3], Ayom Buwono[4], Yeddid Yonatan Eka Darma[5]

(Received: 28 May 2023 / Revised: 29 May 2023 /Accepted: 29 May 2023)

***Abstract*— Efficient propeller performance contributes to better overall ship performance and speed. A well-designed propeller can optimize thrust generation, leading to improved maneuverability, responsiveness, and acceleration. It enables ships to maintain higher speeds while using less power, enhancing their competitiveness in the maritime industry. In this study, the Kaplan series propeller was analyzed by using Computational Fluid Dynamics (CFD). By modifying the angle of attack on the Kaplan propeller with 3, 4, and 5 blades, the distribution of the surface pressure, generated thrust, and torque value were easily identified and analyzed. The result shows that the change in the angle of attack influenced the pressure distribution on the back and face side of the propeller. The angle of attack is increased, and the pressure surface distribution also tends to increase. It has also affected the efficiency of the propeller performance which is expressed by the values of thrust propeller and torque. The more efficient the propeller performance, the less power it requires to produce the desired thrust.**

Keywords— Computational Fluid Dynamics (CFD), angle of attack, efficiency

I. INTRODUCTION

The primary function of a ship's propeller is to generate thrust and propel the vessel through the water. It converts the rotational motion of the ship's engines or motors into forward or backward motion, allowing the ship to move efficiently and maneuver effectively. Without a propeller, a ship would be unable to navigate and fulfill its intended purpose of transportation or maritime operations.

There are various types of ship propellers i.e., Fixed-Pitch Propeller (FPP) and Controllable-Pitch Propeller (CPP) [1][2]. CPP is the most common type of propeller, consisting of fixed blades with a constant pitch angle. Fixed-pitch propellers are simple in design and suitable for ships with constant operating conditions and speed ranges. However, they cannot adjust their pitch, limiting their efficiency in varying load conditions.

Controllable-Pitch Propeller (CPP) allows for adjustable pitch angles, enabling the propeller blades to be rotated and set at different angles during operation [3]. This feature allows for optimal performance across a range of ship speeds and operating conditions.

CPPs are commonly used in vessels that require frequent speed and direction changes, such as tugs, offshore supply vessels, and some larger ships. The efficiency of a ship propeller is often used as an indicator of its quality [4][5].

The efficiency of a propeller refers to how effectively it converts the power input into thrust while minimizing energy losses. A propeller with high efficiency is considered to be of better quality as it can deliver more thrust with less power.

The efficiency of ship propeller performance is of paramount importance for several reasons [6]: A ship's propeller is responsible for generating thrust, which propels the vessel through the water. The more efficient the propeller, the less power it requires to produce the desired thrust.

Reduced power demand translates into lower fuel consumption, resulting in significant cost savings for ship operators. With the rising costs of fuel and increasing environmental concerns, improving propeller efficiency is crucial for minimizing fuel consumption and reducing greenhouse gas emissions.

Ship propulsion is a significant source of greenhouse gas emissions, particularly carbon dioxide (CO_2). By improving propeller efficiency, ships can operate with lower fuel consumption and consequently reduce their carbon footprint. Additionally, improved propeller efficiency can help reduce other harmful emissions such as sulfur oxide (SOx) and nitrogen oxide (NOx), which contribute to air pollution and acid rain.

Propeller efficiency directly affects the operational costs of a ship. More efficient propellers require less power to achieve the same speed, resulting in lower fuel expenses. Moreover, improved efficiency can extend the range of a vessel, allowing it to travel longer distances

Mohammad Danil Arifin, Department of Marine Engineering, Darma Persada University, Jakarta, 13450, Indonesia. E-mail: danilarifin.mohammad@gmail.com
Frengki Mohamad Felayati, Department of Marine Engineering, Hang Tuah University, Surabaya, 60111 Indonesia. E-mail: frengkimuhamad.felayati@gmail.com
Muswar Muslim, Department of Marine Engineering, Darma Persada University, Jakarta, 13450, Indonesia. E-mail: muslim.muswar@gmail.com
Ayom Buwono, Department of Marine Engineering, Darma Persada University, Jakarta, 13450, Indonesia. E-mail: abuwono.energi@gmail.com
Yeddid Yonatan Eka Darma, Department of Ship Manufacture Engineering, State Polytechnic of Banyuwangi, Banyuwangi, 68461, Indonesia. E-mail: yeddidyonatan@gmail.com

without refueling, thereby reducing downtime and increasing operational productivity.

Efficient propeller performance contributes to better overall ship performance and speed. A well-designed propeller can optimize thrust generation, leading to improved maneuverability, responsiveness, and acceleration. It enables ships to maintain higher speeds while using less power, enhancing their competitiveness in the maritime industry.

Inefficient propellers can cause excessive vibrations and noise, resulting in discomfort for passengers and crew members. By enhancing efficiency, propeller design can minimize vibrations and noise levels, leading to a smoother and quieter sailing experience. This is especially important for cruise ships, luxury yachts, and vessels carrying sensitive cargo.

Propeller efficiency is crucial for ensuring the safety and reliability of a ship. An inefficient propeller may struggle to deliver the required thrust, especially in adverse weather conditions or during maneuvers. By improving efficiency, ships can have better control, stability, and reliability, reducing the risk of accidents and enhancing overall maritime safety.

In conclusion, the efficiency of ship propeller performance directly impacts fuel consumption, environmental impact, operational costs, ship performance, comfort, safety, and reliability. Optimizing propeller efficiency is vital for achieving sustainable and cost-effective maritime operations while minimizing environmental harm.

Based on the several references, the efficiency of ship propeller performance can be influenced by several factors i.e., propeller design, advanced technology, etc. The design of the propeller itself plays a critical role in its efficiency [7]. Factors such as blade shape, size, pitch, and number of blades, angle of attack can significantly impact the propeller's ability to convert power into thrust. Optimizing these design parameters based on the specific requirements of the ship and its operating conditions can enhance propeller efficiency.

In case of advanced technology such as computational fluid dynamics (CFD) simulations, hydrodynamic optimization techniques, and the use of advanced materials, have a significant impact on propeller efficiency. These technologies enable more accurate propeller design, analysis, and performance prediction, leading to improved efficiency.

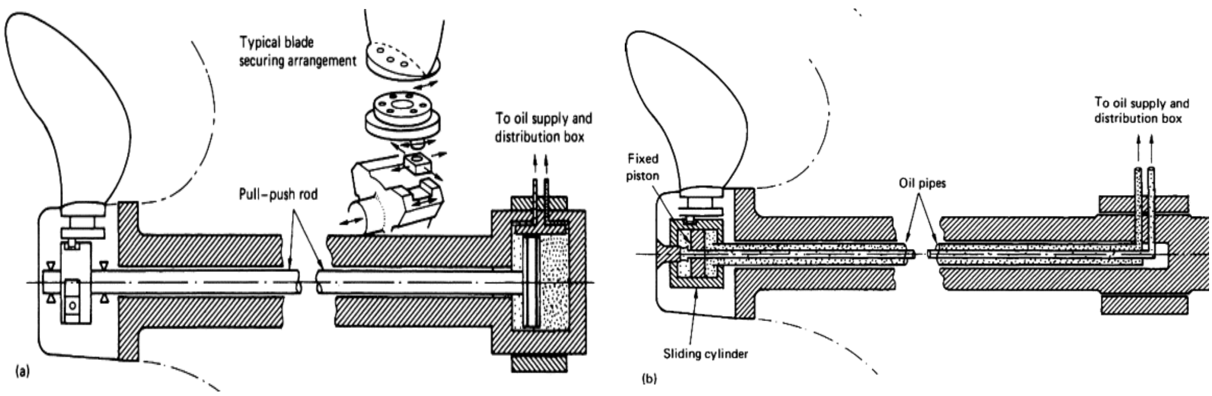


Figure. 1. Illustration of CPP

Computational Fluid Dynamics (CFD) is widely used for analyzing ship propeller performance [8]. CFD can be utilized to simulate and analyze the flow around the propeller blades, investigate hydrodynamic interactions with the ship's hull, predict cavitation phenomena, assess propeller efficiency, and optimize propeller design parameters. It enables engineers and designers to gain valuable insights into propeller performance, improve efficiency, and optimize the overall hydrodynamic performance of ships. It allows for detailed simulations of fluid flow around the propeller blades, providing valuable insights into efficiency and performance.

Smith et al., analyze the hydrodynamic performance of a ship propeller by employing CFD simulations. CFD software is used to simulate the flow around the propeller blades and calculate key performance parameters such as thrust, torque, and efficiency. The analysis reveals the propeller's efficiency at various operating conditions, enabling optimization of design parameters and providing insights for performance improvements.

The study focuses on investigating the impact of cavitation on ship propeller performance using CFD analysis was investigated by Danil et al., [2][7]. The CFD analysis provides visualization of cavitation patterns and quantifies its effects on propeller performance, aiding in the design of cavitation-resistant propellers.

Other research was conducted to optimize ship propeller design for fuel efficiency using CFD-based design optimization techniques [9]. CFD simulations coupled with optimization algorithms are employed to explore different propeller design parameters and identify optimal configurations that minimize fuel consumption. The analysis provides insights into the impact of design variations on propeller efficiency, leading to the identification of optimized propeller geometries that result in improved fuel efficiency.

In the case of the influence of hull-propeller interaction, Lee et al. investigate the interaction between the ship's hull and propeller and its influence on ship performance using CFD analysis [10][11]. CFD simulations are performed to model the combined flow around the ship's hull and propeller, considering their interaction effects. The analysis provides insights into the hydrodynamic interactions, including flow separation, wake effects, and propeller-hull alignment, which can impact propeller performance. This information helps optimize propeller-hull configurations for improved efficiency.

These research examples illustrate how CFD analysis can provide valuable insights into ship propeller performance, facilitating optimization, and design improvements.

In this study, the Kaplan series propeller is simulated by using CFD. By modifying the angle of attack on the Kaplan propeller with 3, 4, and 5 blades, the distribution of the surface pressure, generated thrust, and torque value was analyzed. As a result, the efficiency of the propeller performance was analyzed and discussed.

II. METHOD

A. Propeller Design

As mentioned in the previous section, in this study the Kaplan series with 3, 4, and 5 blades are used for the simulation. The efficiency of the propeller can be analyzed based on the value of thrust (N), and value of torque (Nm). The characteristics of the propeller can be drawn from a diagram function that consists of some parameters i.e., thrust coefficient (*KT*), torque coefficient (*KQ*), advanced speed coefficient (*J*), and propeller efficiency (η_0). The equation of those parameters is shown as follows:

$$KT = \frac{T_{prop}}{\rho \ x \ n^2 \ x \ D^4} \tag{1}$$

$$KQ = \frac{Q_{prop}}{\rho \ x \ n^2 \ x \ D^4} \tag{2}$$

$$\eta_0 = \frac{J \ x \ KT}{2\pi \ x \ KQ} \tag{3}$$

$$J = \frac{Va}{n \ x \ KQ} \tag{4}$$

For the simulations, the angle of attack of the propeller is varied. The propeller design model in this study is Kaplan with 3, 4, and 5 blades with variations of diameter is 0.3, 0.4, and 0.5 cm, and the variation of the angle of attack is 10.3º, 15.3º, and 20.0º. An example an illustration of the angle of attack variation shows in Figure 2.

The angle of attack was calculated by using equations (5) and (6). The value of the angle of attack for the propeller model is shown in Table 1, and the ordinate of the propeller design model for the simulation in this study shows in Tables 2-4.

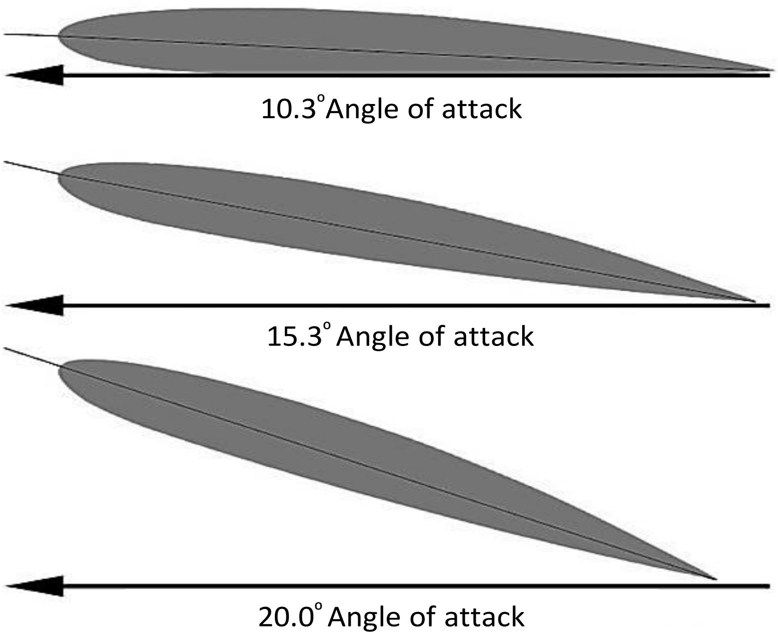


Figure. 2. Illustration of the angle of attack

$$\theta = tan^{-1}\frac{P/D_{0.7}}{2\pi R_{0.7}D} \quad (5)$$

$$\theta = tan^{-1}\frac{P/D_{0.7}}{0.7\pi} \quad (6)$$

The propeller model configuration for the simulation using CFD are consisting of 27 models to be simulated

TABLE 1.
THE ANGLE OF ATTACK FOR THE PROPELLER MODEL

D	RPM	$P/D_{0.7}$	$(P/D_{0.7})/(0.7R)$	θ(degree)
30	500	0.40	0.18189	10.30
		0.60	0.27284	15.30
		0.80	0.36378	20.00
40	500	0.40	0.18189	10.30
		0.60	0.27284	15.30
		0.80	0.36378	20.00
50	500	0.40	0.18189	10.30
		0.60	0.27284	15.30
		0.80	0.36378	20.00

TABLE 2.
ORDINATE OF KAPLAN 3 BLADES

r/R	Centerline to Trailing Edge			Centerline to Leading Edge		
	D30	D40	D50	D30	D40	D50
0.2	3.03	4.54	6.05	1.69	2.54	3.39
0.3	3.11	4.67	6.22	2.11	3.17	4.22
0.4	3.09	4.63	6.18	2.65	3.97	5.3
0.5	3.17	4.75	6.33	3.06	4.59	6.12
0.6	3.34	5.01	6.67	3.33	5.00	6.67
0.7	3.52	5.28	7.04	3.52	5.28	7.04
0.8	3.65	5.48	7.30	3.65	5.47	7.30
0.9	3.78	5.67	7.56	3.78	5.67	7.56
1	3.80	5.71	7.61	3.80	5.70	7.61

TABLE 3.
ORDINATE OF KAPLAN 4 BLADES

r/R	Centerline to Trailing Edge			Centerline to Leading Edge		
	D30	D40	D50	D30	D40	D50
0.2	2.21	3.32	4.42	1.22	1.82	2.43
0.3	2.30	3.46	4.61	1.54	2.32	3.09
0.4	2.29	3.44	4.59	1.96	2.94	3.92
0.5	2.36	3.53	4.71	2.27	3.41	4.55
0.6	2.48	3.73	4.97	2.48	3.73	4.97
0.7	2.62	3.94	5.25	2.62	3.94	5.25
0.8	2.73	4.09	5.45	2.72	4.09	5.45
0.9	2.82	4.24	5.65	2.82	4.24	5.65
1	2.84	4.26	5.69	2.84	4.26	5.69

TABLE 4.
ORDINATE OF KAPLAN 5 BLADES

r/R	Centerline to Trailing Edge			Centerline to Leading Edge		
	D30	D40	D50	D30	D40	D50
0.2	1.75	2.63	3.55	0.95	1.43	2.06
0.3	1.83	2.75	3.68	1.22	1.83	2.51
0.4	1.83	2.74	3.66	1.56	2.34	3.17
0.5	1.88	2.82	3.76	1.81	2.72	3.64
0.6	1.98	2.97	3.96	1.98	2.97	3.96
0.7	2.09	3.14	4.19	2.09	3.14	4.19
0.8	2.18	3.26	4.35	2.18	3.26	4.35
0.9	2.26	3.38	4.51	2.26	3.38	4.51
1	2.27	3.41	4.54	2.27	3.41	4.54

i.e., 9 models for Kaplan 3 blades, 9 models for Kaplan 3 blades, and 9 models for Kaplan 3 blades. The configuration of the propeller is illustrated in Figure 3.

B. Numerical Study

The CFD program is indeed a valuable tool for modeling and analyzing propeller configurations. CFD is a numerical simulation technique used to solve the governing equations of fluid flow and provide detailed information about the flow behavior around complex geometries [12].

When it comes to propeller analysis, CFD programs can help simulate the airflow and fluid forces acting on the propeller blades. By inputting the propeller's geometry and the operating conditions (such as velocity and rotational speed), the CFD software can calculate parameters like thrust, torque, and efficiency, and even visualize the flow patterns around the propeller.

The CFD program breaks down the fluid domain into a grid of small cells and applies mathematical equations to each cell to simulate the flow behavior [13]. These equations consider factors such as fluid viscosity, density, pressure, and velocity to compute the flow characteristics.

Using CFD simulations offers several advantages in the analysis of propeller configurations and fluid dynamics in general. Some of these advantages include:

- Cost and time savings: CFD simulations can be less expensive and time-consuming compared to physical experiments. Building and testing multiple propeller prototypes can be costly and time-intensive, while CFD allows for virtual testing and iteration at a fraction of the cost.
- Design optimization: CFD enables engineers to explore and optimize different propeller designs quickly. It can evaluate numerous design iterations, modifying parameters such as blade shape, twist distribution, and pitch angle, to identify the most efficient and high-performing configuration.
- Visualization of flow behavior: CFD provides detailed visualization of the flow patterns around the propeller blades. This allows engineers to understand the complex fluid dynamics, identify regions of separation, cavitation, and vortex shedding, and gain insights into the performance and potential issues of the propeller design.
- Parameter sensitivity analysis: CFD simulations allow for the investigation of the impact of various operating conditions and design parameters on propeller performance. By systematically varying parameters such as velocity, rotational speed, blade pitch, and number of blades, engineers can assess their influence on thrust, torque, efficiency, and other performance metrics.
- Prediction of forces and moments: CFD simulations can accurately predict the forces and moments acting on the propeller blades, such as thrust, torque, lift, and drag. This information is crucial for assessing propeller performance, structural integrity, and load distribution, aiding in the design process.
- Cavitation analysis: CFD can predict the occurrence and behavior of cavitation, which is the formation and collapse of air or vapor bubbles around the propeller blades. Cavitation can lead to performance degradation, increased noise, and potential damage. By analyzing cavitation patterns, engineers can optimize the propeller design to minimize its occurrence.
- Performance comparison: CFD allows for the direct comparison of different propeller configurations or designs, helping engineers make informed decisions. By quantitatively assessing performance metrics such as efficiency, thrust-to-power ratio, and wake distribution, they can select the most suitable propeller for specific applications.

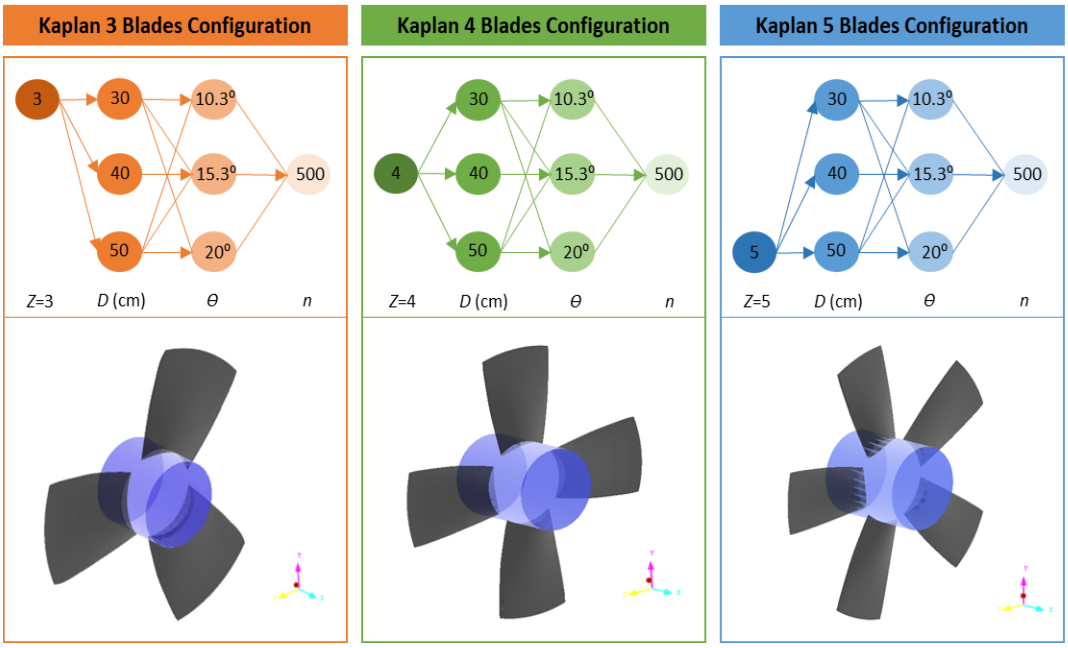
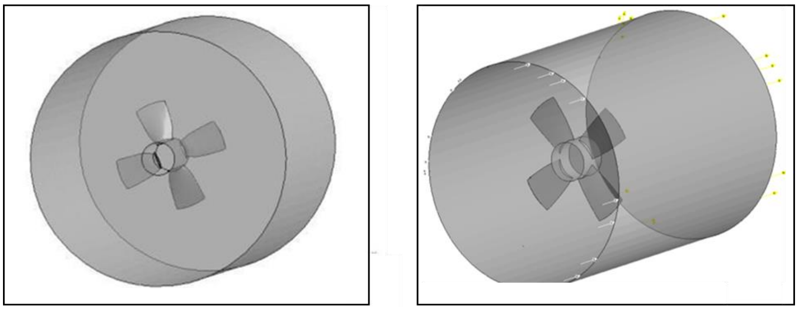


Figure. 3. Illustration of angle of attack

The CFD calculation process consists of three main parts as follows [14]:

- Pre-processing: In the pre-processing stage, the necessary input data, and specifications for the CFD simulation are prepared. This involves defining the geometry of the propeller and its surrounding fluid domain. The propeller geometry can be created using CAD (Computer-Aided Design) software or imported from existing designs. The fluid domain is also defined, including the boundaries and any additional components or structures present in the analysis. The object and boundary and the domain of the propeller model are illustrated in Figure 4(a) and 4(b).
- Mesh generation is a critical step in pre-processing, where the fluid domain is divided into small computational cells or elements to discretize the equations that govern fluid flow. Proper mesh quality is essential for accurate and efficient simulations. The illustration of the meshing steps shows in Figure 5.
- Solver/Computation: Once the pre-processing is complete, the simulation proceeds to the solver or computation phase. The solver numerically solves the governing equations of fluid flow, such as the Navier-Stokes equations, for each cell of the mesh. These equations account for factors like fluid viscosity, density, pressure, and velocity. The solver applies iterative methods to solve the equations, considering the boundary conditions defined in the pre-processing stage. As the solver progresses, it updates the flow variables (velocity, pressure, etc.) in each cell until a converged solution is obtained. This process involves multiple iterations and can be computationally intensive.
- Postprocessing: After the solver has converged and obtained the solution, the postprocessing stage begins. Here, the simulation results are analyzed and visualized to extract meaningful information. Postprocessing involves examining flow characteristics, performance metrics, and other variables of interest. The results can be presented in the form of contour plots, vector fields, graphs, or numerical data. Engineers can evaluate the propeller performance, assess forces and moments, identify flow patterns, analyze cavitation, and compare different design variations. Postprocessing helps in interpreting the simulation results and making informed decisions regarding the propeller configuration.
- It's important to note that the CFD calculation process may involve additional steps depending on the specific analysis requirements and complexity of the propeller configuration. Iterative refinement, validation against experimental data, and sensitivity analysis are some common practices performed throughout the process to ensure accuracy and reliability.



(a) **(b)**

Figure. 4 (a). Object and boundary, **(b)** Domain propeller

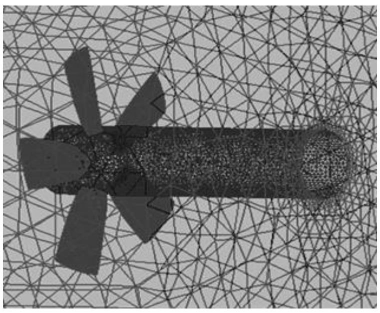


Figure. 5. Propeller mesh generation

International Journal of Marine Engineering Innovation and Research, Vol. 8(2), Jun. 2023. 251-260
(pISSN: 2541-5972, eISSN: 2548-1479)

III. RESULTS AND DISCUSSION

A. Pressure Surface Characteristics

The example of pressure surface distribution comparison between the face and back side of the propellers model for Kaplan 3, 4, and 5 blades with the angle of attack 20º are expressed in Figures 6-8. It was indicated that on the face side of the propeller, the pressure surface distribution is lower than the backside, and it shows a significant difference between those sides due to the influence of the different angles of attack significantly. The higher angle of attack applied; the more distribution of pressure will be produced.

B. Thrust Characteristics

The thrust result based on the simulation on the

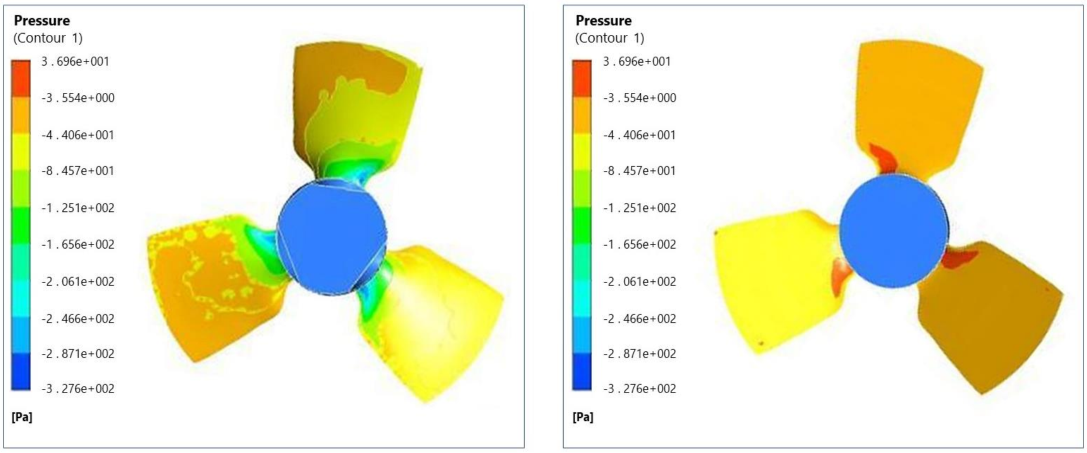


Figure. 6. Pressure distribution of Kaplan 3 blades with angle of attack 20º

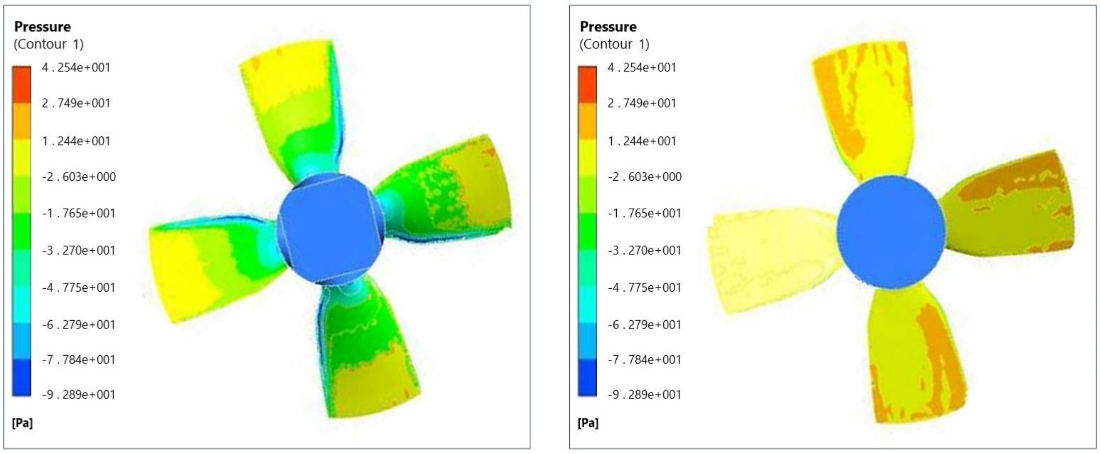


Figure. 7. Pressure distribution of Kaplan 4 blades with angle of attack 20º

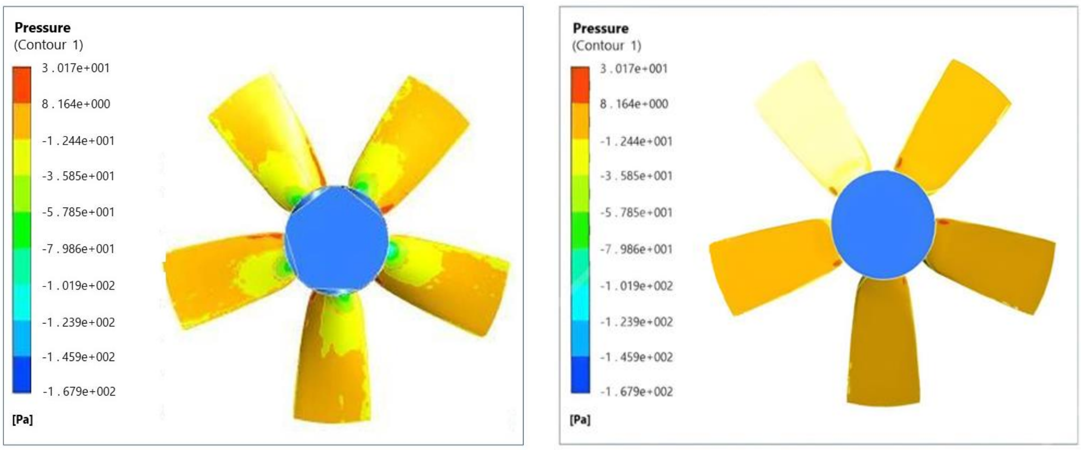


Figure.8. Pressure distribution of Kaplan 5 blades with angle of attack 20º

Kaplan 3, 4, and 5 blades with the angle of attack and diameter variations is shown in Figures 9-11 respectively.

The figure above, it showed that the thrust generated from the Kaplan 3, 4, and 5 blades with D=0.5m is generally shown the highest results for each angle of attack compared to the others model in the case of the same blade number. It shows that the highest thrust for the overall models was produced by Kaplan with 3 blades, with D=0.5m at an angle of attack = 20.0º. Hence, the thrust force is equal to the blade area multiplied by the difference in pressure. Therefore, the force of the Kaplan 3 blades models is relatively high, due to the wide blade area owned by these models.

C. Torque Characteristics

The simulation result of torque for the variation of

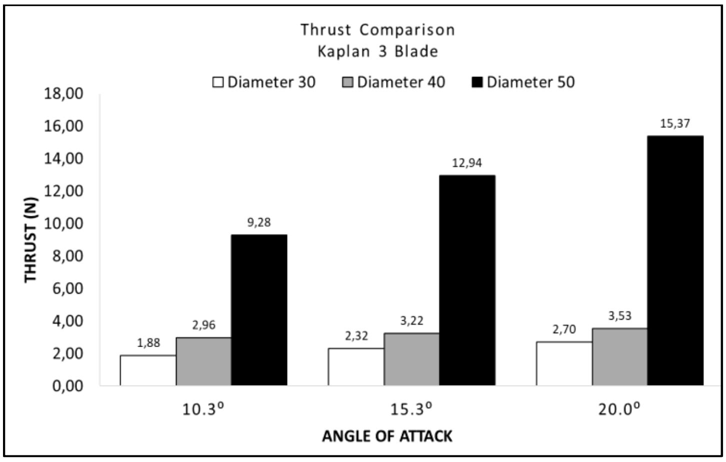


Figure. 9. Thrust comparison of Kaplan 3 blades

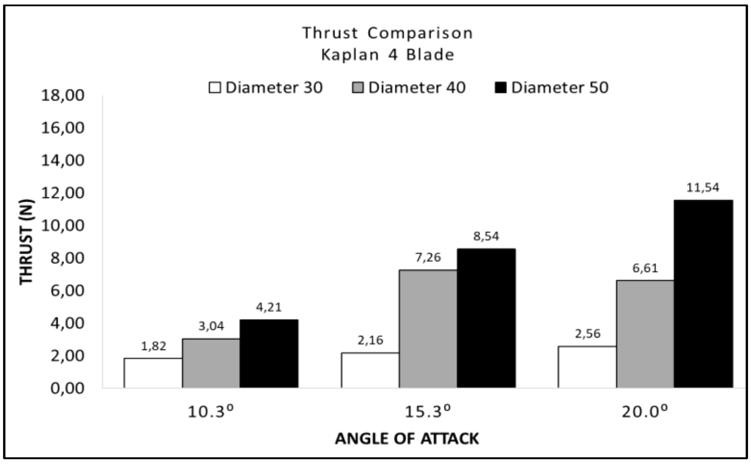


Figure. 10. Thrust comparison of Kaplan 4 blades

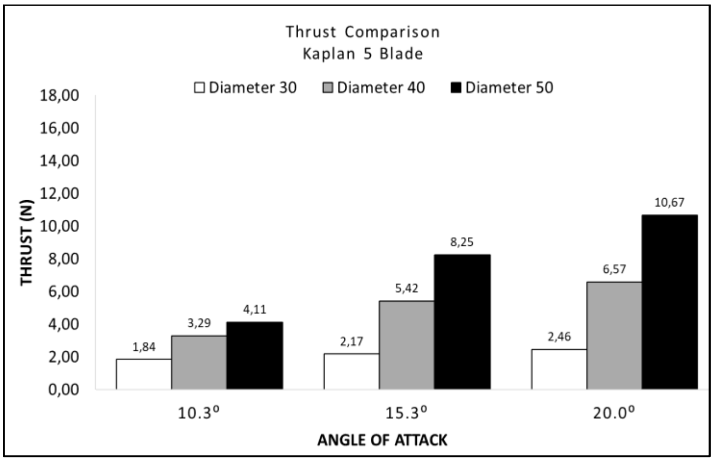


Figure. 11. Thrust comparison of Kaplan 5 blades

Kaplan 3 blades, 4 blades, and 5 blades, with the angle of attack variations 10.3º, 15.3º, and 20º and diameter variations 0.3, 0.4, and 0.5 meters are shown in Figures 12-14 respectively. The results showed that the toque of the propeller model variations is not significantly different, with the highest result produced by the Kaplan series 3 blades, with diameter = 0.5 meter at an angle of attack = 20.0º.

D. Propeller Efficiency Characteristics

To evaluate the propeller efficiency, the simulation result of Kaplan 3, 4, and 5 blades with D=0.3m with the angle of attack variation 10.3º, 15.3º, and 20º taken as an example. The propeller efficiency result is shown in Table 5. The result shows that due to the higher angle of attack, the propeller efficiency increased. It was possible due to the higher thrust value that was achieved at the

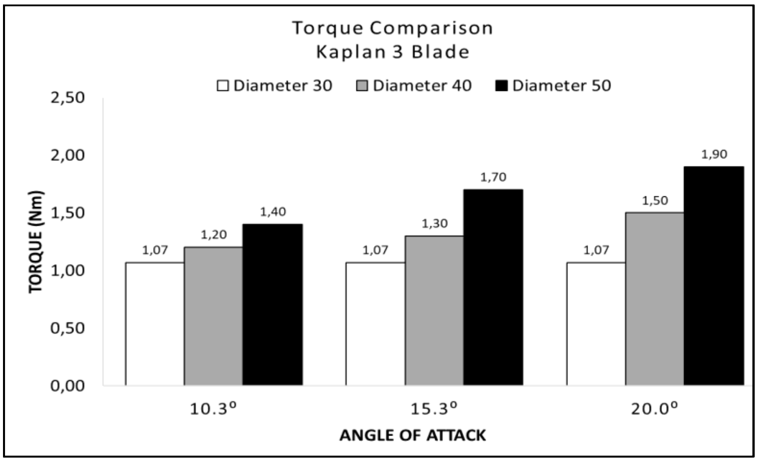


Figure. 12. Torque comparison of Kaplan 3 blades

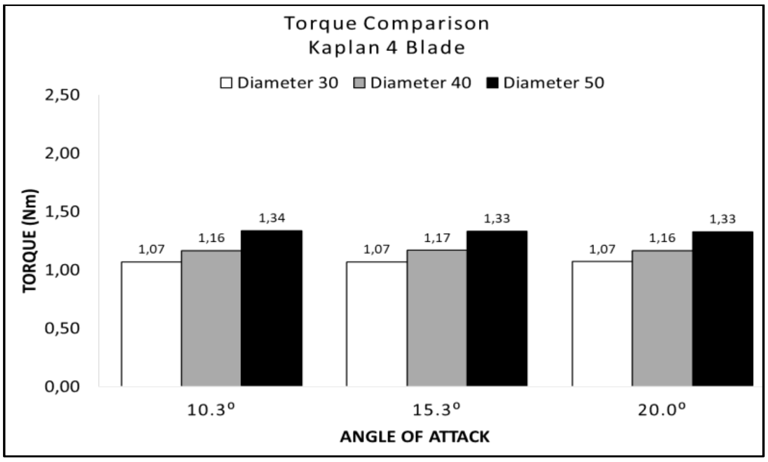


Figure. 13. Torque comparison of Kaplan 4 blades

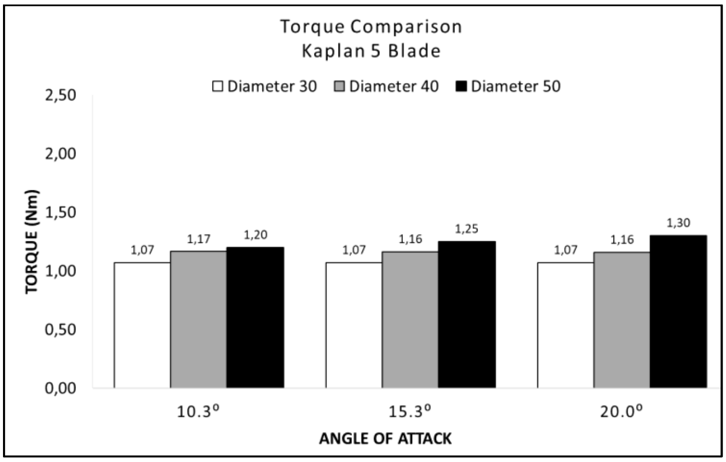


Figure. 14. Torque comparison of Kaplan 5 blades

TABLE 5.
PROPELLER EFFICIENCY

Configuration	Angle of Attack	Propeller efficiency
Kaplan 3 Blades	10.3º	0,50
	15.3º	0,62
	20.0º	0,72
Kaplan 4 Blades	10.3º	0,49
	15.3º	0,58
	20.0º	0,69
Kaplan 5 Blades	10.3º	0,49
	15.3º	0,58
	20.0º	0,66

higher angle of attack. However, the higher the blade number of the propeller, the propeller efficiency is lower.

V. CONCLUSION

The Kaplan series propeller in this study is simulated by using CFD due to the effect of angle of attack and number of blade variations on propeller performance. The numerical study is carried out by variating the number of propeller blades 3, 4, and 5, with diameter *D*= 30, 40, and 50 cm, and angle of attack 10.3º, 15.3º, and 20.0º respectively. The distribution of pressure surface, thrust, and torque tend to increase at the higher diameter and angle of attack. However, it tends to be decreased by the higher blade number of the blade. The CFD results show that the highest torque, thrust, and propeller efficiency were generated by the Kaplan 3 blades with a diameter of 0.5m and an angle of attack of 20.0º. So, it can be concluded that this propeller model has a good performance compared to the other model variations.

REFERENCES

[1] J. Artyszuk. “Wake fraction and thrust deduction during ship astern manoeuvres”. Transactions on the Built Environment Vol 68, © 2003 WIT Press, www.witpress.com, ISSN 1743-3509. Pp. 125-134

[2] MD. Arifin, Danny F, Fanny O, and Karina A. S. “Analysis of the Effect of Changes in Pitch Ratio and Number of Blades on Cavitation on CPP”. International Journal of Marine Engineering Innovation and Research. Vol. 5(4), Dec. 2020. pp. 255-264. DOI: http://dx.doi.org/10.12962/j25481479.v5i4.8285

[3] J S Carlton. “Marine Propellers and Propulsion Second Edition”. Published by Elsevier Ltd. All right reserved. ISBN: 978-07506-8150-6

[4] MD. Arifin, Frengki M.F. “Numerical Study of B-Screw Ship Propeller Performance: Effect of Tubercle Leading Edge”. International Journal of Marine Engineering Innovation and Research. Vol. 6(1), Mar. 2021. pp. 16-23. DOI: http://dx.doi.org/10.12962/j25481479.v6i1.8702

[5] Mohammad Danil Arifin, Frengki Mohamad Felayati and Andi Haris Muhammad 2022. Flow Separation Evaluation on Tubercle Ship Propeller. *CFD Letters*. 14, 4 (May 2022), 43–50. DOI: https://doi.org/10.37934/cfdl.14.4.4350.

[6] Florian Vesting “Marine Propeller Optimisation - Strategy and Algorithm Development”. Thesis for the degree of Doctor of Philosophy. Chalmers Reproservice Gothenburg, Sweden 2015. ISBN 978-91-7597-263-3

[7] MD. Arifin, Frengki M.F. “Cavitation Analysis of Kaplan-Series Propeller: Effect of Pitch Ratio and nProp using CFD”. International Journal of Marine Engineering Innovation and Research. Vol. 6(2), June. 2021. pp. 114-124. DOI: http://dx.doi.org/10.12962/j25481479.v6i2.8747

[8] Mani, R., Gobiraj R., M. Mohamed Abbas. “Design and Analysis of Marine Propeller Using Computational Fluid Dynamics”. International Advanced Research Journal in Science, Engineering and Technology Vol. 8, Issue 7, July 2021 DOI: 10.17148/IARJSET.2021.8784

[9] Mina Tadros, Guedes Soares, Carlos Guedes Soares. “Design of Propeller Series Optimizing Fuel Consumption and Propeller Efficiency”. Journal of Marine Science and Engineering 9(11):1226.

[10] S. Kaidi; H. Smaoui; and P. Sergent. “CFD Investigation of Mutual Interaction between Hull, Propellers, and Rudders for an Inland Container Ship in Deep, Very Deep, Shallow, and Very Shallow Waters”. November 2018Journal of Waterway, Port, Coastal and Ocean Engineering 144(6).

[11] Ghaemi, Mohammad Hossein and Zeraatgar, Hamid. "Impact of Propeller Emergence on Hull, Propeller, Engine, and Fuel Consumption Performance in Regular Head Waves" Polish Maritime Research, vol.29, no.4, 2022, pp.56-76. https://doi.org/10.2478/pomr-2022-0044

[12] H K Versteeg and W Malalasekera. " An Introduction to Computational Fluid Dynamics the Finite Volume Method Second Edition" Printed and bound by Bell & Bain Limited, Glasgow. ISBN: 978-0-13-127498-3.

[13] Tarek J. Jamaleddine & Madhumita B. Ray (2010). “Application of Computational Fluid Dynamics for Simulation of Drying Processes”: A Review, Drying Technology, 28:2, 120-154, DOI: 10.1080/07373930903517458

[14] Nasser Ashgriz & Javad Mostaghimi. “An Introduction to Computational Fluid Dynamics Chapter 20 in Fluid Flow Handbook