Journal Applied Science and Technology on Naval Engineering

Vol. 4, No. 1, February 2026: page 26-32
ISSN: 3031-8866
DOI: 10.54378/astne.v4i2.13485

Cavitation Analysis of a Propeller Based on Variations in Water Temperature and Propeller Rotational Speed Using CFD Simulation

Dian Febriyanti[a], Fakhri Akbar Ayub[b,*]

[a]Teknik Perkapalan, Universitsas Pembangunan Nasional Vetrean Jakarta, Jakarta 12450 Indonesia, *2110313051@mahasiswa.upnvj.ac.id*
[b] Teknik Perkapalan, Universitsas Pembangunan Nasional Vetrean Jakarta, Jakarta 12450 Indonesia, *fakhriakbar@upnvj.ac.id*

Abstract

This study aims to understand how the cavitation phenomenon occurs in ship propellers by varying the water flow temperature and propeller rotation speed. This study investigates cavitation characteristics on the INSEAN E779A marine propeller using Computational Fluid Dynamics (CFD). In ship operations, propeller performance plays a crucial role in propulsion efficiency. However, propellers often experience cavitation phenomena during operation, which can reduce thrust performance and cause surface damage. This study uses Computational Fluid Dynamics (CFD) software. Variations in temperature and rotational speed are applied to see the differences in the results of the propeller thrust coefficient values in each condition. The simulation results show that changes in fluid temperature do not have a significant effect on the occurrence of cavitation. However, increasing the propeller rotational speed clearly increases the potential for cavitation. At a speed of 800 RPM cavitation has not been seen, either at temperatures of 17°C, 21°C, and 30°C. However, at a speed of 1600 RPM cavitation appears, both at temperatures of 17°C, 21°C, and 30°C. This shows that temperature does not have a significant effect, but rotational speed is a very influential factor in the process of cavitation and propulsion efficiency.

Keywords: *propeller, cavitation, temperature, rotation speed*;

1. Introduction

Ships are one of the main modes of transportation that play an important role in global mobility and distribution. To support efficient ship operations, the propulsion system is a crucial component. The propeller, as the primary propulsion device, generates thrust through lift forces produced by rotating blades interacting with the surrounding fluid. Therefore, optimal propeller performance is essential to achieve the desired ship performance.

During operation, propellers often experience cavitation phenomena. Cavitation occurs when the local pressure around the propeller surface drops below the vapor pressure of the fluid, resulting in the formation of vapor bubbles (Long et al., 2023). This phenomenon typically appears in low-pressure regions, particularly along the blade tip vortex path, leading to irregular pressure distribution, increased noise, and more complex flow behavior compared to non-cavitating conditions (Cheng et al., 2024). Cavitation can be triggered by various factors, including blade design, propeller rotational speed, local pressure, and fluid temperature, with excessively low local pressure being the primary cause.

Previous studies have shown that variations in pitch and propeller rotational speed significantly influence the extent of cavitation (Arifin & Felayati, 2021), and the use of CFD simulations is recommended to predict high-risk regions during the design stage (Jin et al., 2024). However, most existing studies have focused on uniform flow conditions and limited propeller rotational speed variations (Aparna et al., 2023). In fact, propeller rotational speed and fluid temperature also play important roles in cavitation occurrence (Arazgaldi et al., 2009), and temperature differences can affect cavitation characteristics on marine propellers (Yusvika et al., 2020).

Therefore, this study analyzes cavitation phenomena on a marine propeller by varying fluid temperature and propeller rotational speed. The analysis is conducted using Computational Fluid Dynamics (CFD) to provide a clearer understanding of cavitation characteristics on marine propellers.

* 2110313051@mahasiswa.upnvj.ac.id

2. Literature Review

Lift force is a force that acts perpendicular to the direction of motion of an object in a fluid and is generated by pressure differences on the surface of a blade or foil. In a ship propeller, lift force is produced by the rotating blades as they interact with the surrounding water and serves as the primary mechanism for generating thrust. However, due to the finite span of the propeller blades, pressure equalization tends to occur near the blade tips, leading to a reduction in lift force and consequently affecting the overall performance of the propeller.

The lift force generated by the propeller blades is subsequently converted into thrust, which propels the ship forward. Thrust arises as a reaction to the mass of water accelerated backward by the rotation of the propeller. The magnitude of thrust is influenced by several parameters, including propeller dimensions, rotational speed, advance velocity, fluid viscosity, and static pressure. To evaluate propeller performance in a more general and comparable manner, thrust is commonly expressed in terms of a non-dimensional parameter known as the thrust coefficient. The following equation represents the formulation of the thrust coefficient.

$$Kt = T/\rho n^2 D_2^4 \tag{1}$$

Where $K_t$is the thrust coefficient, $T$is the thrust force (N), $\rho$is the fluid density (kg/m³), $n$is the propeller rotational speed (rev/s), and $D$is the propeller diameter (m). Under certain operating conditions, particularly when the local pressure around the blade surface decreases significantly, propeller performance can be adversely affected by cavitation. Cavitation occurs when the local pressure falls below the vapor pressure of the fluid, resulting in the formation of vapor bubbles near the blade surface. Although vapor pressure is a function of fluid temperature, cavitation typically occurs due to pressure reduction under relatively constant temperature conditions, distinguishing it from boiling, which is primarily driven by temperature increase.

The occurrence of cavitation in propellers poses a serious challenge in marine propulsion systems, as it can lead to performance degradation, increased noise and vibration, and material damage due to erosion. The initial cavitation process involves the formation of low-pressure cavities that are rapidly filled with vapor, making cavitation a highly dynamic phenomenon that can appear and disappear during propeller rotation.

To predict and evaluate the tendency of cavitation to occur, the cavitation number is used as a non-dimensional parameter. The cavitation number compares the difference between local pressure and vapor pressure to the dynamic pressure of the flow. Its value varies along the blade surface and changes throughout the propeller's rotational process, making it a crucial indicator in cavitation analysis and propeller performance evaluation. The cavitation number can be defined as follows:

$$\text{Cavitation Number} = \sigma = \frac{(pO - p_v)}{\frac{1}{2}pV^2} \tag{2}$$

where σ is the cavitation number, p_∞ is the reference pressure (Pa), $p_v$is the vapor pressure of water (Pa), ρ is the fluid density (kg/m³), and V is the inflow velocity (m/s).

3. Research Method

Lift This research was conducted through a series of systematic stages, starting from problem identification to the achievement of analytical results. The entire research process employed Computational Fluid Dynamics (CFD) simulations; therefore, a proper understanding of CFD software operation is essential to ensure accurate and reliable simulation outcomes.

The initial stage of the study involved problem identification and formulation, including the determination of research background, objectives, problem limitations, and the applied methodology. This stage was intended to ensure that the research has a clear direction and well-defined scope in addressing the investigated problem.

Subsequently, a literature review was carried out as the theoretical foundation of the study. This process involved reviewing relevant journals, books, and scientific articles related to propeller performance and cavitation, which

supported the selection of models, methods, and simulation parameters.
The data collection stage was conducted by obtaining propeller data from reliable reference sources. These data were then used as the basis for modeling and performing CFD simulations to analyze cavitation phenomena on the propeller.

3.1 Propeller Dimensions and Model

Based on the reference journal, the propeller model data used in this study were obtained. Table 1 presents the propeller dimensions.

Table 1. Propeller Dimensions

Propeller	**Dimension**
Propeller Diameter	D_p = 227.27 mm
Number of Blade	Z = 4
Skwe Angle at Blade Up	$\theta^{tip} = 4^0 48'$ (*positive*)
Rake (nominal)	$I = 4^0 35'$ (*forward*)
Expanded Area Ratio	EAR = 0.689
Hub Diameter (at prop. Ref. Line)	D_H = 45.53 mm
Hub Length	L_H = 68.30 mm

Based on the obtained propeller dimension data, the propeller was subsequently modeled according to the given specifications. This modeling process aims to accurately represent the geometric shape and characteristics of the propeller, which serves as the basis for the subsequent CFD simulation.

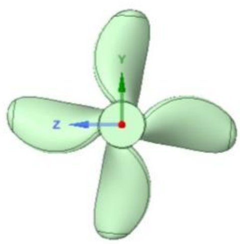

Figure 1. Model Propeller

3.2 Computational Fluid Dynamic Simulation

This study was conducted using Computational Fluid Dynamics (CFD) software. The simulation process consisted of several main stages, including meshing and solver setup. The meshing stage was performed to divide the computational domain into smaller elements in order to improve the accuracy and numerical stability of the simulation.

After the meshing process, the simulation was carried out by defining the appropriate numerical and physical settings. One of the most important steps in the CFD simulation is the determination of boundary conditions, as they govern the interaction between the fluid and the defined domain boundaries and significantly influence the analysis results.

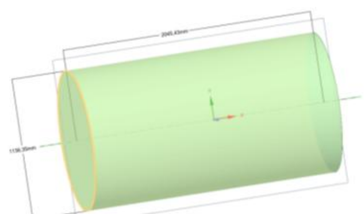

Figure 2. Static Domain

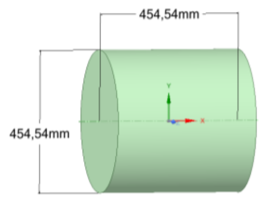

Figure 3. Rotating Domain

In this study, a rotating domain with a diameter of two times the propeller diameter was applied around the propeller. The static domain had a diameter of five times the propeller diameter. The inlet boundary was positioned at a distance of four times the propeller diameter upstream of the propeller, while the outlet boundary was located five times the propeller diameter downstream. The applied boundary conditions follow the reference study titled *"A numerical investigation of the effect of blade number on the performance of an INSEAN E779A marine propeller in a cavitating flow using computational fluid dynamics."*

Subsequently, a validation was performed to evaluate the accuracy of the simulation results. The validation graphs are shown in Figure 4.

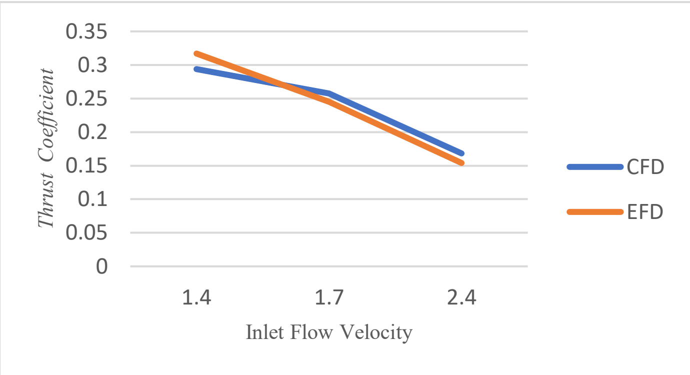

Figure 4. Comparasion of Kt (Validation)

3.3 Variation

After model validation, variations in fluid temperature and propeller rotational speed were applied to analyze cavitation characteristics and changes in the thrust coefficient. The propeller rotational speeds were set to 800 RPM and 1600 RPM based on reference studies. The seawater temperature variations were selected from the environmental conditions of the Sunda Strait, the Mediterranean Sea, and the Atlantic Ocean

Table 2. Temperature Variation Values

Temperature	**Region**
17°C	Mediterranean Sea
21°C	Atlantic Ocean
30°C	Sunda Strait

4. Result and Discussion

After the CFD simulations were performed, results were obtained based on variations in fluid temperature and propeller rotational speed. At a rotational speed of 800 RPM with temperature variations of 17°C, 21°C, and 30°C, the volume fraction contours show a value of 0.00 over the entire propeller surface, indicating that no vapor formation occurs and cavitation does not take place under these conditions. Although the fluid temperature was varied, the changes were not sufficient to reduce the local pressure below the saturated vapor pressure of the fluid; therefore, temperature is not the primary factor triggering cavitation in this case.

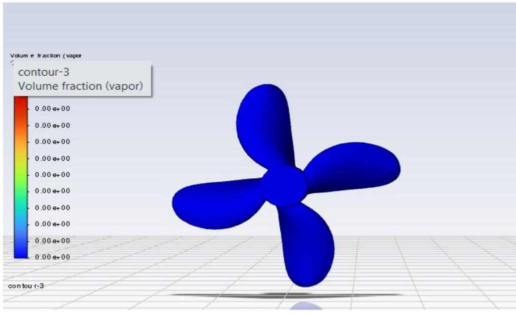

Figure 5. Volume Fraction Contours at 800 RPM

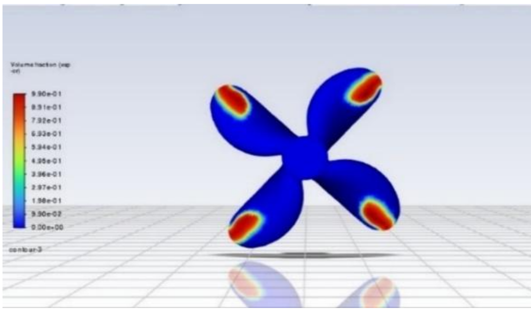

Figure 6. Volume Fraction Contours at 1600 RPM

At a higher rotational speed of 1600 RPM, cavitation begins to appear at the blade tips, as indicated by the presence of red regions in the volume fraction contours. The cavitation patterns observed at temperatures of 17°C, 21°C, and 30°C show similar characteristics, with cavitation consistently occurring at the blade tips. This behavior is attributed to the high relative fluid velocity in these regions, which causes the local pressure to drop below the vapor pressure of the fluid. Although the cavitation pattern remains similar, the cavitation area tends to increase with increasing temperature due to the rise in water vapor pressure from 1930 Pa at 17°C to 4240 Pa at 30°C, making the fluid more susceptible to vapor formation when the local pressure decreases.

Table 3. Thrust Value and Thrust Coefficient

Inlet Velocity (m/s)	**Propeller Rotational Speed (rpm)**	**Temperature (°C)**	**Thrust (N)**	**Thrust Coefficient**
1,7	800	30	142,26	0,066052
1,7	800	21	139,5	0,0647008
1,7	800	17	142,16	0,066006
1,7	1600	30	626,4	0,29084
1,7	1600	21	608,3	0,282436
1,7	1600	17	631,5	0,293208

Table 4. Thrust Coefficient under Non-Cavitating and Cavitating Conditions

Inlet Velocity (m/s)	Propeller Rotational Speed (rpm)	Temperature (°C)	Non-Cavitation Thrust Coefficient	Cavitation Thrust Coefficient
1,7	800	30	0,25675	0,066051975
1,7	800	21	0,25675	0,064700849
1,7	800	17	0,25675	0,066005545
1,7	1600	30	0,25675	0,290840414
1,7	1600	21	0,25675	0,2824365
1,7	1600	17	0,25675	0,293208367

The comparison of thrust coefficient values under non-cavitating and cavitating conditions indicates that cavitation leads to a reduction in propeller performance. Under non-cavitating conditions with an inlet flow velocity of 1.7 m/s, the thrust coefficient value of 0.25675 represents stable and efficient propeller performance. However, under cavitating conditions, the thrust coefficient decreases significantly, particularly at a propeller rotational speed of 800 RPM, indicating a reduction in thrust due to cavitation. At a higher rotational speed of 1600 RPM, a decrease in the thrust coefficient is still observed but is less pronounced compared to that at 800 RPM. These results demonstrate that propeller rotational speed has a significant influence on cavitation occurrence and thrust reduction, while the effect of fluid temperature is relatively insignificant.

5. Conclusion

Based on the CFD simulation results of the INSEAN E779A propeller with variations in water temperature and rotational speed, it can be concluded that cavitation is more strongly influenced by propeller rotational speed than by flow temperature. Cavitation begins to appear at a rotational speed of 1600 RPM, whereas at 800 RPM cavitation has not yet developed significantly. Variations in flow temperature do not show a significant effect on cavitation patterns. Furthermore, the presence of cavitation leads to a reduction in the thrust coefficient compared to non-cavitating conditions, and cavitation initially occurs at the blade tip region of the propeller.

Reference

Aparna, P., Padmaja, P., Pothanna, N., & Ramana Murthy, J. V. (2023). Uniform Flow of Viscous Fluid Past a Porous Sphere Saturated with Micro Polar Fluid. *Biointerface Research in Applied Chemistry*, *13*(1). https://doi.org/10.33263/BRIAC131.069

Arazgaldi, R., Hajilouy, A., & Farhanieh, B. (2009). Experimental and numerical investigation of marine propeller cavitation. *Scientia Iranica*, *16*(6 B), 525–533.

Arifin, M. D., & Felayati, F. M. (2021). *Cavitation Analysis of Kaplan-Series Propeller : Effect of Pitch Ratio and nProp using CFD*. *6*(2), 114–124.

Cheng, Z., Kashyap, S., Smoker, B., Burella, G., Jaiman, R., Columbia, B., Engineers, M., & Columbia, B. (2024). *Proceedings of the OMAE 2024 43rd International Conference on Ocean, Offshore & Arctic Engineering OMAE2024 June 9–14, 2024, Singapore*. 1–12.

Lier, R., Duclut, C., Bo, S., Armas, J., Jülicher, F., & Surówka, P. (2023). Lift force in odd compressible fluids. *Physical Review E*, *108*(2), 1–7. https://doi.org/10.1103/PhysRevE.108.L023101

Long, Y., Zhang, M., Zhou, Z., Zhong, J., An, C., Chen, Y., Wan, C., & Zhu, R. (2023). Research on Cavitation Wake Vortex Structures Near the Impeller Tip of a Water-Jet Pump. *Energies*, *16*(4). https://doi.org/10.3390/en16041576

Lumbanraja, K. C. P., Chrismianto, D., & Samuel, S. (2021). Perhitungan Nilai Maksimum Thrust, Torque, dan Efficiency Propeller Tipe B-4 Series pada Kapal Tugboat dengan Modifikasi Diameter, Rake, dan Pitch menggunakan Metode Komputasi. *Jurnal Rekayasa Hijau*, *5*(1), 63–78. https://doi.org/10.26760/jrh.v5i1.63-78

Salvatore, F., Pereira, F., Felli, M., Calcagni, D., & Felice, F. Di. (2022). *Description of the INSEAN E779A Propeller Experimental Dataset Rapporto Tecnico INSEAN / Technical Report / 2006-085 Description of the INSEAN E779A Propeller Experimental Dataset* (Issue March). https://doi.org/10.5281/zenodo.6077997

Subhas, S. (2012). *CFD Analysis of a Propeller Flow and Cavitation*. *55*(16), 26–33.

Test, M. (1998). *Hydrodynamics of Ship PropellersContents*. 1–21. papers2://publication/uuid/A8943D68-4A4D-4C91-AB8B-EE4B5F40E008

Testa, C., Ianniello, S., Salvatore, F., & Pereira, F. (2006). Theoretical Modelling of Unsteady Cavitation and Induced Noise. *Sixth International Symposium on Cavitation, CAV2006*, *August*.

Tupper, E. C. (2013). Chapter 8. Propulsion. In *Introduction to Naval Architecture* (Fifth Edit). Elsevier Ltd. https://doi.org/10.1016/B978-0-08-098237-3.00008-4