SEMARAK ILMU PUBLISHING

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

Benchmark Study of KP505 Propeller on the Performances of Indonesian Traditional Fishing Vessel Propulsion by using OpenFOAM

Aldias Bahatmaka[1,*], Deni Fajar Fitriyana[1], Samsudin Anis[1], Rizki Setiadi[1], Kriswanto[1], Dony Hidayat Al-Janan[1], Sonika Maulana[1], M. Hilman Gumelar Syafei[1], Febri Budi Darsono[1], Aditya Rio Prabowo[2], Samuel Samuel[3], Rifky Ismail[4], Teguh Muttaqie[5], Achmad Yanuar Maulana[6,7], Mamad Tamamadin[8,9], Sang Won Lee[10], Joung Hyoung Cho[11]

1 Department of Mechanical Engineering, Universitas Negeri Semarang, Gunungpati, Semarang 50229, Indonesia
2 Department of Mechanical Engineering, Universitas Sebelas Maret, Surakarta 57126, Indonesia
3 Department of Naval Architecture, Diponegoro University, Tembalang, Semarang 50275, Indonesia
4 Department of Mechanical Engineering, Diponegoro University, Tembalang, Semarang 50275, Indonesia
5 Research Center for Hydrodynamics Technology, National Research and Innovation Agency (BRIN), Surabaya 60112, Indonesia
6 Department of Chemistry, Dong-A University, Busan, 49315, South Korea
7 Association of Indonesian Researchers in Korea (APIK), Seoul 07342, South Korea
8 University Core Research Center for Disaster-free & Safe Ocean City Construction, Dong-A University, Busan 49315, South Korea
9 Atmospheric Science Research Group, Bandung Institute of Technology, Bandung 40132, Indonesia
10 Department of Fluid, Research and Development, Daewoo Shipbuilding and Marine Engineering, Co. Ltd., Geoje 53305, South Korea
11 Interdisciplinary Program of Marine Design Convergence, Pukyong National University, Busan 48513, South Korea

ARTICLE INFO

Article history:
Received 20 April 2024
Received in revised form 23 July 2024
Accepted 5 August 2024
Available online 30 August 2024

Keywords:
Benchmark; CFD; ship propulsion; performances; OpenFOAM

ABSTRACT

The strategy of design engineering is very important to produce the modernization of technology. The propeller is one of crucial part in the marine design especially for the marine transportation especially for traditional fishing vessel. In this paper, presents a study for analysis of the propeller performance by using numerical simulation in the computational fluid dynamics (CFD). Since the development of CFD simulation, still facing several problems to solve the CFD analysis. The OpenFOAM is open source CFD tool used in this study in order to investigate the propeller performances. OpenFOAM is implemented for open water performances of KCS propeller (KP505). Reynold-averaged Navier Stokes equations (RANS) was applied to estimate the turbulence of K-ω SST model. Unstructured mesh in this simulation was employing a hexahedral mesh as mesh generation. The multiple rotating frame (MRF) was produced as method to analysis of propeller performances. The velocity inlet and pressure outlet were put for inflow and outflow boundaries. The wall was set to a far-field boundary. The current study compared to the experimental result for the thrust and torque coefficient to validate the propeller performances. By using MRF technique, the simulation of interaction for the propeller with a relative error less than 2%. Thus, The MRF technique can be used for the practical technique to simulate the ship propeller model especially for the moving multiple zones.

* *Corresponding author.*
E-mail address: aldiasbahatmaka@mail.unnes.ac.id

https://doi.org/10.37934/arfmts.120.2.3946

1. Introduction

Nowadays, research of propeller is being increase trend by using the numerical approach to solve the experiment problem, especially in the ship propulsion design. The experiment is having risk and also the cost of the production on the marine industry. The numerical simulation is being famous and also as a good option to predict the performances in engineering sectors. In the numerical simulation, there are several tools and methods to investigate and analysis of marine propulsion design. The Computational Fluid Dynamics (CFD) contributes in the design processes. The CFD method is commonly as the potential flow theory which is applied the Navier-Stokes equations. Since it was complicated to complete, and it can be done by implementation of Reynold-Averaged Navier-Stokes equation (RANS) to finish the numerical simulation [1,2]. Some of researchers conducted for the numerical approach using CFD as a method to simulate the propeller thrust [3-7].

At this the current study, the OpenFOAM is applied to investigate the KCS (KP505) propeller performance. The OpenFOAM is open-source packages of CFD libraries and tool which could analysis the numerical simulation. The OpenFOAM is being potential CFD tool which has been conducted by several researchers [8,9]. In the marine propulsion trend, the design analysis is considered the future research such as the optimization of the propeller design which has been done by using genetic algorithm was conducted by Bahatmaka *et al.,* [10]. The use of kort nozzle or duct of propeller influence the propeller thrust and the efficiency of the performance [11-13]. The RANS code application for the ducted in propeller open water test has been produced the measurements of the simulation compared to the experiments as the validation [14,15].

On the other analysis of the propeller for the hydrodynamic characteristics are also needed as the contribution for solving the marine design and simulation [16-19].

Thus, the OpenFOAM is a powerful CFD Open Source tool in order to complete the numerical simulation of a propeller in an open water test, it can be evidenced by using the benchmark or validation of experimental results. The proposed research calculates the coefficient of the trust and the torque of the propeller by comparing the mesh generation and also compared to the experimental result for the KP505 propeller.

2. Methodology

2.1 Model and Governing Equations

The simulation used KCS propeller (KP505). KP505 is five-blade propeller with skew angle which has been designed for the ship. The technical drawing of propeller shown in Figure 1. The database and the main particulars are applied for the simulation. The sections of propellers are according to NACA66. The list of principal dimensions of propeller can be seen in Table 1 and the offset table of KP505 propeller is listed in Table 2.

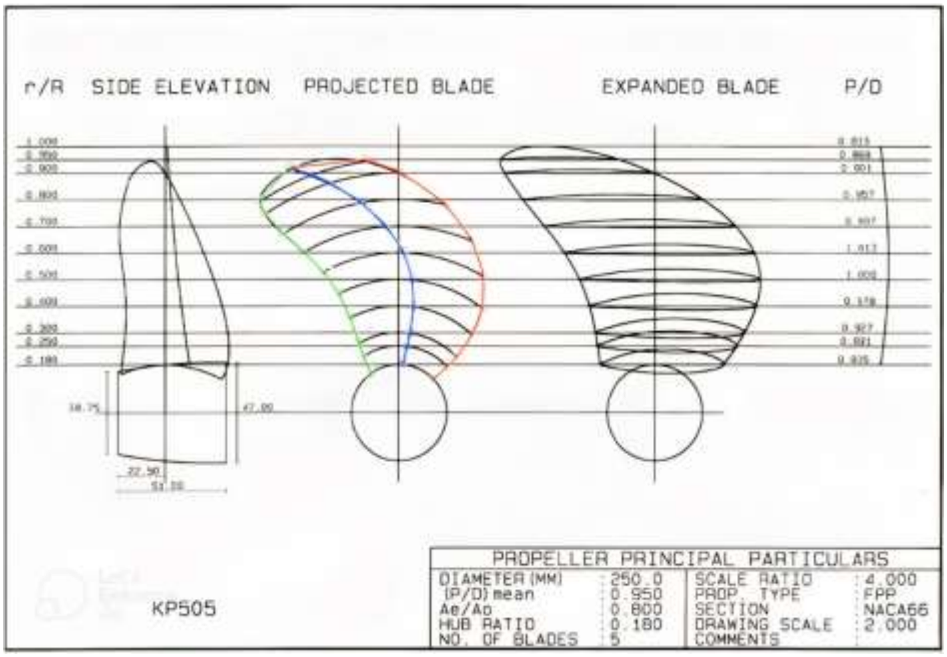

Fig. 1. Technical drawing of KP505 propeller

Table 1
Principal dimension of KP505 propeller

Type	Unit	Value
Diameter (D)	mm	250.00
Ratio of blade expanded area (Ae/Ao)	-	0.80
Ratio of the propeller hub (H)	-	0.18
Number of the blades (Z)	-	5.00

Table 2
Offset table of KP505 propeller

r/R	P/Dp	θ (deg)	C/Dp
0.18	0.8347	-4.72	0.2313
0.25	0.8912	-6.98	0.2618
0.3	0.9269	-7.82	0.2809
0.4	0.9783	-7.74	0.3138
0.5	1.0079	-5.56	0.3403
0.6	1.013	-1.5	0.3573
0.7	0.9967	4.11	0.3590
0.8	0.9566	10.48	0.3376
0.9	0.9006	17.17	0.2797
0.95	0.8683	20.63	0.2225
1.0	0.8331	24.18	0.0001

By using the equation of the Reynold-Average Navier Stokes (RANS), open water test is used as a large number of problems. The velocity for the thrust and torque is the main objective which depend on the mean flow characteristics. It can be described as below

$$u = \bar{u} + u' \tag{1}$$

$$p = \bar{p} + p' \tag{2}$$

where $\bar{u}$ denotes as velocity, $\bar{p}$ denotes as pressure. And u' denotes as fluctuating of velocity an p' as fluctuating pressure. The detail of term written in Eq. (1) and Eq. (2). The Navier-Stokes equation as the time-average continuity as a replacement of velocity and pressure. It can be described as

$$\frac{\partial \bar{u}}{\partial t} + \nabla.(\bar{u}.\bar{u}) - \nabla.(v\nabla\bar{u}) = -\nabla\bar{p} + \nabla.(\bar{u}'\bar{u}') \quad (3)$$

where the Reynold stress tensor is denoted for $\bar{u}\bar{u}$ in Eq. (3) which relate to turbulence flow as the fluctuating variable. The coefficient of the thrust and the torque for propeller characteristics conventionally was explained by Willemsen [20] and written as below

$$K_T = \frac{T}{\rho n^2 D^5} \quad (4)$$

$$K_Q = \frac{Q}{\rho n^2 D^5} \quad (5)$$

$$J = \frac{V}{nD} \quad (6)$$

The propeller efficiency is described as

$$\eta_0 = \frac{K_T}{K_Q}\frac{J}{2\pi} \quad (7)$$

where η0 denotes the efficiency and π denotes as the motion in circle which shown in Eq. (7).

2.1 Model and Governing Equations

In this present study, Moving Rotation Frame (MRF) strategy is implemented to compute the simulation and boundaries. The zone is divided to be two zones. The rotational zone is the zone around the propeller, and the other is the stationary zone. the velocity inlet and pressure outlet are obtained as the inflow and outflow boundaries. The wall was set to the far field boundary. The detail boundaries can be seen in Figure 2. The MRF conditions in the OpenFOAM employed an outer and a small cylinder. The small cylinder is the rotating domain and outer is stationary. For the set up boundaries in the OpenFOAM is listed in Table 3.

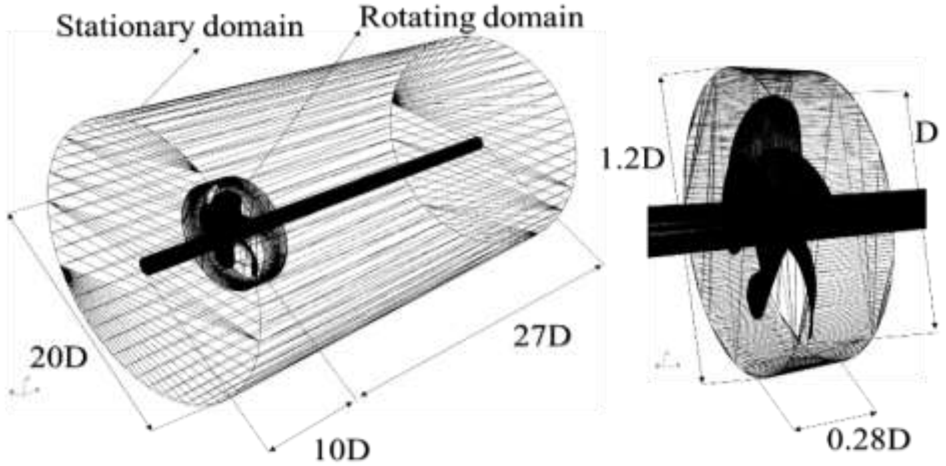

Fig. 2. The MRF boundaries

Table 3
The boundary setting in OpenFOAM

The boundary	P	U	k	epsilon
Wall	symmtryPlane	symmtryPlane	symmtryPlane	symmtryPlane
Blade/Hub	wallFunction	fixedValues	wallFunction	wallFunction
Inlet	zeroGradient	fixedValues	fixedValues	fixedValues
Outlet	fixedValues	inletOutlet	inletOutlet	inletOutlet

The domain consists of an internal rotating cylinder enclosing the propeller and an external stationary cylinder with a radius of 10D. In addition, the inlet uniform boundary condition is located at 10D upstream of the propeller plane, and the outlet that is a constant pressure condition was located at 27D downstream of the propeller. The MRF method is implemented as the condition of steady state pressure-based solver. The k-ω SST model is used in this simulation and analysis. The advance coefficient (J), the thrust coefficient (KT), the torque coefficient (KQ), and the open water efficiency (η) as parameters of calculation on hydrodynamic performances of a propeller. The steady-state solver employing the SimpleFoam as solver of the flow follows the blade rotation according to the Moving Reference Frame (MRF) method.

3. Results
3.1 Benchmarking Study

The present study compared results from numerical simulation and experimental result [21]. OpenFOAM is implemented in order to fix the ship propeller simulation in open water condition. The mesh strategy is divided several conditions such as coarse, medium, fine, and the finest mesh. The grid mesh of the model is compared to the experiments. According to the previous studies, the advance of velocity in range 0.7 ≤ J ≤ 0.9, and the result mostly closer to the experimental result [22-24]. For this study, the simulation performed at J value of 0.5. It is the state of art for this current research and to calculate the phenomenon with the possibility of this configuration. The comparison against experiments for all grids are shown in Figure 3.

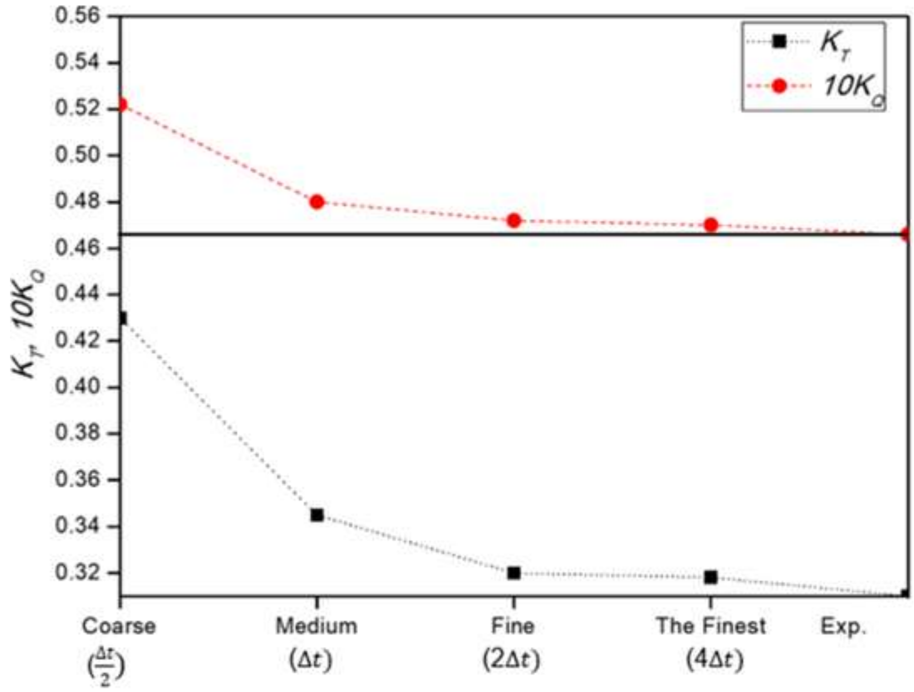

Fig. 3. Benchmark study to experimental data

The unstructured mesh was selected in the meshing process which are implemented to the numerical simulation with hexahedral mesh. The four different time steps were used from the coarse to the finest mesh. The grid meshes are calculated the coefficient of the thrust and the torque. The detail of grid meshes validation can be seen in Table 4.

Table 4
Grid mesh validation

Results	Coarse (0.3M)	Medium (1.5M)	Fine Mesh (5M)	The Finest Mesh (8M)	Exp. [21]
KT	0.430	0.345	0.320	0.318	0.310
KQ	0.0522	0.0480	0.0472	0.0470	0.0466

The biggest fluctuation flow occurs on the coarse mesh compared with the others. The simulation result is only slightly different from the experimental result for the KT and KQ. It means the validation reach shows acceptable achievement. The initial mesh created consisted of coarse mesh 0.3M (Million elements), medium mesh 1.5M, fine mesh 5M, and the finest mesh 8M. At the present study, the ducted propeller was investigated with Computational Fluid Dynamics (CFD) by using MRF method. The MRF method is used for this simulation and the result shows a similar value to the experimental. The MRF method relative error of the thrust under 2% has a good in agreements. This research is in accordance with the theory in previous related research of MRF [25].

3.2 Post-processing Result

Figure 4 and Figure 5 show the flow motion appears near the propeller tip and a large velocity gradient occurs in the wake sheet and by using the MRF approach, it can be seen that the propeller has stronger flow from the simulation also has a minimum pressure in the same advance velocity. It means the MRF has a good result and could performed the best rotational velocity. Therefore, the MRF is useful for modelling of interaction around propeller blade with a relative error of the thrust below 2% and the MRF is considered the practical way to model moving multiple zones and it fits on the theory.

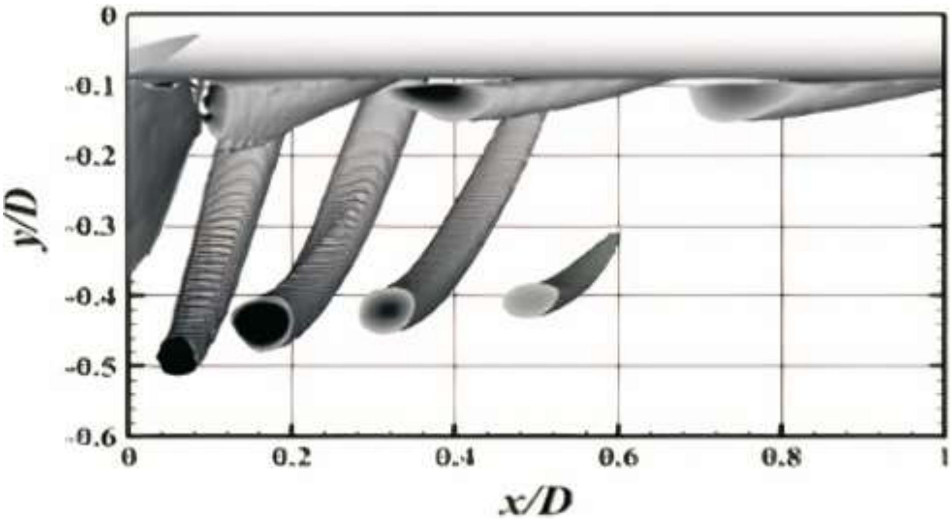

Fig. 4. Iso-surface conditions during acceleration for the advance coefficient 0.5

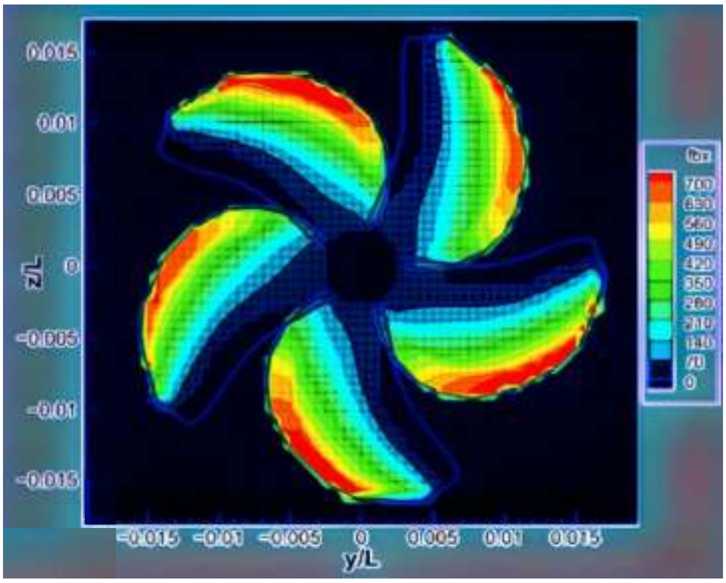

Fig. 5. Body force distribution for the advance coefficient 0.5

4. Conclusions

This research has employed the RANS equations for using the OpenFOAM. This current research investigates hydrodynamics performance of the KCS propeller (KP505) and conducted in the open water test. The simulation is produced a comparison of numerical simulation in several meshing strategies to experiments.

Observing the results, the MRF technique having a good agreement results to experimental result. MRF was applied in order to fix the propeller calculation for both the coefficient of the thrust and the torque in an open water test. Thus, the MRF is good technique for simulation of interaction of the propeller with a relative error less than 2%. The MRF can be used for the practical technique to simulate the ship propeller model especially for the moving multiple zones.

For the propeller simulation in Open-Water test, the cavitation analysis and the self-propulsion simulation and analysis should be conducted around the rudder and propeller. In order to achieve better accuracy, another work that should be considered is about meshing quality using the panel method to get the best interface on the post-processing of the simulations. Further works can include investigation of grid dependency in detail. More refinement of the mesh around the edge of the models is needed.

Acknowledgement
This research was funded by a grant from the DPA Faculty of Engineering, Universitas Negeri Semarang, Indonesia under scheme of "Penelitian Dosen" (DPA FT) – Year 2024 (Grant 118.22.3/UN37/PPK.05/2024).

References

[1] Samuel, Samuel, Rizal Kurnia Praja, Deddy Chrismianto, Muhammad Luqman Hakim, Ahmad Fitriadhy, and Aldias Bahatmaka. "Advancing Interceptor Design: Analyzing the Impact of Extended Stern Form on Deep-V Planing Hulls." *CFD Letters* 16, no. 5 (2024): 59-77. https://doi.org/10.37934/cfdl.16.5.5977

[2] Niknahad, Ali, and Abdolamir Bak Khoshnevis. "Numerical study and comparison of turbulent parameters of simple, triangular, and circular vortex generators equipped airfoil model." *Journal of Advanced Research in Numerical Heat Transfer* 8, no. 1 (2022): 1-18.

[3] Chase, Nathan, and Pablo M. Carrica. "Submarine propeller computations and application to self-propulsion of DARPA Suboff." *Ocean Engineering* 60 (2013): 68-80. https://doi.org/10.1016/j.oceaneng.2012.12.029

[4] Bahatmaka, Aldias, Dong-Joon Kim, Deddy Chrismianto, Joga Dharma Setiawan, and Aditya Rio Prabowo. "Numerical investigation on the performance of ducted propeller." In *MATEC Web of Conferences*, vol. 138, p. 07002. EDP Sciences, 2017. https://doi.org/10.1051/matecconf/201713807002
[5] Bhattacharyya, Anirban, Vladimir Krasilnikov, and Sverre Steen. "A CFD-based scaling approach for ducted propellers." *Ocean Engineering* 123 (2016): 116-130. https://doi.org/10.1016/j.oceaneng.2016.06.011
[6] Chamanara, Mehdi, and Hassan Ghassemi. "Hydrodynamic characteristics of the Kort-nozzle propeller by different turbulence models." *American Journal of Mechanical Engineering* 4, no. 5 (2016): 169-172.
[7] Go, J. S., H. S. Yoon, and J. H. Jung. "Effects of a duct before a propeller on propulsion performance." *Ocean Engineering* 136 (2017): 54-66. https://doi.org/10.1016/j.oceaneng.2017.03.012
[8] Bensow, Rickard E. "Simulation of unsteady propeller blade loads using OpenFOAM." In *16th Numerical Towing Tank Symposium, NuTTS*, vol. 13. 2013.
[9] Bahatmaka, Aldias, and Dong-Joon Kim. "The influence of meshing strategies on the propeller simulation by CFD." *Journal of Advanced Research in Ocean Engineering* 4, no. 2 (2018): 78-85.
[10] Bahatmaka, Aldias, Dong-Joon Kim, Deddy Chrismianto, Nguyen Hai, and Aditya Rio Prabowo. "Optimization of thrust propeller design for an ROV (Remotely Operated Vehicle) consideration by Genetic Algorithms." In *MATEC Web of Conferences*, vol. 138, p. 07003. EDP Sciences, 2017https://doi.org/10.1051/matecconf/201713807003
[11] Gaggero, Stefano, Diego Villa, Giorgio Tani, Michele Viviani, and Daniele Bertetta. "Design of ducted propeller nozzles through a RANSE-based optimization approach." *Ocean Engineering* 145 (2017): 444-463. https://doi.org/10.1016/j.oceaneng.2017.09.037
[12] Zondervan, Gert-Jan, Martin Hoekstra, and Jan Holtrop. "Flow analysis, design and testing of ducted propellers." In *SNAME Propeller and Shafting Symposium*, p. D011S001R006. SNAME, 2006. https://doi.org/10.5957/PSS-2006-06
[13] Adietya, Berlian Arswendo, Husein Syahab, Mochammad Nasir, Wasis Dwi Aryawan, and I. Ketut Aria Pria Utama. "Numerical Analysis into the Improvement Performance of Ducted Propeller by using Fins: Case Studies on Types B4-70 and Ka4-70." *CFD Letters* 16, no. 10 (2024): 12-42. https://doi.org/10.37934/cfdl.16.10.1242
[14] Kuiper, G. "New developments and propeller design." *Journal of Hydrodynamics, Series B* 22, no. 5 (2010): 7-16. https://doi.org/10.1016/S1001-6058(09)60161-X
[15] Haimov, Henri, Jorge Vicario, and Javier Del Corral. "RANSE code application for ducted and endplate propellers in open water." In *Proceedings of the Second International Symposium on Marine Propulsors*, pp. 1-9. 2011.
[16] Paik, Kwang-Jun. "Numerical study on the hydrodynamic characteristics of a propeller operating beneath a free surface." *International Journal of Naval Architecture and Ocean Engineering* 9, no. 6 (2017): 655-667. https://doi.org/10.1016/j.ijnaoe.2017.02.006
[17] Parra, Carlos. "Numerical investigation of the hydrodynamic performances of marine propeller." *Master's thesis, University of Galati, Gdynia* (2013).
[18] Wang, Lian-Zhou, Chun-Yu Guo, Yu-Min Su, and Tie-Cheng Wu. "A numerical study on the correlation between the evolution of propeller trailing vortex wake and skew of propellers." *International Journal of Naval Architecture and Ocean Engineering* 10, no. 2 (2018): 212-224. https://doi.org/10.1016/j.ijnaoe.2017.07.001
[19] Yao, Jianxi. "Investigation on hydrodynamic performance of a marine propeller in oblique flow by RANS computations." *International Journal of Naval Architecture and Ocean Engineering* 7, no. 1 (2015): 56-69. https://doi.org/10.1515/ijnaoe-2015-0005
[20] Willemsen, J. C. "Improving Potential Flow Predictions for Ducted Propellers." *University Twente* (2013).
[21] Yeo, Dong Jin. "KCS-Propeller-Openwater-Data-KRISO." *Journal of the Society of Naval Architects of Korea* 1, no. 5 (1997): 296-305.
[22] Fitriadhy, Ahmad, N. Amira Adam, and C. J. Quah. "Computational prediction of a propeller performance in open water condition." *Sinergi* 24, no. 2 (2020): 163-170. https://doi.org/10.22441/sinergi.2020.2.010
[23] Horvat, Marko. "A study of rotor-stator interaction models for numerical performance prediction of centrifugal pumps." *Master's thesis, Faculty of Mechanical Engineering and Naval Architecture, University of Zagreb* (2016).
[24] Subhas, S., V. F. Saji, S. Ramakrishna, and H. N. Das. "CFD analysis of a propeller flow and cavitation." *International Journal of Computer Applications* 55, no. 16 (2012). https://doi.org/10.5120/8841-3125
[25] Permadi, Niki Veranda Agil, and Erik Sugianto. "CFD Simulation Model for Optimum Design of B-Series Propeller using Multiple Reference Frame (MRF)." *CFD Letters* 14, no. 11 (2022): 22-39. https://doi.org/10.37934/cfdl.14.11.2239