*CFD Letters* 15, Issue 4 (2023) 17-30

CFD Letters

Journal homepage:
https://semarakilmu.com.my/journals/index.php/CFD_Letters/index
ISSN: 2180-1363


# Investigation of the Effects of the Pre-Duct in a Ship on Propeller–Hull Interactions Using the CFD Method

I Made Ariana[1,*], Riyan Bagus Prihandanu[1], Dhimas Widhi Handani[1], AAB Dinariyana[1]

[1] Department of Marine Engineering, Faculty of Marine Technology, Institut Teknologi Sepuluh Nopember, 60111 Surabaya, Indonesia

**ARTICLE INFO**

***Article history:***
Received 3 September 2022
Received in revised form 5 October 2022
Accepted 4 November 2022
Available online 1 April 2023

***Keywords:***
Energy Saving Device; Pre-Duct; Propulsion System

**ABSTRACT**

The power requirement of a ship propulsion system is directly proportional to the fuel consumption and the emissions released. By reducing the engine power, the fuel consumption and emissions can be reduced. One of the energy saving devices (ESDs) that is positioned in a region between the stern hull and propeller is the pre-duct. The ESD can improve the propulsive coefficient of the propulsion system. This research explains the effect of a circular pre-duct on the hull-propeller interaction. A numerical simulation uses the Japan bulk carrier (JBC) standard model for the hull and propeller. A circular pre-duct was applied to the ship with different diameters, stands, and lengths of the chord. The simulation has already been validated with the result of the resistance and self-propulsion test in the towing tank. The results show that the diameter of the pre-duct affects the water flow to the propeller. The model with 1Dp can make the positive value of the propulsive coefficient be about 1.72%. The size of the foil chord of the pre-duct can improve the performance until 2.88% at 1D 2S NS model. The stand of the pre-duct has a bad effect on the propulsive coefficient. The enlarged shape of the foil pre-duct can increase the water flow on the suction side. Also, making the diameter larger than the propeller diameter and eliminating the stand on the pre-duct make the incoming flow has no resistance or damages the rotary flow before the propeller. So that, the rampant flow to the propeller becomes larger and there are no significant obstacles until the propulsive coefficient value increases significantly.

## 1. Introduction

An efficient engine and propulsion system is needed to move a ship. The engine power requirement is directly proportional to the emissions released; however, the hull design also affects the engine power requirement which Setiawan *et al.,* [1] already stated the interaction between ship hull to ship resistance. Reducing the engine power can reduce gas emissions. One of the ways to reduce the torque required by the propeller is by reducing the engine power. When the required torque decreases, the propulsive coefficient value will decrease, and the engine power requirements will also decrease without reducing the ship and rotational speed.

* *Corresponding author.*
*E-mail address: ariana@its.ac.id (I Made Ariana)*

https://doi.org/10.37934/cfdl.15.4.1730

According to Mysa *et al.,* [2], over the past decades Energy Saving Devices (ESDs) like propeller ducts, pre-nozzles, and pre-swirl stators have been investigated to reduce energy consumption for in-operation and new ship designs, so the ship can gain more benefit in a business point of view. Adietya *et al.,* [3] found that the ducted propeller design will create better fluid flow in a certain propeller design. However, the geometric design of the duct may variously affect the fluid flow. This is depicted in the research by previous study [4], in which the particle dispersion of cavity flow was analyzed. Mewis and Hollenbach [5] reported that there is a 3-9% potential savings of propulsion efficiency from various types of ESDs which varies depending on the type of different measures. Over the past years, the application of those devices has been limited, since there is a lack of confidence in measuring the real efficiency and benefits in real full-scale ships. Shin *et al.,* [6] reported that the utilization of the advances in computational fluids dynamics (CFD) have provided an alternative approach for clear interpretation from model scale tests to better understand the uncertainties in the prediction of the ESDs efficiency in full-scale ship operations.

Some research has been carried out using the CFD method to better understand the flow of fluids, such as coal-air mixing flow [7], flame characteristics [8], combustion of engine [9], air flow in an airfoil [10], air flow on vortex added airfoil [11], underwater noise [12], ship resistance [13] and analysis of hydrokinetic turbine [14]. Detailed flows obtained from CFD will provide a good platform for further understanding the effects of ESDs as well as mechanisms to enhance the propeller efficiency. This remains the future work for the further development and application of this model towards the design of more effective ESDs as stated by Mysa *et al.,* [2]. However, Schulling & Van Terwisga [15] already compared model scale and full scale by using the CFD method which the total power savings was increased by 0.6%. This means that the change in the scale of the ship does not have a significant effect on the results of the simulation that has been carried out.

From the research of Nowruzi & Najafi [16] and Prihandanu *et al.,* [17], the circular pre-duct has no significant effect on the propulsive coefficient (PC) value. The flow that enters the propeller is important to increase the value of the PC. A greater flow to the propeller decreases the torque and increases the efficiency. However, the concept and properties of the pre-duct are not the same as the ducting concept, which is commonly applied to the Kaplan propeller, as stated by van Terwisga [18]. The pre-duct prioritizes the analysis on the hull-propeller interaction. So, the main focus is the flow before the propeller and after the stern of the hull.

One of the components that affects the pre-duct performance, according to A Munazid *et al.,* [19], is the angle of attack, and changing the angle of attack on the pre-duct can increase the value of the PC. By increasing the radius and the angle of attack of the duct's inner fins, the axial mean wake fraction first decreased and then increased, while the tangential mean wake fraction increased gradually, as stated by Chang *et al.,* [20]. However, pre-duct variations do not have a positive impact compared to those without a pre-duct, and the installation of a pre-duct does not have a positive impact on all types of ships, as claimed by Nowruzi & Najafi [16]. In another way, the results from simulation-based design optimization (SBDO) show sensible improvements in the overall propulsive efficiency when the design of the pre-duct is tailored to the hull wake shape of actual ship, as Furcas *et al.,* [21] reported. So, it can be concluded that the optimal pre-duct design cannot be determined with certainly.

Currently, there are not enough findings that show a clear effect of the pre-duct on the performance of hull-propeller interactions. Mostly, the design of the pre-duct that appears in the literature has a diameter less than the diameter of the propeller. This research proposed a CFD simulation of a self-propulsion test that shows an interaction between the hull and propeller with some variation of the pre-duct diameter. The pre-ducts that were designed and simulated in this paper have a trailing edge diameter greater than the propeller diameter. In addition, the pre-duct stand position was removed and the scale of the chord foil length of the pre-duct was changed. Increasing the size of the pre-duct diameter can smooth out the flow into the propeller. The stand on the pre-duct can affect the flow into the propeller, especially affecting the fluid rotation before it enters the propeller. However, on the variation size of the foil chord length, the impact enlarges and forces the flow into the propeller from all parts of the stern of the ship, so the advance velocity value for the propeller becomes better.

## 2. Methodology

### *2.1 Boundary Condition*

As shown in Figure 1, the unit domain size or boundary simulation refers to the Length between Perpendicular (LWL) of the ship, where the boundary inflow has a distance of 2.5 LWL. As for the vertical size, it has a value of 2 LWL with a size of 1.3 LWL on the top side and 1.7 LWL on the bottom. The right and left sides have a size of 2 LWL. The backside, which is a place to see the flow of the waves and the propulsion system, is made with a size that is 4 LWL longer. The center point or measurement reference is placed on the ship's AP. The surface of the hull and propeller was set as no-slip, and the simulation was set to the entire body.

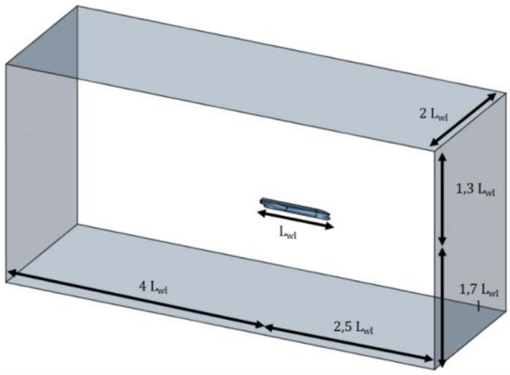



**Fig. 1.** Schematic of the computational domain and boundary condition

### *2.2 Geometry*

This simulation used the Japan bulk carrier ship used at the Tokyo Conference 2015 made by the National Maritime Research Institute (NMRI), Yokohama National University, and Ship Building Research Center of Japan (SRC). The main particulars are shown in Table 1 and Figure 2 shows the body plan. For the simulation, a 1:40 scale model is used according to the validation data at the Tokyo Conference 2015.

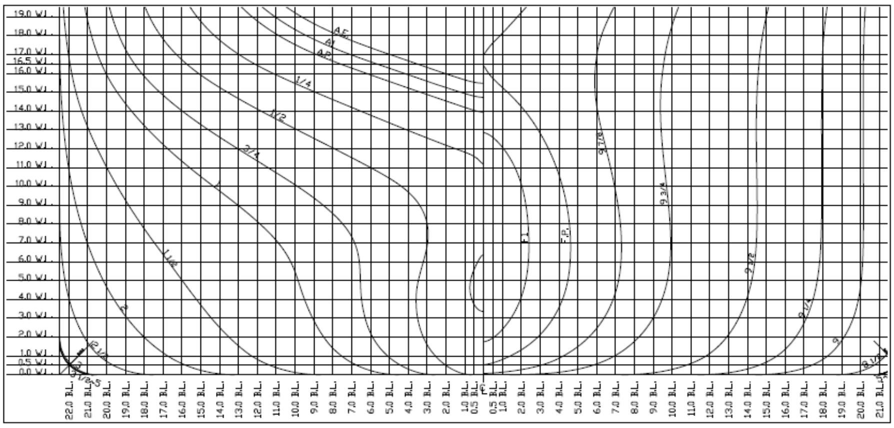


**Fig. 2.** Body plan of the Japan bulk carrier

**Table 1**
General Parameters of the Japan Bulk Carrier at Full Scale

| Main particular | Parameter (Unit) | Full Scale |
|---|---|---|
| Length between perpendiculars | LPP (m) | 280 |
| Length of waterline | LWL (m) | 285 |
| Maximum beam of waterline | BWL (m) | 45 |
| Depth | D (m) | 25 |
| Draft | T (m) | 16.5 |
| Block coefficient | Cb | 0.858 |
| Displacement volume | $\nabla$ ($m^3$) | 178369.9 |
| Wetted surface area | S ($m^2$) | 19556.1 |
| Design speed | V | 14.5 |
| Scale | λ | 40 |

Meanwhile, the MP678 propeller model was used based on the AU-Series model, as shown in Table 2. The propeller geometry is based on the Tokyo Conference 2015. The boss ratio is 0.18Dp. The value of the pitch ratio is 0.75 or 155.25 mm, with a 0.5 expanded area ratio (EAR). For the Au-Series, 5 degrees of rake is used. This propeller used a 5-blade number with a clockwise rotation.

**Table 2**
General Parameters of the Propeller MP678 at Model Scale

| Main particular | Parameter (Unit) | Model Scale |
|---|---|---|
| Diameter | Dp (mm) | 203 |
| Boss ratio | - | 0.18 |
| Pitch | P (mm) | 152.25 |
| Expanded area ratio | - | 0.5 |
| Max. blade width ratio | - | 0.2262 |
| Blade thickness ratio | - | 0.05 |
| Angle of rake | Deg (°) | 5 |
| Number of blades | - | 5 |
| Blade section | - | AU-Series |
| Direction for rotation | - | Clockwise |

The ducting form will use the NACA4420 model with a 20-degree angle of attack, as shown in Figure 3 and Table 3. The distance with the propeller is 14.72 mm. The chord length of the profile foil pre-duct is 60.9 at the original size or 0.3Dp, and the chord length of the stand pre-duct is 30.5

mm with 4 mm for the width. The height of the stand follows the pre-duct size, and the position is 12 mm from the trailing edge of the pre-duct.

**Table 3**
General Parameters of the Propeller MP678 at Model Scale

| Main particular | Parameter (Unit) | Model Scale |
|---|---|---|
| Chord length | Diameter of propeller (Dp) | 0.3 |
| Opening angle | Deg (°) | 20 |
| Foil section | - | NACA4420 |

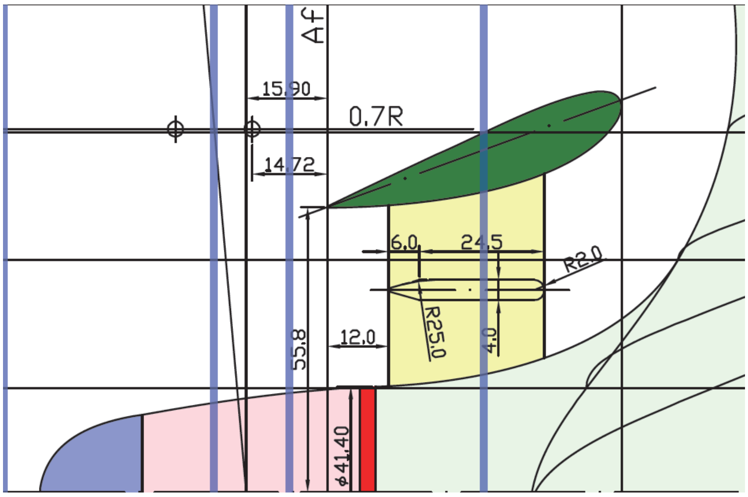
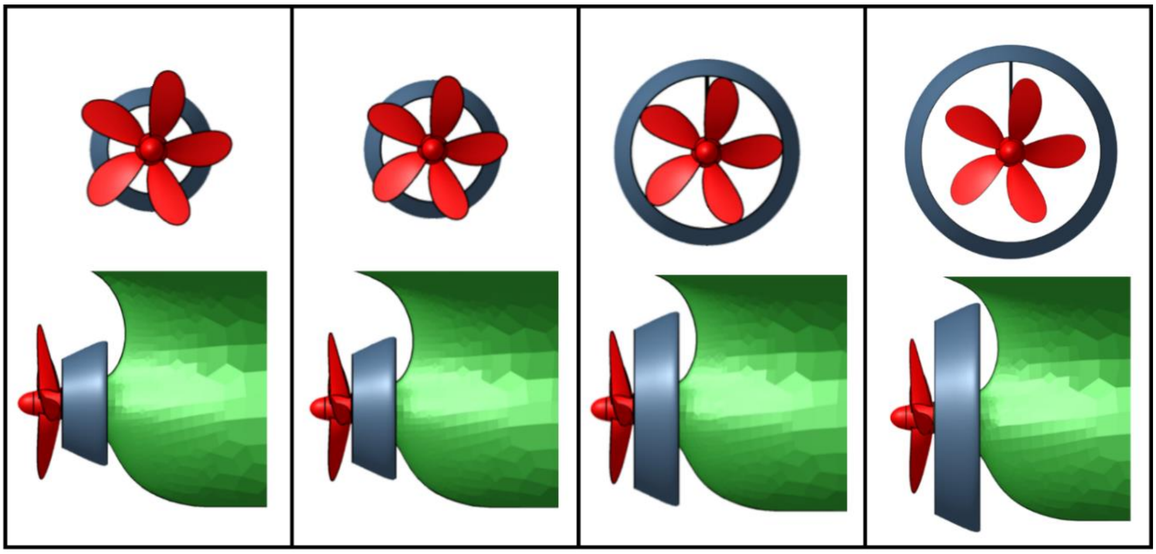


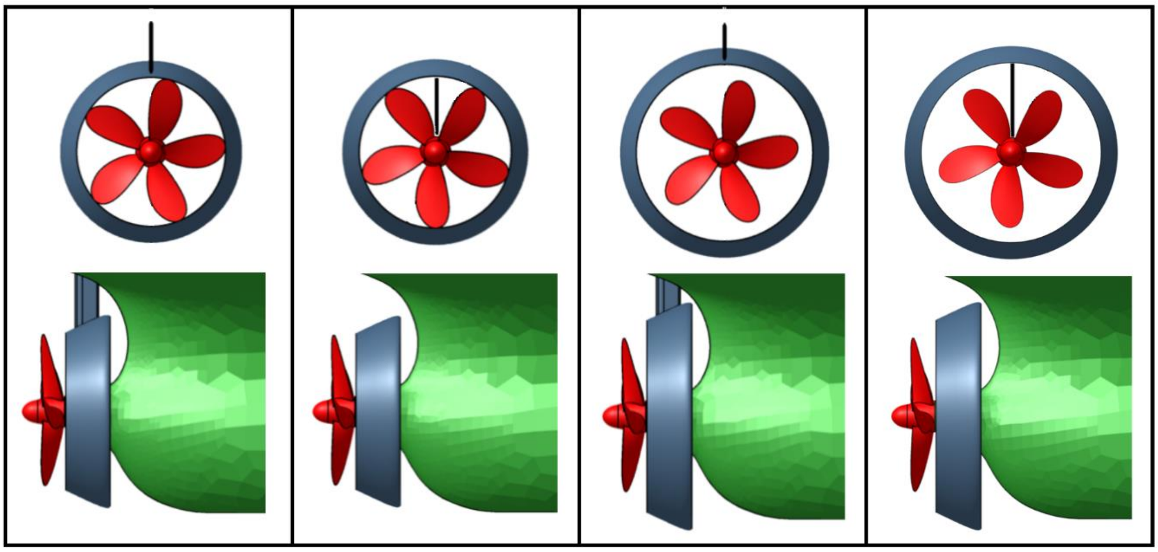
**Fig. 3.** Pre-duct design of the Japan bulk carrier

Simulations are carried out with several types of variations of the pre-duct model. In the first scheme, the diameter of the pre-ducts is varied from 0.5Dp to 1.2Dp, where Dp is the diameter of the propeller. This simulation is carried out with stand inner the Pre-Duct side. The second scheme is done without and with a stand on the pre-duct. The purpose is to clearly see the effect of the pre-duct stand on the flow before the propeller. The third scheme is to change the chord length of the profiles foil pre-duct with the same angle of attack and foil profile so that there will be 2 chord lengths of the pre-duct, namely, 60.9 mm and 121.8 mm. The simulation model variations can be seen in Figures 4–6. The name of the variation is determined from S for Size (chord length), WS for with stand, and NS for no stand or without stand, as explained in Table 4.



(a) (b) (c) (d)

**Fig. 4.** Diameter of the pre-duct variation: (a) 0.5D 1S WS; (b) 0.7D 1S WS; (c) 1D 1S WS; and (c) 1.2D 1S WS



(a) (b) (c) (d)

**Fig. 5.** Stand variation: (a) 1D 1S NS; (b) 1D 1S WS; (c) 1.2D 1S NS; and (d) 1.2D 1S WS

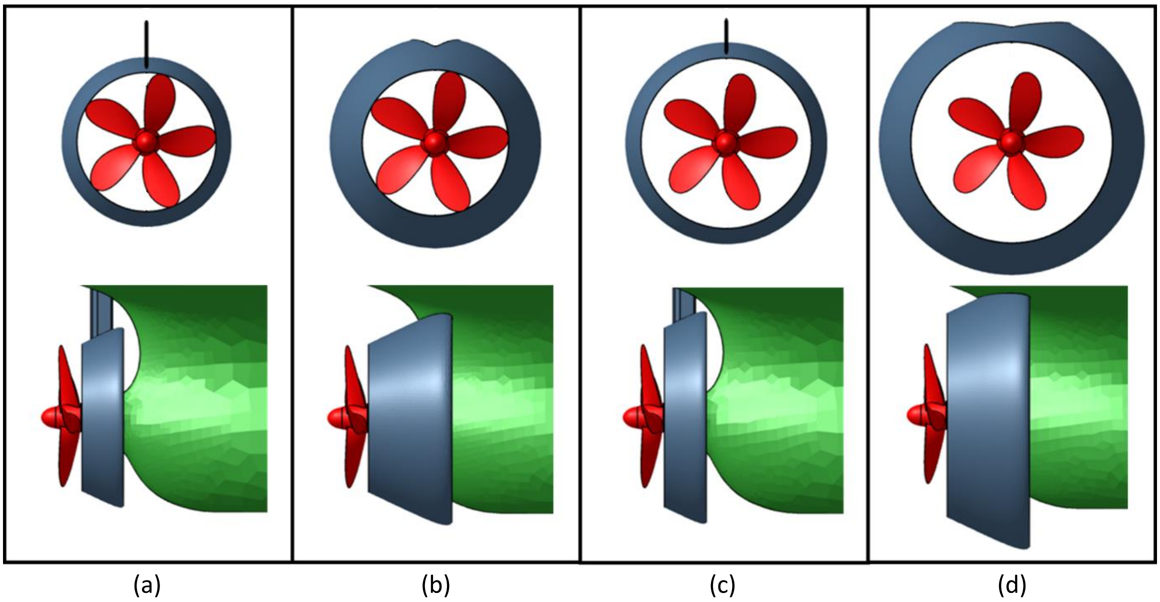



**Fig. 6.** Length of the chord variation: (a) 1D 1S NS; (b) 1D 2S NS; (c) 1.2D 1S NS; and (d) 1.2D 2S NS

**Table 4**
General Parameters of the Propeller MP678 at Model Scale

| Diameter of Pre-Duct | | Length of Chord | | Stand | |
|---|---|---|---|---|---|
| 0.5D | 50% of Dp | 1S | 60.9 mm | WS | With Stand |
| 0.7D | 70% of Dp | 2S | 121.8 mm | NS | Without Stand |
| 1D | 100% of Dp | | | | |
| 1.2D | 120% of Dp | | | | |

### *2.3 Validation*

The measurement of the model validation or the comparison between Experimental Fluid Dynamics (EFD) and CFD is carried out in two stages. The first is the independent grid stage, which is measuring the mesh value or the number of cells/grids that is in accordance with the simulation to get a small error ratio. The resistance test data derived from the resistance coefficient is used as the comparative data. As shown in Figures 7 and 8, the simulation error or gap compared with EFD is 1.9% and stable in 2.5M–4M grids or cells. Then, the range of the mesh simulation is from 2.5M–4M grids or cells.

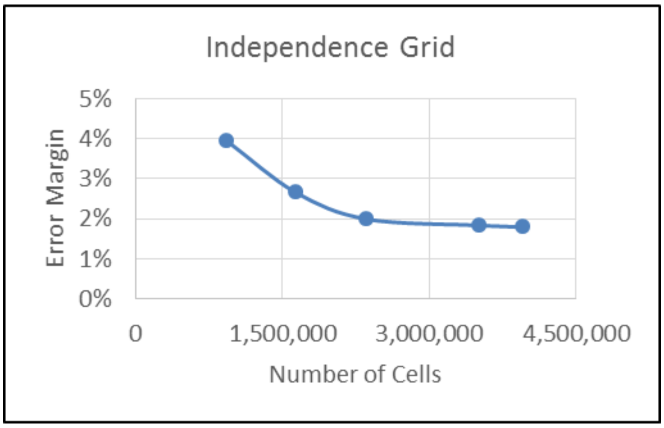



**Fig. 7.** Independent grid

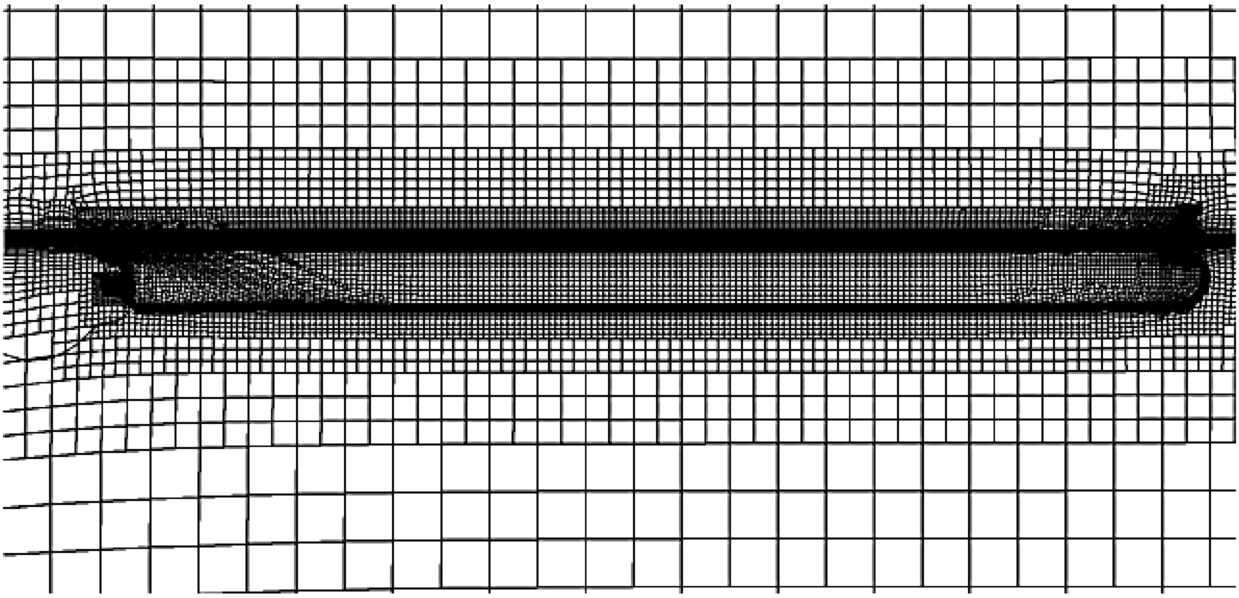


**Fig. 8.** Grid or cell distribution

In the next stage, a comparison was made on the values of the relative rotative efficiency (ηR), open water efficiency (ηO), hull efficiency (ηH), and propulsive coefficient (PC). The validation was carried out at the propeller rotational speed according to the data available at the 2015 Tokyo Conference, which is 7.8 rps. The computational fluid dynamic (CFD) was compared with the experiment fluid dynamic (EFD). The dual domain and actuator disk simulation was used for this validation. As shown in Table 5, the error from all efficiencies was under 5%. Then, the meshing of the hull and propeller is acceptable for use in the simulation with different pre-duct models, as claimed by Abar & Utama [22], Mysa *et al.,* [2], and Suastika *et al.,* [23].

**Table 5**
Validation of EFD and CFD

| Name | CT | KT | KQ | RT-T | 1-t | 1-w | ηR | ηO |
|---|---|---|---|---|---|---|---|---|
| EFD | 0.004811 | 0.217 | 0.0279 | 18.10 | 0.812 | 0.552 | 1.015 | 0.501 |
| CFD | 0.004959 | 0.220 | 0.0286 | 18.28 | 0.779 | 0.540 | 0.995 | 0.494 |
| Err. % | 3.1% | 1.4% | 2.5% | 1.0% | -4.0% | -2.1% | -2.0% | -1.4% |

## 3. Results and Discussion

### *3.1 Flow Analysis*

Flow analysis is based on the simulation data that previously have been done. The fluids flow caused by the changes in the diameter and length of the chord foil is taken based on two things, namely, the intersection of the flow on the Z axis at the centre of the propeller and the wake field or advance velocity distribution. Meanwhile, the fluids flow for model variations without and with a stand is taken for an intersection in the middle of the pre-duct stand to see the flow on the pre-duct stand, as shown in Figures 9–11.

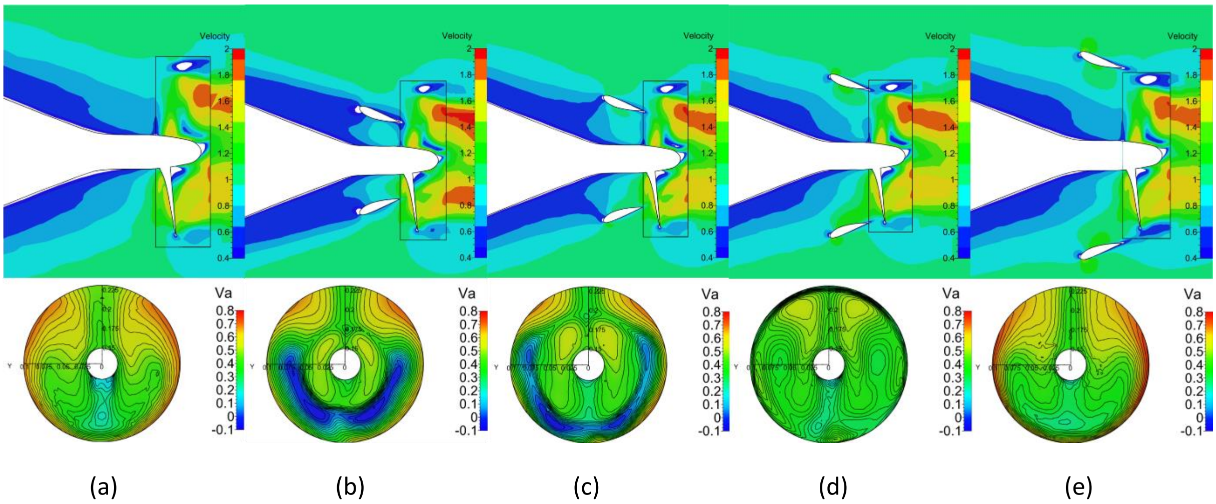



**Fig. 9.** Velocity (top) and wake field (below) stern ship (a) without pre-duct; (b) 0.5D 1S WS; (c) 0.7D 1S WS; (d) 1D 1S WS; and (e) 1.2D 1S WS

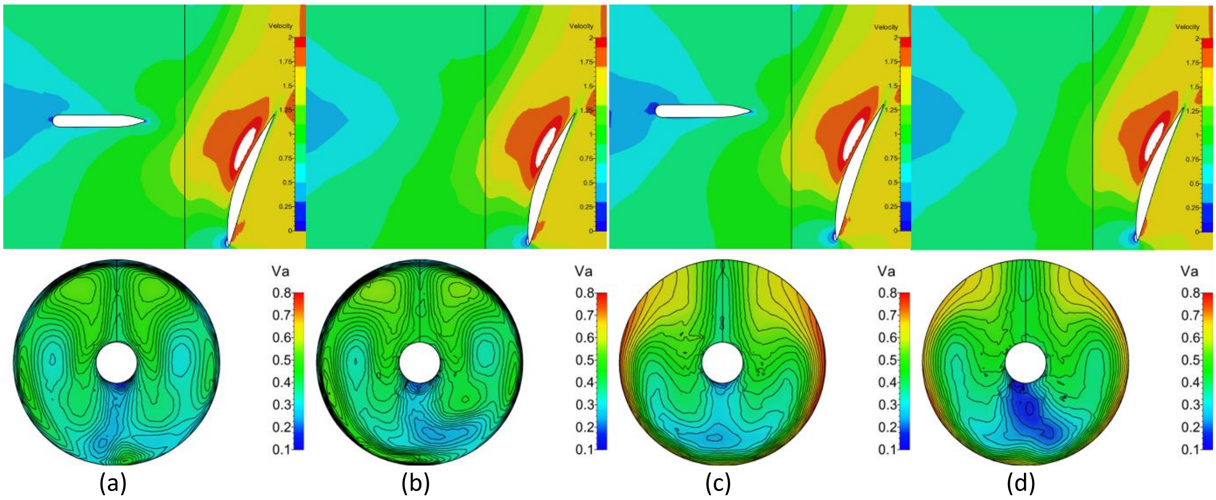



**Fig. 10.** Velocity (top) and wake field (below) stern ship (a) 1D 1S WS; (b) 1D 1S NS; (c) 1.2D 1S WS; and (d) 1.2D 1S NS

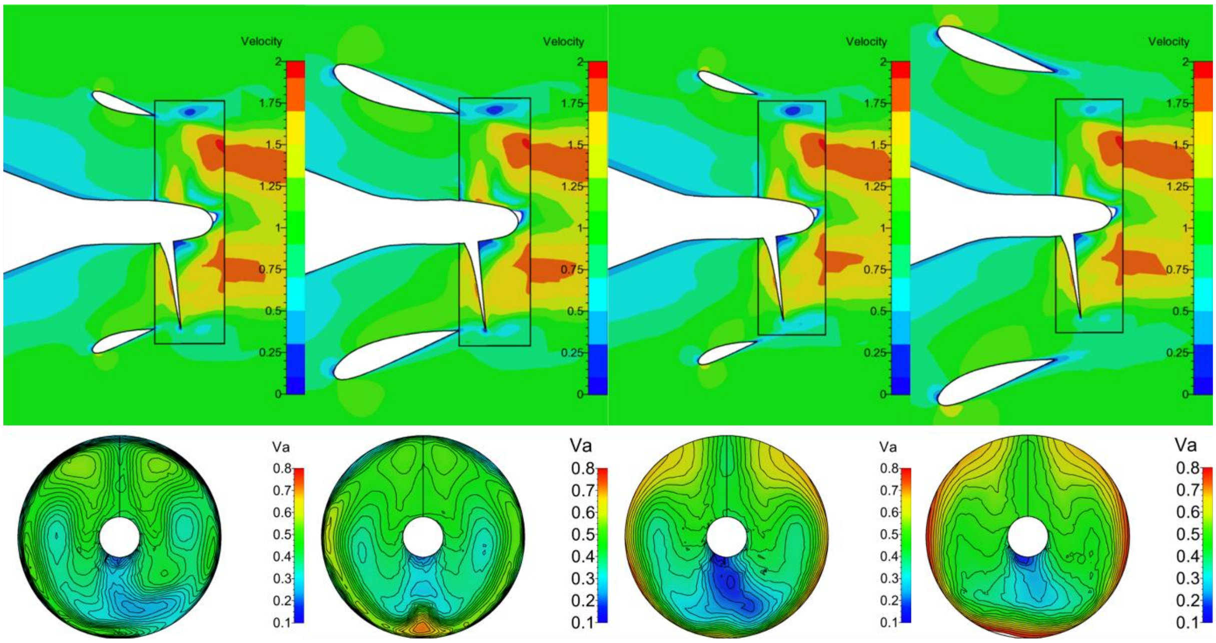



**Fig. 11.** Velocity (top) and wake field (below) stern ship (a) 1D 1S NS; (b) 1D 2S NS; (c) 1.2D 1S NS; and (d) 1.2D 2S NS

The velocity distribution and wake field of the variation of the ducted diameter when viewed from the picture of the centre intersection of the propeller are shown in Figure 9. It can be concluded that the flow at the diameter of 0.5Dp and 0.7Dp before entering the pre-duct is lower than the 1Dp or 1.2Dp. The pre-duct position, which is close to the ship hull, will reduce the ship speed as a result of the stagnation pressure at the tip of the pre-duct. At that position, the flow velocity for the vessel without the pre-duct is higher than the flow velocity for all variations of the pre-duct diameter. Although there is a decrease in the flow velocity at the inlet of the pre-duct, the distribution of the velocity of the flow out of the pre-duct and towards the propeller becomes more homogeneous. The installation of the pre-duct results in a more homogeneous wake field at the top of the propeller axis, with an average speed distribution that is lower than the speed distribution without the pre-duct. The pre-duct with a diameter of 1D shows the most homogeneous wake field; it can be understood that the flow that will go to all parts of the propeller has been streamlined.

In the variation with and without a stand, it can be seen in a subtle way that the stand has an effect in inhibiting the slow flow to the propeller. This affects the flow particularly at the top of the propeller, although it is insignificant, as shown in Figure 10. However, in the variation of the chord foil length, the flow that enters the propeller for 2S is higher than 1S, especially on the tip of the propeller, as shown in Figure 11. For ducts with long chords, the flow velocity in the hull stern is higher than for short chords or without pre-ducts. A decrease in the velocity of the fluid at the hull stern can cause an increase in the thrust deduction fraction. On the other hand, increasing the length of the chord and/or the diameter of the duct causes the ship's resistance to increase. So, propeller performance can be increased by improving the fluid flow while we need to concern to the ship resistance. Other factors that can increase the ship resistance are water depth since the shallow water increases ship resistance [24].

### *3.2 Propulsion Performance*

The propulsive coefficient (PC) formula, as shown in Eq. 1, is used as a comparison for each model variation. The formula can be simplified as Eq. 5 by Carlton [25].

$$PC = \eta H \times \eta R \times \eta O \tag{1}$$

$$\eta H = \frac{1-t}{1-w} \tag{2}$$

$$\eta R = \frac{KQ_o}{KQ_b} \tag{3}$$

$$\eta O = \frac{KT}{KQo} \times \frac{J}{2\pi} \tag{4}$$

$$PC = \frac{R_T V_s \eta R}{2\pi K Q_o \rho n^3 D^5} \tag{5}$$

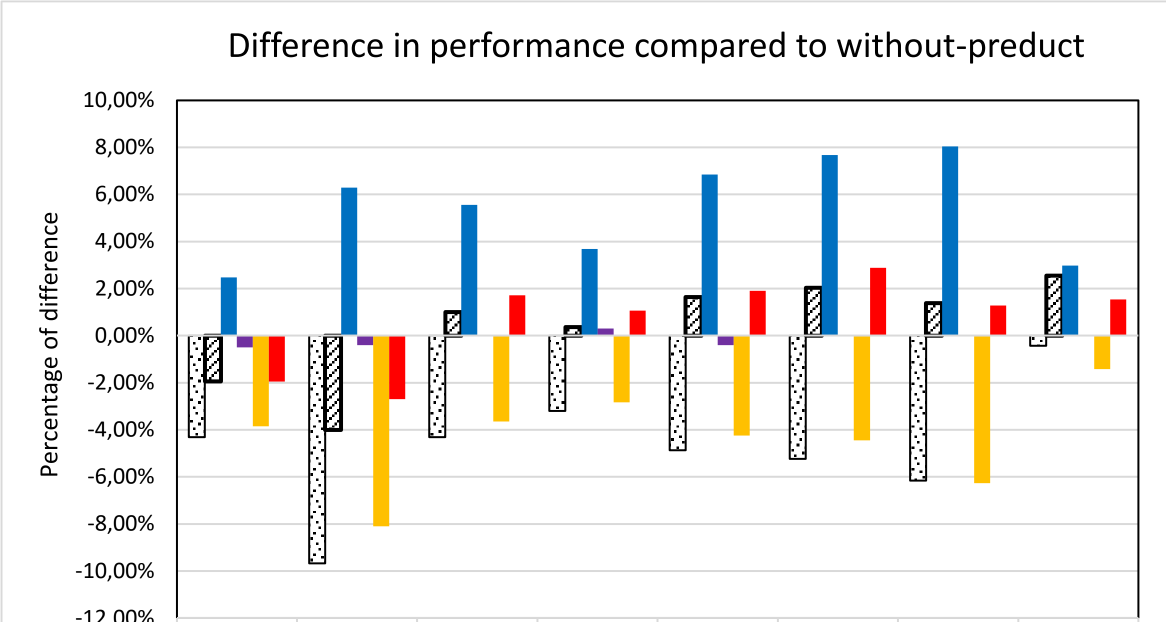
From the simulation results carried out on several schemes, variations of the pre-duct diameter affect the propulsive coefficient. A larger diameter results in a smoother incoming flow. It is proven that in the 1.2D diameter, the open water efficiency value is higher than for the other diameters. This is also supported by the effect of the foil, which accelerates the inflow of the propeller so that the Va value will increase, and then the open water efficiency increases without reducing the hull efficiency. In general, the relative rotative efficiency has decreased but not significantly. However, in the hull efficiency, the largest increase occurred in the smaller pre-duct diameter or below 1D because the value (1-t) is smaller than the value (1-w), which means that the value of resistance increases and the value of Va decreases, which results in a significant decrease in the value of the open water efficiency.

**Table 6**
Effect of the Pre-Duct Diameter on the Propulsive Performance

| Name | 1-w | 1-t | ηH | ηR | ηO | PC |
|---|---|---|---|---|---|---|
| Without Pre-Duct | 0.540 | 0.779 | 1.442 | 0.995 | 0.494 | 0.709 |
| 1S0.5DWS | 0.517 | 0.764 | 1.478 | 0.990 | 0.475 | 0.695 |
| 1S0.7DWS | 0.448 | 0.748 | 1.533 | 0.991 | 0.454 | 0.690 |
| 1S1DWS | 0.517 | 0.787 | 1.522 | 0.995 | 0.476 | 0.721 |
| 1S1.2DWS | 0.523 | 0.782 | 1.495 | 0.998 | 0.480 | 0.716 |

**Table 7**
Effect of the Foil Chord Length and Without Stand on the Propulsive Performance

| Name | 1-w | 1-t | ηH | ηR | ηO | PC |
|---|---|---|---|---|---|---|
| 1S1DNS | 0.514 | 0.792 | 1.541 | 0.991 | 0.473 | 0.722 |
| 2S1DNS | 0.512 | 0.795 | 1.553 | 0.995 | 0.472 | 0.729 |
| 1S1.2DNS | 0.507 | 0.790 | 1.558 | 0.995 | 0.463 | 0.718 |
| 2S1.2DNS | 0.538 | 0.799 | 1.485 | 0.995 | 0.487 | 0.720 |




| | 1S0,5DWS | 1S0,7DWS | 1S1DWS | 1S1,2DWS | 1S1DNS | 2S1DNS | 1S1,2DNS | 2S1,2DNS |
|---|---|---|---|---|---|---|---|---|
| 1-w | -4,31% | -9,68% | -4,31% | -3,20% | -4,87% | -5,24% | -6,16% | -0,43% |
| 1-t | -1,95% | -4,00% | 1,00% | 0,36% | 1,64% | 2,03% | 1,39% | 2,54% |
| Etta H | 2,47% | 6,28% | 5,55% | 3,68% | 6,84% | 7,67% | 8,05% | 2,98% |
| Etta R | -0,50% | -0,40% | 0,00% | 0,30% | -0,40% | 0,00% | 0,00% | 0,00% |
| Etta O | -3,85% | -8,10% | -3,64% | -2,83% | -4,25% | -4,45% | -6,28% | -1,42% |
| PC | -1,96% | -2,70% | 1,72% | 1,06% | 1,90% | 2,88% | 1,28% | 1,53% |

**Fig. 12.** Percentage of the difference of the propulsive performance compared to without pre-duct

It is known that the addition of a stand on the pre-duct in front of the propeller affects the propeller performance by 1%. By removing the stand on the pre-duct, the flow that enters the propeller will not have any obstacles so that the rotating effect of the fluid before the propeller can be formed completely.

Increase in the size of the pre-duct foil profile means there is e enlargement to 2 times from its original. This enlargement turned out to have a very positive impact, especially on the hull efficiency and propeller/open water efficiency. By increasing the size of the foil, the effect of the foil, which accelerates the flow into the propeller, will be bigger and more obvious. The simulation data show that the hull efficiency increases by 5% and the open water efficiency increases by 6%. From the results of all simulations, it can be concluded that a suitable pre-duct design is 1.2 times the diameter of the propeller without any obstructions in front of the propeller. This is done to keep the propeller fluid from being affected by anything. Meanwhile, the size of the foil profile can be enlarged to get the effect of accelerating the flow into the propeller, but this must also consider the pre-duct material needed and the shipload or center of gravity, which will be heavier on the stern which causes trim by stern.

## 4. Conclusions

From all the simulations that have been carried out, it can be concluded that the hull and open water efficiency has opposite properties or effects. The addition of a pre-duct tends to increase the efficiency of the hull and decrease the efficiency of open water. However, in some types of pre-ducts, the open water efficiency can be increased significantly to produce a better propulsive coefficient. This is because the flow into the propeller must be smooth as in the open water test conditions.

The enlarged shape of the foil pre-duct can increase the water flow on the suction side. Also, making the diameter larger than the propeller diameter and eliminating the stand on the pre-duct make the incoming flow has no resistance or changes the rotary flow into more unregular before the propeller. So that the rampant flow to the propeller becomes larger and there are no significant obstacles until the propulsive coefficient value increases significantly.

**Acknowledgement**
The authors gratefully acknowledge financial support that is provided by The Institut Teknologi Sepuluh Nopember (ITS) Surabaya through funding of the research grant.

**References**

[1] Setiawan, Wira, Amalia Ika Wulandari, Aditya Miftahul Huda, and Syerly Klara. "Comparative Study of Ship Resistance and Fuel Consumption between Axe Bow and Moor Deep Ram Bow using CFD Method." *CFD Letters* 14, no. 8 (2022): 71-80. https://doi.org/10.37934/cfdl.14.8.7180

[2] Mysa, Ravi Chaithanya, Le Quang Tuyen, Ma Shengwei, and Vinh-Tan Nguyen. "Numerical investigation of energy saving device effects on ships with propeller-hull interactions." In *International Conference on Offshore Mechanics and Arctic Engineering*, vol. 51272, p. V07BT06A044. American Society of Mechanical Engineers, 2018. https://doi.org/10.1115/OMAE2018-78600

[3] Adietya, Berlian Arswendo, I. Ketut Aria Pria Utama, and Wasis Dwi Aryawan. "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

[4] Sahak, Ahmad Sofianuddin A., Nor Azwadi Che Sidik, and Siti Nurul Akmal Yusof. "A Brief Review of Particle Dispersion of Cavity Flow." *Journal of Advanced Research in Applied Sciences and Engineering Technology* 20, no. 1 (2020): 27-41. https://doi.org/10.37934/araset.20.1.2741

[5] Mewis, Friedrich, and Uwe Hollenbach. "Special measures for improving propulsive efficiency." *HSVA NewsWave* 1 (2006).

[6] Shin, Hyun-Joon, Jong-Seung Lee, Kang-Hoon Lee, Myung-Ryun Han, Eui-Beom Hur, and Sung-Chul Shin. "Numerical and experimental investigation of conventional and un-conventional preswirl duct for VLCC." *International Journal of Naval Architecture and Ocean Engineering* 5, no. 3 (2013): 414-430. https://doi.org/10.2478/IJNAOE-2013-0143

[7] Triwibowo, Bayu, Heni Wahyu Widayanti, and Miftakhul Indra Rukmanasari. "Prediction of Erotion Rate in Two Elbows for Coal-Air Flow Based on Computational Fluid Dynamics Simulation." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 97, no. 2 (2022): 115-125. https://doi.org/10.37934/arfmts.97.2.115125

[8] Jankaew, Nilnet, Mohd Faizal Mohideen Batcha, Arkom Palamanit, Maizirwan Mel, and Makatar Wae-hayee. "The Effects of Pressure and Temperature on Flame Characteristics of Crude Palm Oil Combustion." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 98, no. 2 (2022): 165-174. https://doi.org/10.37934/arfmts.98.2.165174

[9] Ariani, Betty, I. Made Ariana, and Aguk Zuhdi M. Fathallah. "Numerical Study of Emissions on DDF Engine with 20% CNG with Variation on Compression Ratio." In *Recent Advances in Renewable Energy Systems*, pp. 289-294. Springer, Singapore, 2022. https://doi.org/10.1007/978-981-19-1581-9_32

[10] Elsafty, Kareem Ahmed Ismail, Atif Mohamed Emad Elsherif, Ahmed Abdelhamid Ibrahim, Omar Sherif Mohamed, and Ahmed Mohamed Reda Elbaz. "Investigating the Performance of a Novel Multi-Element Airfoil Concept Using Numerical Analysis." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 97, no. 2 (2022): 126-145. https://doi.org/10.37934/arfmts.97.2.126145

[11] Niknahad, Ali. "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.

[12] Purwana, Agung, I. Made Ariana, and Wisnu Wardhana. "Numerical study on the cavitation noise of marine skew propellers." *Journal of Naval Architecture and Marine Engineering* 18, no. 2 (2021): 97-107. https://doi.org/10.3329/jname.v18i2.38099

[13] Wawan Kusdiana, Semin, I Made Ariana. "Analysis of Ship Resistance Based on Horizontal Placement of Fin Stabilizer Using CFD Software." *International Review on Modelling and Simulations* (I.RE.MO.S.), no. 15(2022), N. 1 : 53 – 63. https://doi.org/10.15866/iremos.v15i1.20104

[14] Kunalan, Kerishmaa Theavy, Cheng Yee Ng, and Nauman Riyaz Maldar. "A performance investigation of a multi-staging hydrokinetic turbine for river flow." *Progress in Energy and Environment* 17 (2021): 17-31. https://doi.org/10.37934/progee.17.1.1731

[15] Schuiling, Bart, and Tom van Terwisga. "Hydrodynamic working principles of Energy Saving Devices in ship propulsion systems." *International Shipbuilding Progress* 63, no. 3-4 (2017): 255-290. https://doi.org/10.3233/ISP-170134

[16] Nowruzi, Hashem, and Amin Najafi. "An experimental and CFD study on the effects of different pre-swirl ducts on propulsion performance of series 60 ship." *Ocean Engineering* 173 (2019): 491-509. https://doi.org/10.1016/j.oceaneng.2019.01.007

[17] Prihandanu, R. B., I. M. Ariana, and D. W. Handani. "Analysis of Stern Shape Effect on Pre-Duct Propeller Performance Based on Numerical Simulation." In *IOP Conference Series: Materials Science and Engineering*, vol. 1052, no. 1, p. 012016. IOP Publishing, 2021. https://doi.org/10.1088/1757-899X/1052/1/012016

[18] van Terwisga, Tom. "On the working principles of Energy Saving Devices." In *Proceedings of the Third International Symposium on Marine Propulsors. Launceston, Tasmania, Austrailia*. 2013.

[19] Munazid, A., I. M. Ariana, and I. K. A. P. Utama. "Numerical Study of Pre-Duct on Traditional Fishing Vessels as Energy Saving Device (ESD)." In *IOP Conference Series: Earth and Environmental Science*, vol. 557, no. 1, p. 012049. IOP Publishing, 2020. https://doi.org/10.1088/1755-1315/557/1/012049

[20] Chang, Xin, Shuai Sun, Yuchang Zhi, and Yuming Yuan. "Investigation of the effects of a fan-shaped Mewis duct before a propeller on propulsion performance." *Journal of Marine Science and Technology* 24, no. 1 (2019): 46-59. https://doi.org/10.1007/s00773-018-0530-x

[21] Furcas, Francesco, Giuliano Vernengo, Diego Villa, and Stefano Gaggero. "Design of wake equalizing ducts using RANSE-based SBDO." *Applied Ocean Research* 97 (2020): 102087. https://doi.org/10.1016/j.apor.2020.102087

[22] Abar, Insanu Abdilla Cendikia, and I. K. A. P. Utama. "Effect of the incline angle of propeller boss cap fins (PBCF) on ship propeller performance." *International Journal of Technology* 10, no. 5 (2019): 1056-1064. https://doi.org/10.14716/ijtech.v10i5.2256

[23] Suastika, Ketut, Fajar Nugraha, and I. K. A. P. Utama. "Parallel-middle-body and stern-form relative significance in the wake formation of single-screw large ships." *International Journal of Technology* 8, no. 1 (2017): 94-103. https://doi.org/10.14716/ijtech.v8i1.3542

[24] Muhammad, Andi Haris, 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

[25] Carlton, John. *Marine propellers and propulsion*. Butterworth-Heinemann, 2018. https://doi.org/10.1016/B978-0-08-100366-4.00002-X