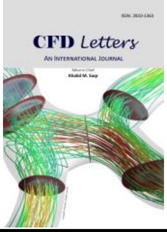
CFD Letters

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



Application of CFD Simulation to Determine the Optimal Horizontal Position of the Nozzle Inlet of a Propeller Flow Cooling System (PFCS)

Faisal Mahmuddin[1,*], Muhammad Iqshal Naitullah Jasman[1], Syerly Klara[1], Ahmad Fitriadhy[2]

[1] Marine Engineering Department, Engineering Faculty, Hasanudin University, Indonesia
[2] Program of Naval Architecture, Faculty Ocean Engineering Technology, and Informatics, Universiti Malaysia Terengganu, Malaysia

ARTICLE INFO

Article history:
Received 11 April 2023
Received in revised form 15 May 2023
Accepted 14 June 2023
Available online 10 December 2023

Keywords:
Inlet Horizontal Distance; Computational Fluid Dynamic; Engine Cooling System; Propeller Flow Cooling System; Experiment Validation

ABSTRACT

Research on the use of fluid flow caused by the propeller to be used for engine cooling is an interesting research topic. The system is known as the Propeller Flow Cooling System (PFCS). It is necessary to develop the most optimal position to produce maximum water discharge in this system. This research carried out simulation and analysis with an elliptical inlet nozzle as the utilization of the stern flow of the ship. This study aims to determine the difference in the volume of water generated from several horizontal positions of the nozzle inlet. In this study, the simulated horizontal distance of the inlet nozzle to the propeller at 0.7R propeller blade that is, 5 cm, 7.5 cm, 10 cm, 12.5 cm, 15 cm and using the computational dynamic fluid (CFD) method to analyze the optimal horizontal distance of the inlet nozzle which produces the highest water flow rate. Based on the research result, it was shown that the optimal horizontal distance of the inlet nozzle to the propeller is a distance of 7.5 cm. The water flow generated in the computational simulation in this case was 14.24 liters/minute. In that case, it can be concluded that the effect of the horizontal distance of the inlet nozzle on the propeller greatly affects the flow of water produced.

1. Introduction

In the area of South Sulawesi, Indonesia, it is common that traditional fishing boats to take advantage of the flow due to propeller thrust by placing the position of the inlet nozzle for cooling the main engine of the ship. The system is known as the Propeller Flow Cooling System (PFCS). Unfortunately, the system has been implemented without a proper scientific approach, especially related to the distance of the nozzle to the propeller. Therefore, the resulting fluid flow cannot be channeled optimally. The resulting consequences of the lacking of cooling water engines on the ship are very dangerous for the ship's engines [1, 2].

The placement of the position of the inlet nozzle both vertically and horizontally determines the produced water flow amount to meet the cooling water needs of the ship's engine [3, 4]. The current approach suggests utilizing dynamic pressure measurements that may be acquired at a single place. No one has previously employed dynamic pressure measurements and deep learning approaches

* *Corresponding author.*
E-mail address: f.mahmuddin@gmail.com (Faisal Mahmuddin)

https://doi.org/10.37934/cfdl.16.2.5566

together to distinguish between distinct system flows in horizontal pipes [5]. Therefore, the optimal horizontal distance position with maximum fluid flow needs to be determined so that the amount of fluid that enters the cooling system of the ship's engine can be obtained to the maximum as well.

Research on the utilization of fluid flow due to the thrust of the ship's propeller as a source of cooling water of the ship's engine without the use of pumps needs to be studied especially related to the optimal position horizontally so that the fluid flow discharge produced is maximal. Mahmuddin [6] conducted research on the shape and vertical position of the inlet nozzle on the utilization of the ship's stern flow as the cooling water source of the ship's main engine. In this study, it was proven that the optimal vertical position of the inlet nozzle is located at 0.7R propeller blade. Previous research also conducted an investigation on variations in aspect ratio of inlet nozzles the most optimal form of ellipses. This study proves that the most optimal aspect ratio is 0.9.

However, the development of technology to research the utilization of fluid flow caused by propeller thrust needs to be done regarding the most optimal horizontal distance placement in order to produce maximum water discharge. The propeller is an engine component mounted on a shaft that is directly connected to the ship's engine. With the rotating propeller, the ship will get the power to move. The flow caused by the rotation of the ship's propeller will be higher in pressure if the rotation of the driving motor increases. The flow of thrust due to propeller rotation is what will be used as a source of meeting the water needs of the ship's engine cooling machine without the use of a pump. The flow phenomenon due to propeller thrust can be utilized by installing an inlet nozzle horizontally to capture the flow of propeller thrust water as a cooling water source of ship engines without the use of pumps. Therefore, the present study will analyze the horizontal distance of the PFCS inlet nozzle to the propeller blade.

2. Literature Review

2.1 Propeller Flow Cooling System (PFCS)

A cooling system is extremely needed in the machinery industry [7, 8]. The Propeller Flow Cooling System (PFCS) is a system that uses the high-pressure flow in front of the propeller to be directed to the main engine to cool the engine of the boat [6]. Because the cooling system on a boat is not smooth, the performance of the ship depends heavily on the outcomes of the machine's modified work with the cooling lines in the engine main. By using the system, the boat does not need to install a cooling pump anymore. In order to use the system, an intake nozzle is placed in front of the propeller as shown in Figure 1. When the boat propeller rotates, it will create a high-pressure flow. Some of the water flow will enter the intake. The water will continuously flow as long as the propeller rotates. This flow is directed and circulated to the main engine using a pipe. The water circulation will cool the main engine. After cooling the engine, the hot water is directed to be discharged outboard. The PFCS main components and concepts are sketched in Figure 1.

PFCS has several main components as shown in Figure 1, which are the intake nozzle, cooling pipes (hot and cold water pipes), and outtake nozzle. Even though PFCS has several variations, the differences in the system are mainly in the position and shape of the intake nozzle. For the shape of intake nozzle, the most commonly installed shapes are circular and elliptical shapes. Therefore, conducting research related to the performance of these shapes is an interesting topic.

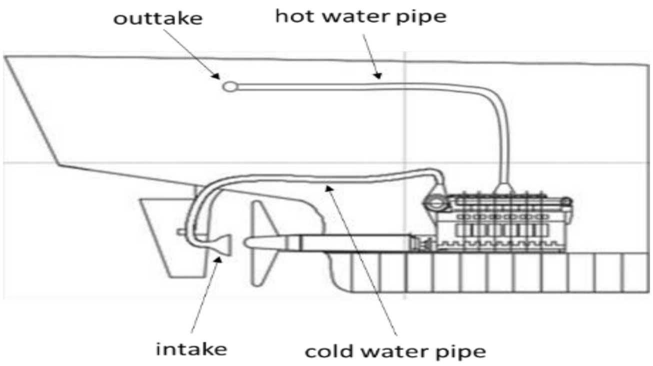


Fig. 1. The PFCS main components and concept [6]

2.2 Ansys Fluent

One of the technologies that may be utilized to explore the issue and save money and time is Computational Fluid Dynamics (CFD). However, using the program effectively demands knowledge and is highly challenging and requires much study [9]. One of the CFD software that is commonly used to modeling the particle flow is ANSYS Fluent. The software performs rather well when generating crucial forecasts [10]. Therefore, Ansys Fluent was used in the present study. FLUENT is a computer program used to simulate fluid flow and heat transfer. The flow and transfer of heat from various fluids can be simulated in complex shapes/geometries [11]. By using the FLUENT program, the desired flow and heat transfer parameters can be determined. The pressure distribution, flow speed, mass flow rate, and temperature distribution can be known at each point in the system analyzed [12]. FLUENT is powered by triangular-quadrilateral type 2D mesh, 3D tetrahedral-hexahedral-pyramid-wedge, and mixed mesh (hybrid). FLUENT also makes it possible to smooth or enlarge an existing mesh. FLUENT has an efficient and more flexible data structure, as FLUENT is written in C language. FLUENT can also be run as a separate process simultaneously there are desktop workstation clients and computer servers.

3. Research Methodology

3.1 Problem Definition

When it is important to determine the direction and characteristics of fluid flow when leaving or entering an enclosed space in a pipe, a nozzle is a device used to identify the direction and properties of the flow. In general, the nozzle's purpose is to enhance fluid flow velocity while simultaneously lowering pressure [13]. The shape of the inlet nozzle that will be simulated is the shape of the ellipse while the position of the inlet nozzle is at 0.7R propeller blade as shown in Figure 2.

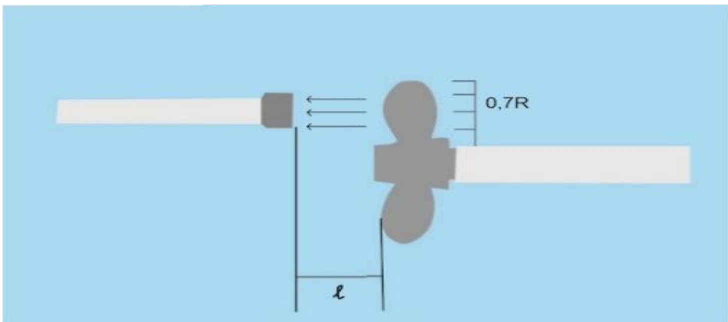


Fig. 2. Horizontal distance of the nozzle inlet to the propeller

For the purpose of the results validation, cases computed in the present study are the same cases as the ones tested in the study conducted by Safiu [14]. In that study, the variations in the horizontal position of the inlet nozzle were divided into 5 cases which are shown in Table 1.

Table 1
Horizontal position variation of the inlet nozzle

No	Name	Horizontal distance to propeller (ℓ) cm
1	Case 1	5.0
2	Case 2	7.5
3	Case 3	10
4	Case 4	12.5
5	Case 5	15

3.2 Models and Meshing

Researchers are increasingly employing complex methods such as computational fluid dynamics (CFD) to analyze flow and performance in the analysis of complex heat exchangers when designing new equipment and processes, which is a recent development [15]. The purpose of this complex method is to explain and solve complex problems. The main input needed to useb by CFD method usually are CAD geometry, computational mesh, material properties, boundary conditions, etc [16]. The analysis to be carried out is modeling, simulation, and calculation of fluid flow velocity in the pipe. Pre-processing, processing, and post-processing are the three phases of the simulation [17]. The sole phase covered by this study is pre-processing. Importing geometry from computer-aided design (CAD) software into Ansys Fluent, the meshing process, and setting up the simulation or specifying boundary conditions make up the pre-processing activity [18, 19]. The computed model geometry design data is shown in Table 2. The illustration of the model is shown in Figure 3.

Table 2
Computed model geometry design

No	Name	Horizontal distance to propeller (ℓ) cm
1	Body	Part 1
2	Surface area	0.01048 m^2
3	Faces	3
4	Edges	3
5	Vertices	2
6	Fluid/Solid	Fluid
7	Shared topology method	Automatic
8	Geometry type	Workbench

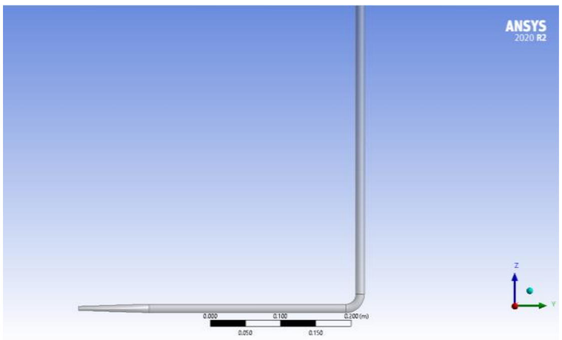


Fig. 3. Design of simulated model geometry

Figure 3 shows that each test simulation uses the same design. In the ansys simulation, variations in water flow velocity are used as test analysis parameters. The geometric meshing conditions that are used in the computed are written in Table 3. The meshing appearance is shown in Figure 4.

Table 3
Geometric meshing conditions

No	Parameter	Information
Scope		
1	Scoping methods	Geometry selection
2	Geometry	1 Body
Definition		
3	Suppressed	No
4	Element Size	3.e-003 m
5	Number of Element	74.246
6	Node	23,261
7	Behavior	Soft
8	Local Min Size	(2.8024e-005m)

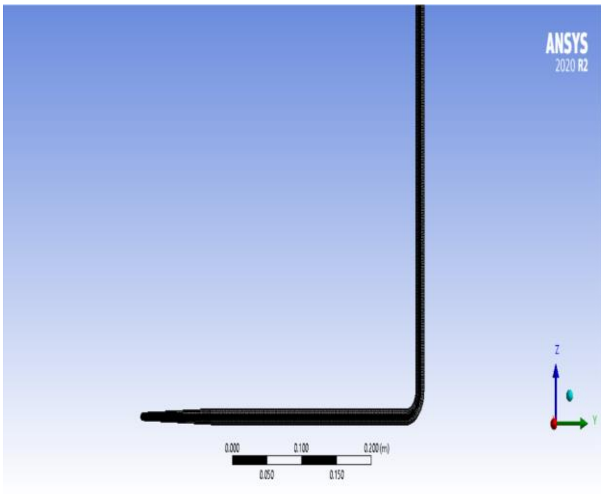


Fig. 4. Meshing geometry in ansys fluent

Table 4 shows the set-up condition in the fluid domain. The setup is mainly related to the material applied in the model.

Table 4
Set-up conditions in the fluid domain

No	Parameter	Information
1	Viscous model	k-epsilon (2 eqn)
2	K-epsilon model	Realizable
3	Near-wall treatment	Enhanced wall treatment
4	Material	Water-liquid
5	Density(kg/m)	998.2 constant
6	Viscosity (kg/ms)	0.001 constant

The reference values in the boundary conditions are shown in Table 5.

The next stage is the final stage of the simulation process, namely the running stage. At this stage, the results of the simulation will be displayed. In this study, the desired result is the value of the water flow rate (m^3/s) inside the nozzle inlet.

Table 5
Reference values in boundary conditions

No	Parameter	Information
1	Area (m2)	1
2	Density (kg/m3)	998.2
3	Enthalpy (j/kg)	None
4	Length (m)	1.6
5	Pressure (Pascal)	None
6	Temperature (k)	288.16
7	Viscosity (kg/ms)	0.001003
8	Reference Zone	Part 1

3.3 Flow Rate vs Engine Speed

In the present research, the engine rotation was arranged to four engine rotations (n), namely: 900 rpm, 1100 rpm, 1300 rpm, and 1500 rpm. These engine rotations determine the water velocity. The higher the engine rotations, the higher the water velocity.

3.4 Model Validation
3.4.1 Circulation water channel

A circulation water channel (CWC) basically is a tank equipped with a ship propulsion system simulator (propeller). This tank is designed with the principle that the flow of water simulator ship propulsion system can circulate in the tank (see Figure 5).

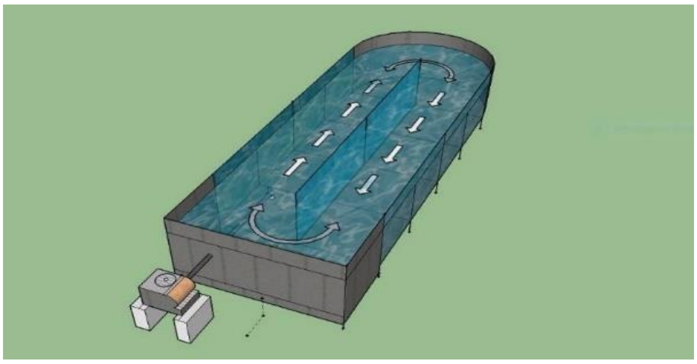


Fig. 5. The sketch of the Circulation Water Channel (CWC)

The main dimension of the CWC is shown in Table 6. The facility is located at Marine Engineering Department, Hasanuddin University.

Table 6
CWC main dimension [6]

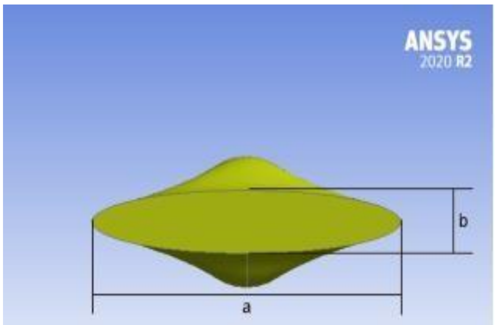
Parameter	value
Length (L)	9.0 m
Breadth (B)	2.4 m
Height (H)	1.2 m
Water weight (M)	21 tons

3.4.2 Propeller

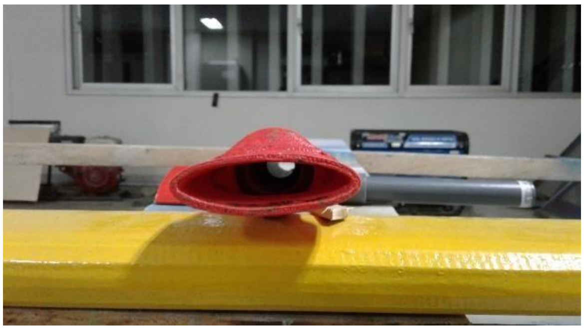
Propeller is one of the parts or components of the ship's motion device that is moved by an engine that serves to produce thrust and direct the movement of the ship [20]. Propellers are often mounted on a holder low on the ship's stern [21]. The thrust will depend on the specific location of the shaft and the angle at which the propeller shaft is inclined [22]. In this study, the propeller simulated in the present study is also the same as the one used in the experiment conducted by Safiu [14] which is a propeller with two blades and a propeller diameter of 8 inches.

3.4.3 Shape description

The inlet shape that will be computed and analyzed in the present study is circular. The previous research found that a circular shape will give a higher performance than the circular one [6]. The shape is drawn in Figure 6(a). The present research will compare the computational results with the results of an experiment conducted by Safiu [14]. In this experiment, the manufactured shape is shown in Figure 6(b).



(a)



(b)

Fig. 6. The shape of the nozzle inlet [14]

The dimension of the shape is shown in Table 7.

Table 7
Dimensions of shapes

Parameter	Value
Diameter	Da = 0.58 cm, Db = 6.2 cm
Surface area	0.028 m^2
Aspect ratio	0.09
Length of pipe distribution	4.5 M

4. Results and Discussion

4.1 Ansys Fluent Simulation

In the computational simulation using ANSYS, the engine speed is set to four levels of engine speed (n), namely 900 rpm, 1100 rpm, 1300 rpm, and 1500 rpm. The produced water debit that enters the inlet nozzle for each case is shown in Table 8.

Table 8
Produced water debit (liter/minute) from the computational simulation

Case	900 rpm	1100 rpm	1300 rpm	1500 rpm
1	6.69	7.70	9.20	9.87
2	8.03	9.87	11.21	13.05
3	7.53	8.70	10.87	12.04
4	6.36	7.70	8.70	9.37
5	5.69	6.02	7.86	8.53

Table 8 shows the comparison between the results from the computational simulation and the ones obtained from the experiment. It shows that the highest water discharge at 1500 rpm engine speed was at Case 2 which was approximately 13.05 liters/minute. In other engine rotations, the highest water debit also occurred in Case 2. Therefore, it can be said the most optimal setting is the one when the distance of the propeller from the inlet nozzle was 7.5 cm.

In order to verify the results obtained from the simulation, these results are compared with the ones obtained from the experiment conducted by a previous study. The experiments were conducted in the Circulating Water Channel (CWC) shown in Figure 5. The tested model is shown in Figure 6. The experimental set-up is shown in Figure 2. The experiments are conducted to validate the computational results for 5 (five) cases. Each case is conducted for 4 (four) propeller rotations. The experiment results and their percentage difference are shown in Table 9.

Table 9
Produced water debit obtained from experiment and their percentage difference with simulaton results

Case	900 rpm		1100 rpm		1300 rpm		1500 rpm	
	Q (liter/minute)	Err (%)	Q (liter/minute)	Err (%)	Q (liter/minute)	Err (%)	Q (liter/minute)	Err (%)
1	6.86	2.54	7.88	2.34	9.41	2.28	10.04	1.72
2	8.24	2.62	9.98	1.11	11.44	2.05	13.28	1.76
3	7.68	1.99	8.88	2.07	11.04	1.56	12.24	1.66
4	6.48	1.89	7.78	1.04	8.87	1.95	9.58	2.24
5	5.86	2.99	6.69	11.13	7.92	0.76	8.75	2.58
	Avg=	2.4	Avg=	3.5	Avg=	1.72	Avg=	2.0

From Table 9, it can be noted that the average error was approximately 2% which can be considered to be small. Therefore, it can be concluded that the results obtained were reliable. For the easy comparison, the results from there 2 (two) methods are plotted in a graph as shown in Figure 7.

Based on the graph in Figure 7, in all cases, the experimental results are slighty higher than the one from experiment. However, in general, it can be seen a good agreement between computational and experimental results so therefore, it can conclude that the computations are valid and reliable.

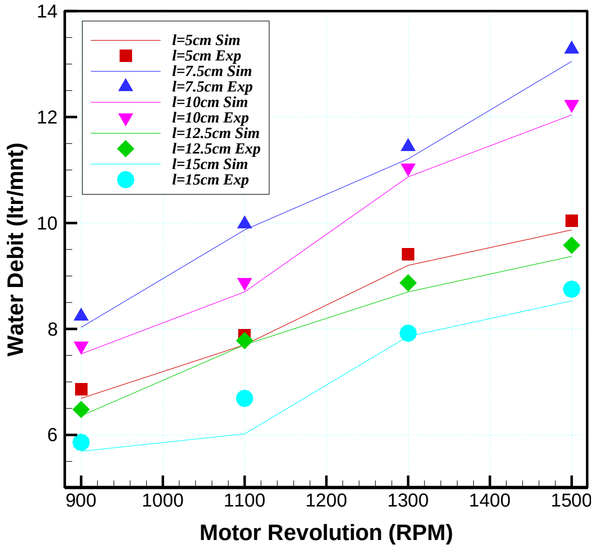


Fig. 7. Comparison and validation of water discharge obtain from experimental method and CFD simulation

The velocity contours of water flow inside the nozzle pipe from each case was also computed and obtained as shown in Figure 8-11.

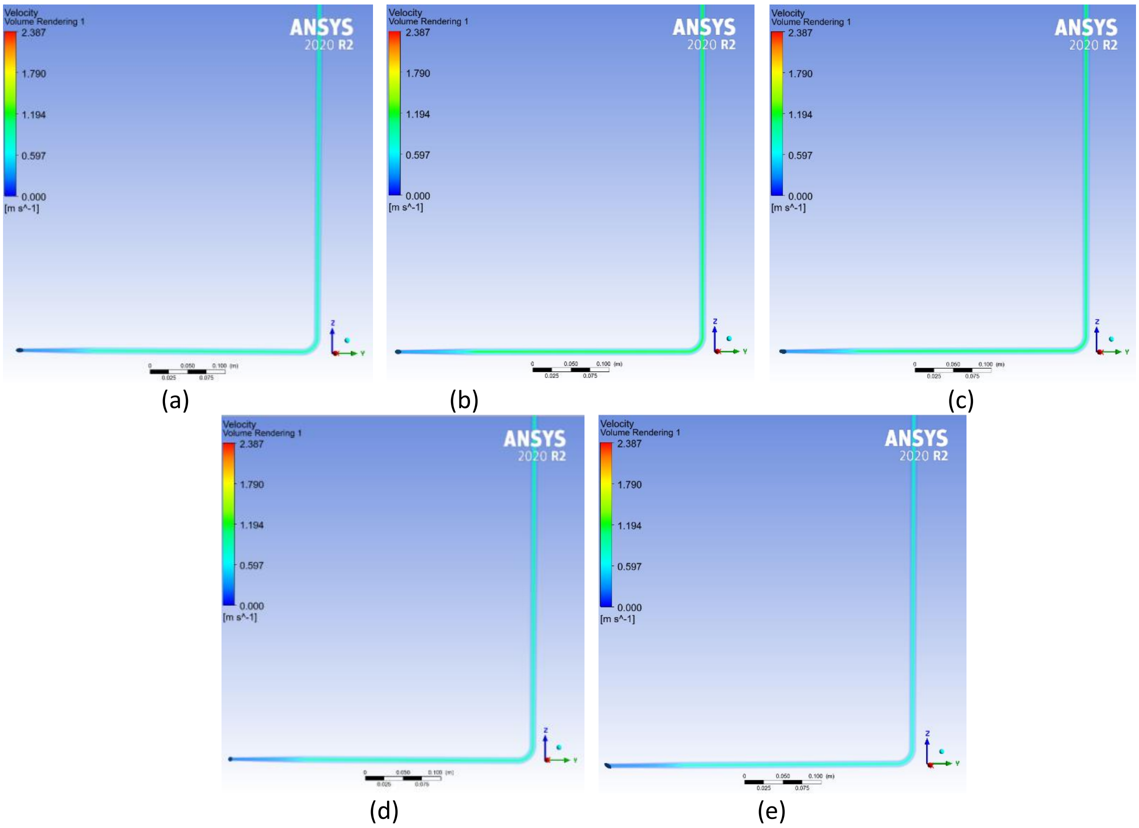


Fig. 8. Velocity contours of water velocity at inlet nozzle obtained when n=900 rpm (a) Case 1 (5 cm) (b) Case 2 (7.5 cm) (c) Case 3 (10 cm) (d) Case 4 (12.5 cm) (e) Case 5 (15 cm)

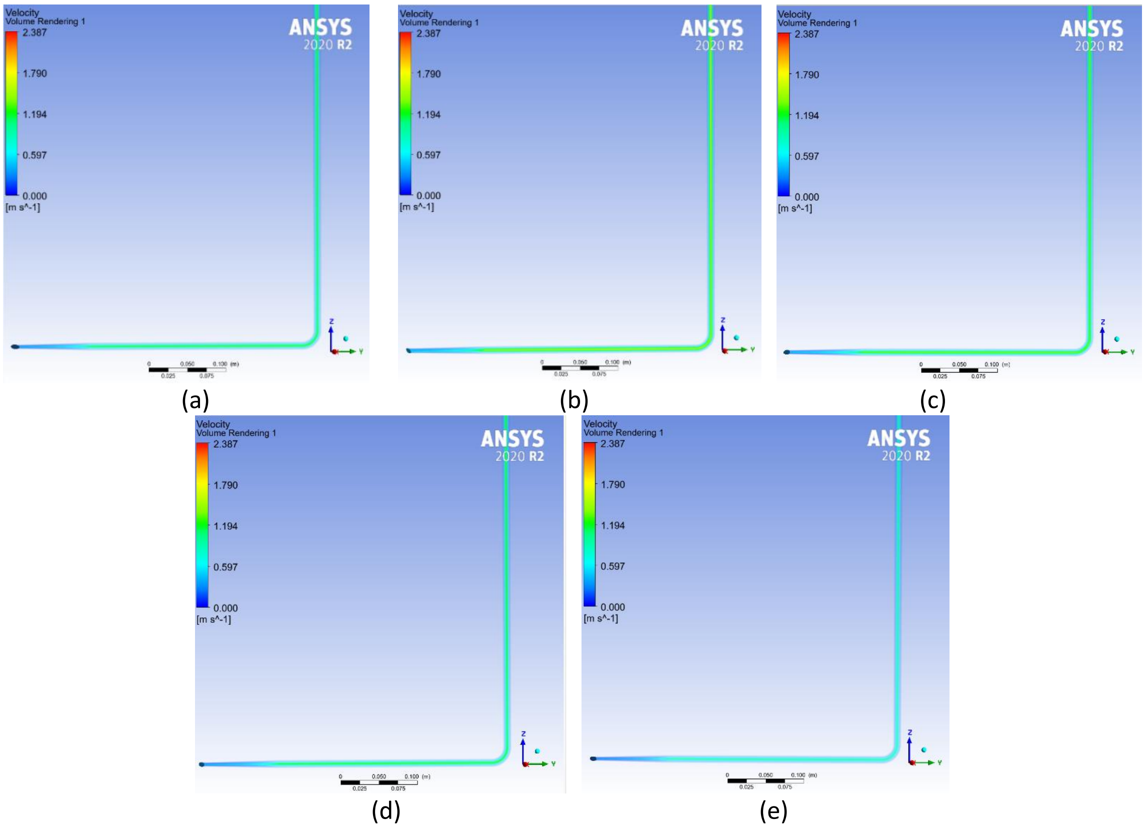


Fig. 9. Velocity contours of water velocity at inlet nozzle obtained when n=1100 rpm (a) Case 1 (5 cm) (b) Case 2 (7.5 cm) (c) Case 3 (10 cm) (d) Case 4 (12.5 cm) (e) Case 5 (15 cm)

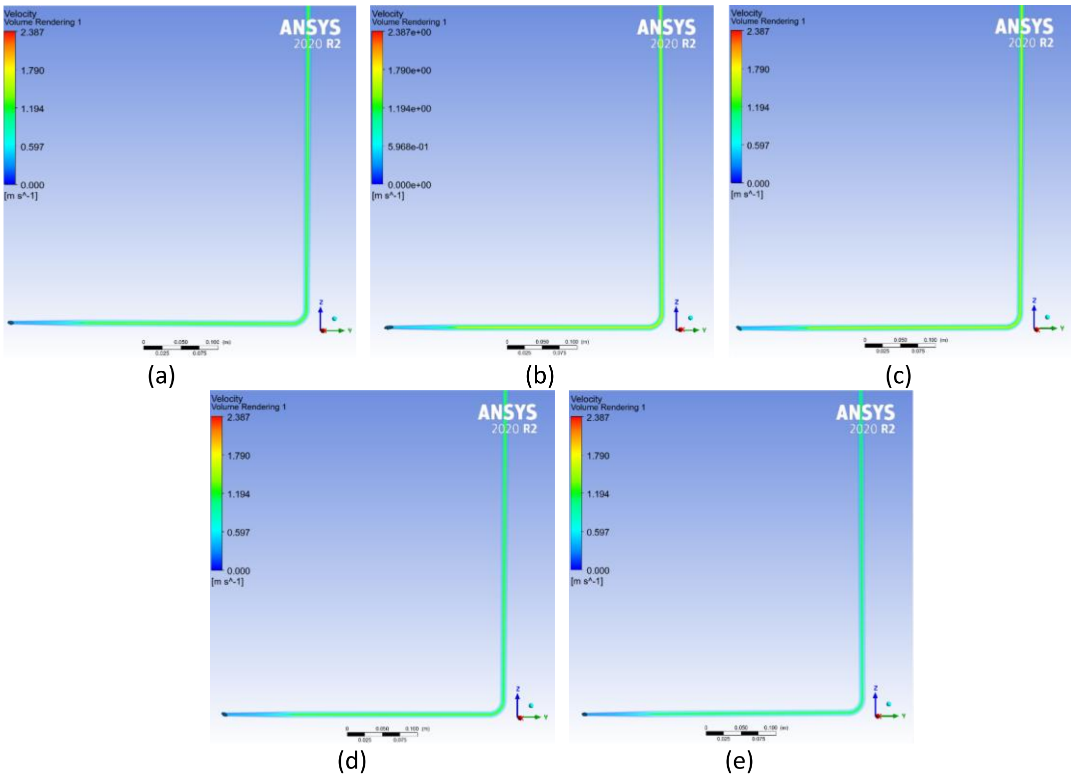
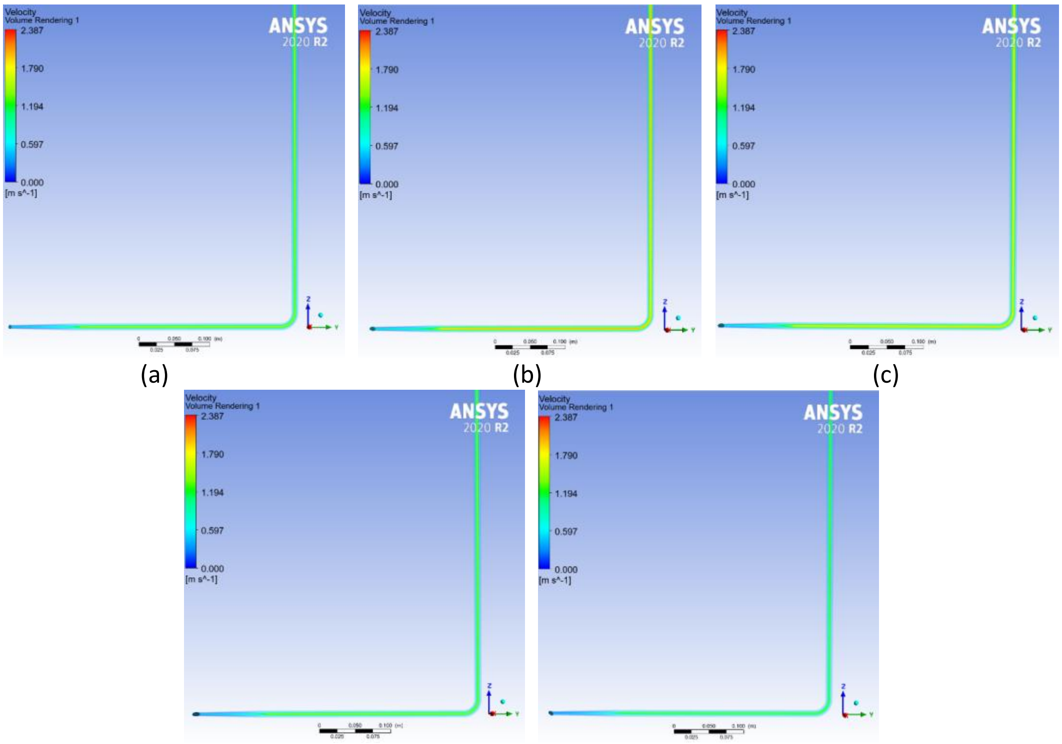


Fig. 10. Velocity contours of water velocity at inlet nozzle obtained when n=1300 rpm (a) Case 1 (5 cm) (b) Case 2 (7.5 cm) (c) Case 3 (10 cm) (d) Case 4 (12.5 cm) (e) Case 5 (15 cm)



(a) (b) (c)

(d) (e)

Fig. 11. Velocity contours of water velocity at inlet nozzle obtained when n=1500 rpm (a) Case 1 (5 cm) (b) Case 2 (7.5 cm) (c) Case 3 (10 cm) (d) Case 4 (12.5 cm) (e) Case 5 (15 cm)

In Figs. 8-11, the water velocity set at the nozzle inlet was the one obtained from the experiment. From these figures, it can be seen the flow velocity for each case for 4 (four) engine rotations. The shown velocity is not only in the nozzle inlet but also at all points in the distribution pipe. It can be seen from these figures that except at the inlet nozzle, the water velocity inside the pipe is constant and in case of all engine rotation, the highest performance was obtained for case 2 which has also been described previously.

5. Conclusion

Research shows that the horizontal distance of the inlet nozzle to the propeller greatly affects the increase in the water flow produced. The optimal horizontal position of the inlet nozzle for the utilization of the ship's stern flow as a source of cooling water for the ship's main engine based on experimental results is at a distance of 7.5 cm from the propeller with a flow velocity of 0.68 m/s which is 13.28 liters/minute and the results of computational simulations which is 14.24 liters/minute. From the results validation, it was also noted a good agremeent between the results fromcomputer simution and the ones from experimental methods were observed.

References

[1] Ariafar, Kavous, David Buttsworth, Navid Sharifi, and Ray Malpress. "Ejector primary nozzle steam condensation: Area ratio effects and mixing layer development." *Applied thermal engineering* 71, no. 1 (2014): 519-527. https://doi.org/10.1016/j.applthermaleng.2014.06.038

[2] I. Triyanti, "Analysis of the Effect of Foil Section Nozzle Shape on Ship Propulsion Efficiency on Tugboats," Ten November Institute of Technology, 2015.

[3] D. Hafiz, "Analysis of the Effect of Fluid Flow Caused by Propeller Rotational Movements on Fishing Vessels on Propeller Pressure With Approach," Diponegoro University, 2011.

[4] F. M. White, *Fluid Mechanics*, 2nd ed. New York, 1991.

[5] Khan, Umair, William Pao, Nabihah Sallih, and Farruk Hassan. "Flow Regime Identification in Gas-Liquid Two-Phase Flow in Horizontal Pipe by Deep Learning." *Journal of Advanced Research in Applied Sciences and Engineering Technology* 27, no. 1 (2022): 86-91. https://doi.org/10.37934/araset.27.1.8691
[6] Mahmuddin, F., S. Ramadhan, A. Fitriadhy, and S. Klara. "Performance Comparison Between Ellipse and Circular Intake Shapes of Propeller Flow Cooling System (PFCS) with Numerical and Experimental Methods." In *IOP Conference Series: Materials Science and Engineering*, vol. 676, no. 1, p. 012020. IOP Publishing, 2019. https://doi.org/10.1088/1757-899X/676/1/012020
[7] Legiman and F. Sulaiman, "Maintenance And Repair Of The Mitsubishi Galant Engine Cooling System 2500 Cc," *Technovation J.*, vol. 1, no. 1, pp. 26–34, 2014.
[8] Astriawati, N. I. N. G. R. U. M., and W. A. R. I. S. Wibowo. "Perawatan sistem pendingin mesin diesel pada whell loader komatsu wa120-3cs." *Jurnal Teknovasi: Jurnal Teknik Dan Inovasi* 7, no. 2 (2020): 76-85.
[9] Anuar, Norsyazliana Syamira Saiful, Siti Hidayah Abu Talib, Nur Fatin Izureen Radzaly, Syarifah Intan Najla Syed Hashim, and Muhammad Salleh Abustan. "The effect of sedimentation to the pump sump system by using computational fluid dynamics (CFD) model." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 68, no. 1 (2020): 86-97. https://doi.org/10.37934/arfmts.68.1.8697
[10] B. Harinaldi, *Fluid Systems*. Jakarta: Erlangga Publisher, 2015.
[11] G. Mustafa, "CFD Analysis of Nozzle Effect on Jet Formation," Sweden, 2015.
[12] Li, Ben Q. Discontinuous finite elements in fluid dynamics and heat transfer. Springer Science & Business Media, 2006. https://doi.org/10.1007/1-84628-205-5
[13] Vahaji, Sara, Aliakbar Akbarzadeh, Abhijit Shridhar Date, Chi Pok Cheung, and Jiyuan Tu. "Study on the efficiency of a convergent-divergent two-phase nozzle as a motive force for power generation from low temperature geothermal resources." In *Proceedings of the World Geothermal Congress 2015 Australia-New Zealand (WGC 2015)*, pp. 1-14. Arinex Pty Ltd, 2015.
[14] Safiu, Mahmudin. "Pengaruh Jarak Horizontal Nosel Inlet Terhadap Performa Sistem Pendingin Mesin Berbasis Daya Dorong Propeller= Effect Horizontal Distance Of Intake Shape To Performance Propeller Flow Cooling System (Pfcs)." PhD diss., Universitas Hasanuddin, 2021..
[15] Bhutta, Muhammad Mahmood Aslam, Nasir Hayat, Muhammad Hassan Bashir, Ahmer Rais Khan, Kanwar Naveed Ahmad, and Sarfaraz Khan. "CFD applications in various heat exchangers design: A review." *Applied Thermal Engineering* 32 (2012): 1-12. https://doi.org/10.1016/j.applthermaleng.2011.09.001
[16] Andersson, Bengt, Ronnie Andersson, Love Håkansson, Mikael Mortensen, Rahman Sudiyo, and Berend Van Wachem. *Computational fluid dynamics for engineers*. Cambridge university press, 2011. https://doi.org/10.1017/CBO9781139093590
[17] Cifuentes, Oscar Darío Monsalve, Jonathan Graciano Uribe, and Diego Andrés Hincapié Zuluaga. "Numerical Simulation of a Propeller-Type Turbine for In-Pipe Installation." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 83, no. 1 (2021): 1-16. https://doi.org/10.37934/arfmts.83.1.116
[18] Alfarawi, Suliman SS, Azeldin El-sawi, and Hossin Omar. "Exploring discontinuous meshing for cfd modelling of counter flow heat exchanger." *Journal of Advanced Research in Numerical Heat Transfer* 5, no. 1 (2021): 26-34.
[19] T. Bikmukhametov, "CFD Simulations of multiphase flows with particles," NTNU, 2016.
[20] K. Yan, "Analisis Investigasi Pada Industri Pengecoran Propeller Kapal," Politeknik Negeri Ujung Pandang, 2012.
[21] Nursalim and A. Winarno, "Style Analysis Push the DPS IX Tug Because It Is Addition of Free Rotating Propeller With Using the Computed Fluid Method Dynamics (CFD)," Hang Tuah University, 2018.
[22] A. Munawir, "Study Propeller Shaft Against Marine Propulsion," *V-Mac J.*, vol. 2, no. 1, pp. 18–24, 2017.