Effect Horizontal Distance of Intake Shape to Performace Propeller Flow Cooling System (PFCS)

Mahmudin Safiu, Faisal Mahmuddin and Syerly Klara

September 29, 2021

Effect Horizontal Distance of Intake Shape to Performance Propeller Flow Cooling System (PFCS)

Mahmudin Safiu[1)], **F Mahmuddin**[1)], **and S Klara** [1)]

[1] *Marine Engineering Departement, Engineering Faculty, Hasanudin University, Indonesia*

,

Email : [a)] *mahmudinsafiu@gmail.com*
[b)] *f.mahmuddin@gmail.com*
[c)] *elikapal83@gmail.com*

Abstract : Research on the evolution of technology for the use of fluid flow caused by propeller thrust, it is necessary to develop the most optimal position to produce maximum water discharge. In previous research on the shape and variation of the vertical position of the inlet nozzle on the utilization of the ship's stern flow as a source of cooling water for the ship's main engine. This research carried out an analysis carried out is by using an ellips intlet 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 method uses is an experimental method that analyzes the 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 to produce the highest water flow rate in each test case. Based on the research result, it shows that the optimal horizontal distance of the inlet nozzle to the propeller is a distance of 7,5 cm. As from the water flow generated in the experiment is 13,38 liters/minute and the water flow generated in the computational simulation is 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 affect the flow of water produced. In process data collection it is better to use commonly uses on ship.

Key Words : Horizontal Distance of The Inlet Nozzle, Computational Fluid Dynamic, Water Discharge

1. INTRODUCTION

Generally in traditional fishing boats take advantage of the flow due to propeller thrust by placing the position of the inlet nozzle in the cooling system of the main engine of the ship without using a scientific approach to the distance of the nozzle to the right propeller leaves, so that the resulting fluid flow cannot be channeled properly. The resulting consequences of the unmet needs of cooling water engines on the ship are very dangerous for the ship's engines, one of which is overheating. In the placement of the position of the inlet nozzle both vertically and at the horizontal distance of the inlet nozzle determines the water flow capture power to meet the cooling water needs of the ship's engine. Then it needs to be set to get a horizontal distance position with maximum fluid flow capture, 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 about the optimal position horizontally so that the fluid flow discharge produced is maximal.. Syahrun (2018) 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 leavesIn previous research also conducted research on variations in aspect ratio of inlet nozzles the most optimal form of ellips. 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. 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, this study will analyze the horizontal distance to the propeller leaves.

2. LITERATURE REVIEW

2.1 Fluid Flow

The amount of liquid flowing through the latitude of the flow of each unit of time is called a flow and is noted Q. Flow discharge is usually measured in the volume of the liquid per unit of time, so that the unit is meters cubic per se (m^3/s) (Triatmodjo, 2014). In an ideal liquid, where there is no friction, the flow speed of V is the same at each point at latitude. If the appearance of the flow is perpendicular to the direction of the flow, then the flow discharge can be formulated.”

$$Q = A \cdot V \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \quad (2.\ 1)$$

Where,

Q = Water discharge (m^3/s)
A = Cross sectional area (m^2)
V = Velocity (m/s)

2.2 Cotinuity Equation

If a non-compressible liquid flows continuously through a pipe or open channel, with a constant or non-constant flow, then the volume of liquid passing through each unit of time is the same in all appearances. This state is called the law of continuity of the flow of liquid substances. (Triatmodjo, 2014). In pictures 2.3 It displays one-dimensional and steady flow, with an average speed of V and the appearance of flow A. The flow flows from point 1 on V1 and A1, to point 2 on V2 and A2.

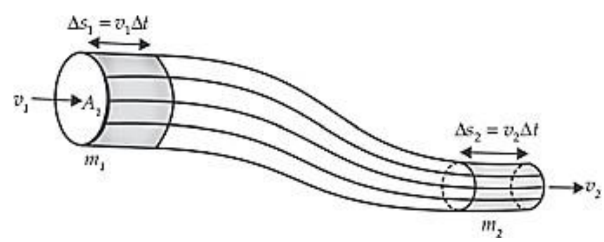

Figure 1. Flow tube to lower continuity equation

The volume of liquids entered through the appearance of 1 per unit of time: V1 A1. The volume of liquids coming out of the appearance of 2 per unit of time: V2 A2. Therefore, no liquid is lost in the flow tube, hence:

$$V1A1 = V2A2 \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \quad (2.\ 2)$$

or

$$Q1 = Q2 = konstan \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (2.\ 3)$$

2.3 Bernoulli Equation

Bernoulli's decrease in equations for flow along current lines is based on Newton's SECOND law of motion ($F = ma$) (Triatmodjo, 2014).

a) Liquids are ideal, so they have no viscosity (energy loss due to friction is zero)
b) Liquids are homogeneous and unsup contained (the density of the mass of liquids is constant)
c) Flow is continuous and along current lines.
d) Flow speed evenly in a cross-section.
e) Styles that work only heavy and pressure styles.

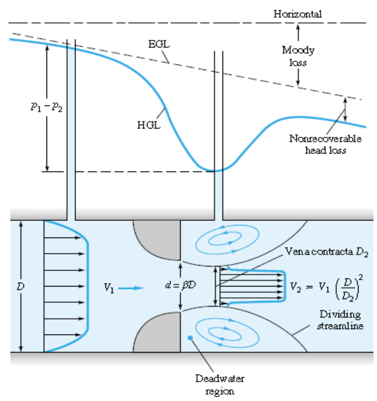

Figure 2. Changes in pressure and speed past Bernoulli obstruction meter.

2.4 Hydrodynamics Propeller

Hydrodynamics is an event in which the speed between the top and bottom of the hydrofoils occurs. The fluid that passes through the top of the airfoil travels faster than the fluid that passes through the bottom. This is due to the pressure difference between the upper fluid flow and the lower fluid flow. As we know that the amount of pressure is inversely proportional to the magnitude of the speed. So what happens is the flow of fluid through the bottom of the hydrofoil is slower when compared to the top of the hydrofoil. The difference in pressure that occurs is what then eventually gives rise to the phenomenon of lift or lifting style.

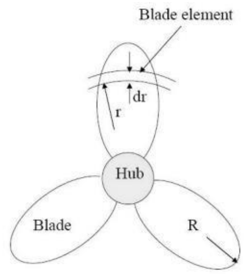

Figure 3. Blade element

2.5 Fluid Kinematics

In the application of engineering fields related to fluid systems, generally the fluid involved is in a moving state or better known as "flowing". Fluid kinematics studies different aspects of fluid movement without reviewing the forces required to produce such movements. Kinematic studies of such movements include speed, acceleration of flow field and depiction and visualization of the movement. An understanding of fluid flow kinematics is an important basis for understanding fluid dynamics. (Harinaldi, 2015).

2.6 Head Losses

Head loss is a phenomenon of flow losses in the piping system. Flow losses always occur in piping systems using a variety of fluids, such as liquid fluids and gases. In general, the largest flow loss occurs in liquid fluids, this is due to the solid molecular properties compared to gases and has greater friction against the media that is traversed is greater, then the friction that occurs will be greater. In addition, the most basic consequence with the loss of flow (head loss) is that it can cause the amount of energy needed to move fluid flow which has an impact on the increasing use of electricity in fluid drive machines such as pumps. Friction will cause a decrease in pressure or loss of energy along the flow.

Based on the location of the onset of loss, in general, the loss of pressure due to friction or loss is classified into 2 types of flow losses, namely major losses and minor losses. Major loss is the loss of pressure due to friction of fluid flow on a straight pipe called "major" because the pipe is straight as its main component. While minor losses are the loss of pressure that occurs in addition to straight pipes such as losses on valves, T connections, L connections, and so on. Called "minor" because the valves, T joints, L connections are components of the support pipe. Two fluid flow losses due to friction along this flow component are called head losses. (Triatmodjo, 2014).

$$H_{Total} = H_{lmy} + H_{lm} \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \quad (2.\ 5)$$

Where,

H_{lmy} = Head Loss Mayor (m)
H_{lm} = Head Loss Minor (m)

2.6.1 *Head Loss Mayor*

In the flow of the friction coefficient value only as a function of Reynold numbers only, because the kaminar flow is not influenced by the roughness factor of the pipe surface. But with the higher reynolds number, the coefficient of friction (f) is only a function of the relative roughness of the pipe surface. Pressure decrease (P) in turbulent flow is a function of reynolds (Re), Ratio of lenght and pipe diameter (L/D) (Triatmodjo, 2014).

$$H_{lmy} = = f.\frac{L}{D}.\frac{v^2}{2.g} \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \quad (2.\ 6)$$

Where,

H_{lmy} = Head Loss Mayor (m)
f = Friction coefficient
L = Length of pipe (m)
v = Velocity (m^3/s)
g = Acceleration of gravity (m/s^2)
D = Diameter (m)

The value f is affected by the type of flow (laminer, transition or turbulence) and pipe material. Pipe material shows the magnitude of roughness on the surface of the pipe.

$$Re = \frac{\rho V D}{\mu} \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \quad (2.\ 7)$$

Where,

Re = Reynolds number
ρ = Density (kg/m^3)
μ = Viskocity ($N.s/m^2$)

Based on experiments conducted by Osbourne Reynolds set about the value of Reynold numbers used to distinguish between laminar, turbulent, and transition flows. The range of values is:

- In Re <2300, laminar flow
- In Re = 2300 – 4000, transition flow
- In Re > 4000, turbulent flow

2.6.2 *Head Loss Minor*

Minor head loss is a small flow loss on the plumbing system. In the flow that passes through the turn and valve the resistance coefficient (K) is a function of the ratio of length of the equivalent of the pipe component, the diameter of the pipe component (Le/D), and the relative roughness of the pipe (f). It can be calculated by equation, that is :

$$H_{lmy} = K.\frac{v^2}{2.g} \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \quad (2.\ 8)$$

Because

$$K = f.\frac{l}{D} \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \quad (2.\,9)$$

Where,

H_{lm}	= *Head Loss Minor* (m)
K	= Resistent coefficient
v	= Velocity (m^3/s)
f	= Relative roghness

2.7 Ansys Fluent

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. By using the FLUENT program, you can know the desired flow and heat transfer parameters. The pressure distribution, flow speed, mass flow rate, and temperature distribution can be known at each point in the system analyzed. 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 *Circulation Water Channel*

Circulation water channel (CWC) Basically, it 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.

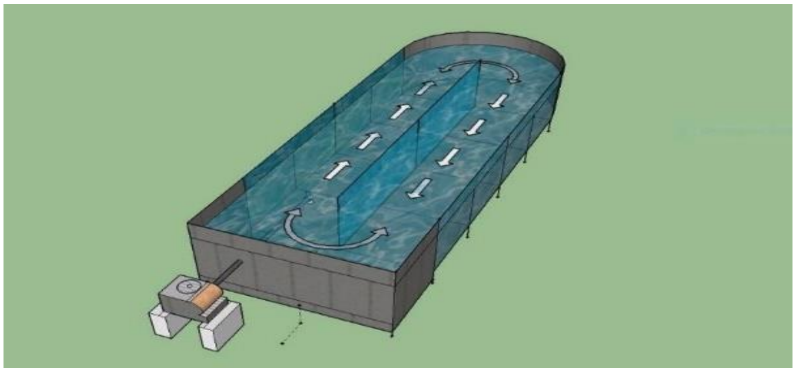

Figure 4. The sketch of *Circulation Water Channel (CWC)*

The main parameters of the CWC are shown in Table 1.

Table 1. CWC Dimension

Parameter	value
Length (L)	9,0 m
Breadth (B)	2,4 m
Height (H)	1,2 m
Water Weight (M)	21 ton

3.2 *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 to the movement of the ship. In this study, the propeller used is a propeller with two leaves and a propeller diameter of 8 inches.

Figure 5. The sketch of propeller

3.3 *Shape description*

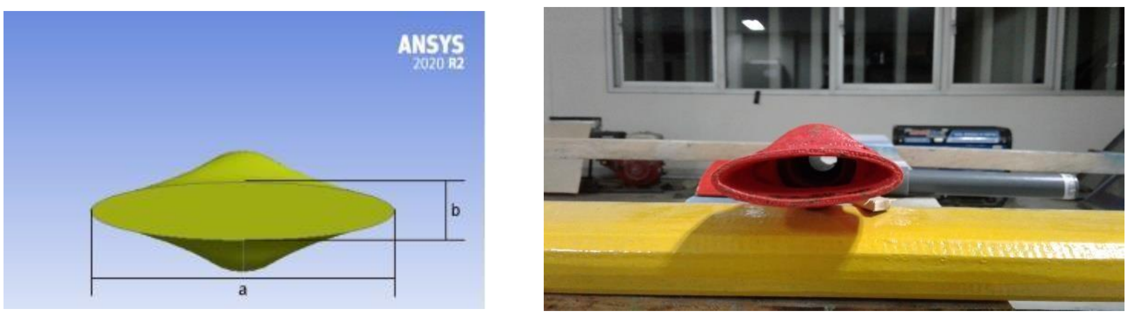

Figure 6. The sketch of Ellipse

Table 2. Dimension of shapes

Parameter	Value
Diameter	D_a = 0,58 cm, D_b = 6,2 cm
Surface area	0,028 m^2
Aspect ratio	0,09
Length of pipe distribution	4,5 m

3.4 *Horizontal distance*

The shape of the inlet nozzle that will be tested is the shape of the ellips while the position of the inlet nozzle is at 0.7R propeller leaves. Cases tested with variations in the horizontal position of the inlet nozzle are divided into 5 cases, namely:

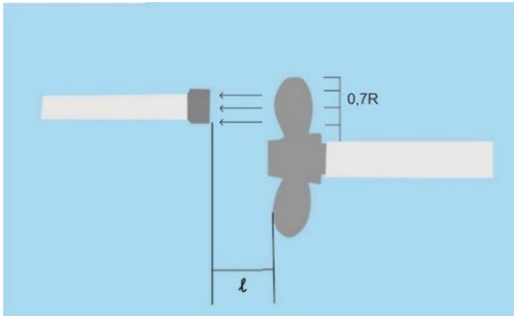

Figure 7. Horizontal distance intake shape to propeller

Table 3. Horizontal position variation of 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

4. RESULT AND DISCUSSION

4.1 *Schematic measurement*

At this stage researchers will connect the Flow Sensor to arduino uno to measure the speed of water flow in the cooling pipe of the ship's main engine which is then connected to the computer device in order to input the results of the discharge measurement with the flow sensor. To connect requires an arduino program. Flow sensor serves to input data on the speed of water flow in the engine cooling pipe, while the arduino application serves as a data input and data manager of the flow sensor.

4.2 *Flow rate vs Engine Speed*

This research, the engine rotation was arranged to four levels of spin, namely 900 rpm, 1100 rpm, 1300 rpm and 1500 rpm. The results of flow discharge measurements for various nozzle positions are shown in the table and gravik below:

Table 4. Experimental result

RPM	Q – 5 cm Liters/minute	Q – 7,5 cm Liters/minute	Q – 10 cm Liters/minute	Q – 12,5 cm Liters/minute	Q – 15 cm Liters/minute
900	6.68	8.24	7.68	6.48	5.86
1100	7.88	9.98	8.88	7.78	6.69
1300	9.41	11.44	11.04	8.87	7.92
1500	10.04	13.28	12.24	9.59	8.75

The average increase in water discharge at the horizontal nozzle distance of 5 cm is 1.18 liters / minute in each variation of engine rotation. In case 2 with a nozzle distance of 7.5 cm the average value of the increase in water discharge at each engine rotation is 1.68 liters / minute. The difference in discharge increase between the distance of the nozzle of 5 cm and the distance of the inlet nozzle of 7.5 cm is 0.5 liters / minute.

The average increase in water discharge at the horizontal distance of the inlet nozzle of 10 cm is 1.52 liters / minute. In case 4 with a nozzle distance of 12.5 cm, the average value of the discharge increase in each engine rotation is 0.96 liters / minute.

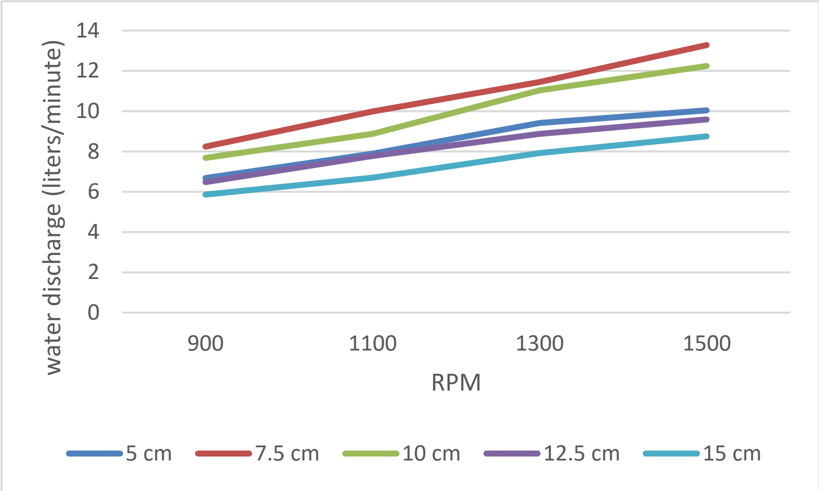

Figure 8. Flow rate in the experimental case

Based on the graph of the experimental results above, the comparison of the water discharge value at 1500 rpm rotation with the horizontal distance of the inlet nozzle 5 cm and 7.5 cm is 3.24 liters/minute. The water discharge in case 1 is lower than case 2, in case 1 the highest water discharge is at a flow velocity of 0.68 m/s, namely 13.28 liters/minute with the lowest water discharge value being 7.40 liters/minute at a flow velocity of 0, 45 m/s. Case 2 at a distance of 7.5 cm produces the highest water discharge from cases 1,3,4 and 5. In case 2 the water discharge produced at a flow velocity of 0.68 m/s is 13.28 liters/minute. Meanwhile, at a distance of 15 cm, the lowest water discharge is 8.75 liters/minute.

4.3 *Head Losses*

Based on the calculation of fluid flow velocity and head losses calculation, it can be explained that the fluid flow velocity in the pipe greatly affects the head loss in the piping system.

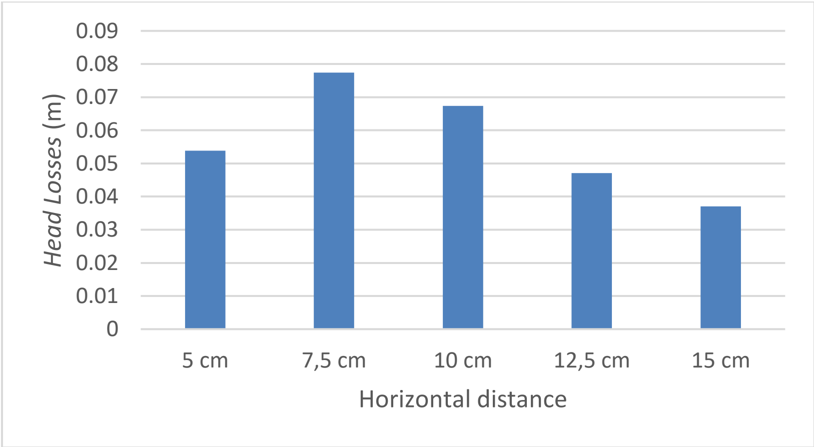

Figure 9. Head loss at 900 rpm

From the graph above, it shows that the highest Head Losses at 900 rpm engine speed with a horizontal horizontal distance of 7.5 cm from the inlet nozzle is 0.07744 m. Head losses at 1100 rpm engine speed at a distance of 10 cm from the propeller leaf are 0.09091 m. The highest head losses at 1300 rpm engine speed at a horizontal distance of the inlet nozzle of 12.5 cm, namely 0.09091 m. Meanwhile, the head losses of the piping system at 1500 rpm engine speed at 15 cm nozzle range are 0.08754. This shows that each engine speed and at different distances will give a different value of head losses, this proves that the head losses in this study depend on the speed of the water flowing in the pipe.

4.4 *Ansys Fluent Simulation*
In the computational simulation, the engine speed is set to four levels of engine speed, namely 900 rpm**,** 1100 rpm, 1300 rpm dan 1500 rpm. CFD simulation results are shown in the table and graph below :

Table 5. CFD simulation

RPM	Q1	Q2	Q3	Q4	Q5
900	6.92	8.56	7.84	6.42	5.98
1100	8.03	10.18	9.22	7.6	6.91
1300	9.61	12.01	10.81	8.84	8.01
1500	11.02	14.02	12.8	10.43	9.4

Based on the CFD simulation results, the highest water discharge at 1500 rpm engine speed is 14.02 liters/minute. In the experimental results, the average increase in the highest water discharge per engine speed in Q2 is 1.68 liters/minute, while the average increase in water discharge for each engine speed in Q2 in the CFD simulation results is 1.82 liters/minute.

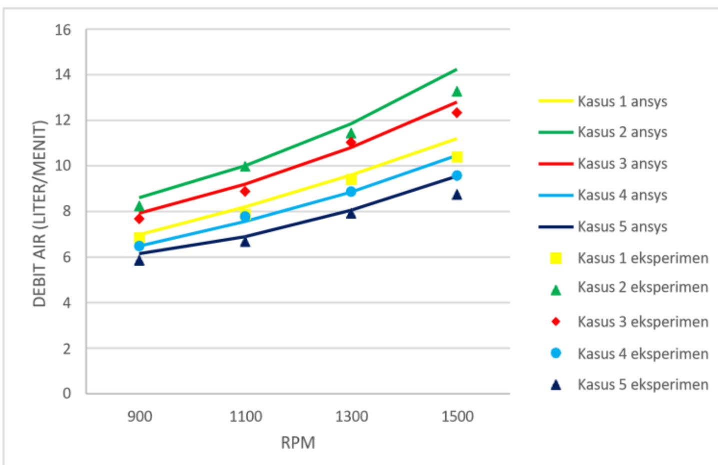

Figure 10. Comparasion water discharge an experimental and simulation CFD

Based on the graph above, the difference in water flow between the experimental results and the CFD simulation is at 1500 rpm the value of the Q2 water flow rate based on the experiment is 13.28 liters/minute, while the CFD simulation Q2 water discharge value is 14.02 liters/minute. At 1300 rpm the engine speed of Q1 water discharge value based on experimental results is 9.41 liters/minute, while the CFD simulation water discharge value in Q1 is 9.61 liters/minute.

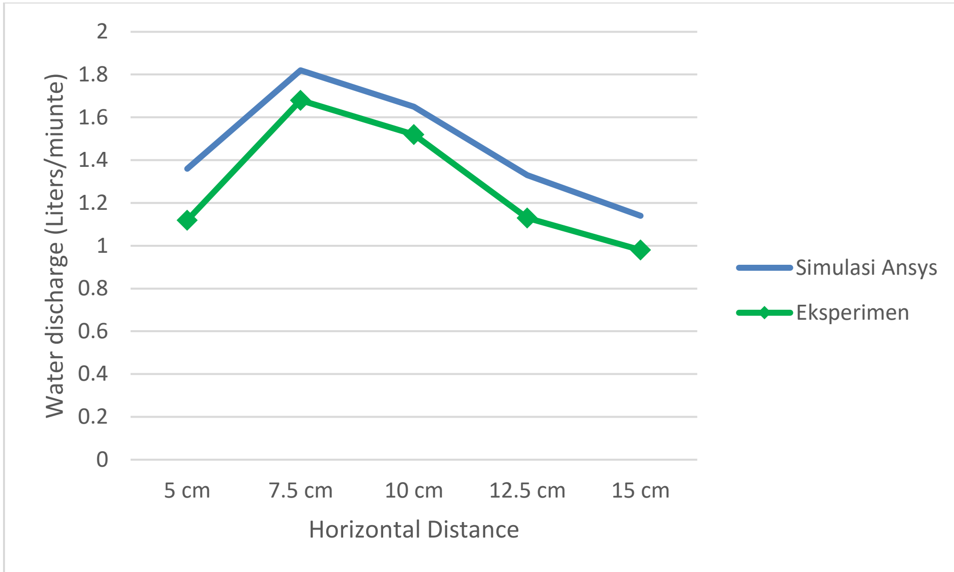

Figure 11. Ration of the average increase in water discharge at each engine speed

The comparison between the experimental results and the results of the Ansys Fluent analysis can be seen in the increase in the water discharge value for each engine speed. The average increase in engine speed in Q1 experimentally is 1.12 liters/minute, while the average increase in water discharge using CFD simulation is 1.36 liters/minute. The average increase in water discharge in Q2 experimentally is 1.68 liters/minute, while the average increase in water discharge at each engine speed using CFD simulation is 1.82 liters/minute.

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. The results showed that if the distance of the inlet nozzle to the propeller was getting closer, the smaller the flow would be.

6. ACKNOWLEGMENT

This research was conducted as one of the requirements for the completion of undergraduate studies at the Faculty of Engineering, Hasanuddin University. This research is very important for the application of the engine cooling system on fishing boats.

7. REFERENCES

1. Ariafar, 2014, *Ejector Primary Nozzle Steam Condensation : Area Ratio Effects and Mixing Layer Development*. Australia : University of Southern Queensland.
2. Hafiz. Dian, 2011, *Analisa Pengaruh Aliran Fluida Yang Ditimbulkan Oleh Gerakan Putaran Propeller Pada Kapal Ikan Terhadap Tekanan Propeller Dengan Pendekatan.* Semarang : Universitas Diponegoro
3. Harinaldi. Budiarso, 2015. *Sistem Fluida*. Jakarta : Penerbit Erlangga
4. Kondo. Yan, 2012, *Analisis Investigasi Pada Industri Pengecoran Propeller Kapal.* Makassar : Politeknik Negeri Ujung Pandang
5. Martinez, 2016. *Nozzles*. Mexico : University of Nuevo Leon

6. Mustafa. Guducu, 2015, *CFD Analysis Of Nozzle Effect On Jet Formation.*Swedia : Royal Institute of Technology.
7. Triatmodjo. Bambang, 2014. *Hidraulika*. Yogyakarta : Beta Offset.
8. Triyanti. Irmiyana, 2015, *Analisa Pengaruh Bentuk Foil Section Nozzle Terhadap Efisiensi Propulsi Kapal Pada Kapal Tunda.* Surabaya : Institut Teknologi Sepuluh November.
9. Vahaji, 2015 *Study On The Efficiency of a Concergent-Divergent Two- Phase Nozzle as a Motive Force for Power Generation from Low Temperature Geothermal Resource*. Australia : Proceeding World Geothermal Congress.
10. White F.M. 1991, *Fluid Mechanics 2th Edition*. New York : United States of America
11. Syahrun, 2018, *Studi Bentuk Dan Posisi Optimal Nosel Inlet Pada Pemanfaatan Aliran Buritan Kapal Sebagai Sumber Air Pendingin Mesin Utama Kapal*. Makassar : Universitas Hasanuddin.