

muddin_2019_IOP_Conf._Ser._
Mater._Sci._Eng._676_012020.p
df
by

Submission date: 19-Jan-2022 08:30PM (UTC+0700)

Submission ID: 1744113684

File name: muddin_2019_IOP_Conf._Ser._Mater._Sci._Eng._676_012020.pdf (945.23K)

Word count: 3029

Character count: 15245

PAPER · OPEN ACCESS

Performance **Comparison** Between Ellipse and Circular Intake Shapes of Propeller Flow Cooling System (PFCS) with Numerical and Experimental Methods

To cite this article: F Mahmuddin *et al* 2019 *IOP Conf. Ser.: Mater. Sci. Eng.* **676** 012020

¹
View the [article online](#) for updates and enhancements.



ECS **240th ECS Meeting**
Oct 10-14, 2021, Orlando, Florida

**Register early and save
up to 20% on registration costs**

Early registration deadline Sep 13

REGISTER NOW

Performance Comparison Between Ellipse and Circular Intake Shapes of Propeller Flow Cooling System (PFCS) with Numerical and Experimental Methods

F Mahmuddin^{1*}, S Ramadhan¹, A Fitriadhy² and S Klara¹

¹Marine Engineering Department, Engineering Faculty, Hasanuddin University, Indonesia

²Programme of Maritime Technology, School of Ocean Engineering, Universiti Malaysia Terengganu, Malaysia

*Email: f.mahmuddin@gmail.com

Abstract. This research carried out an analysis on the use of high pressure flow behind propeller to cool the ship main engine known as Propeller Flow Cooling System (PFCS). The system is commonly found on relatively small fishing boats at South Sulawesi area of Indonesia. Using the system, the special pump for engine cooling purpose is not needed. The present research conducted an investigation on PFCS intake shape that would be optimal for delivering water to main engine which was arisen from the propeller rotation and to find the difference of water capacity produced from different shapes of intake. The implemented research methods were numerical and experimental methods that analysed the shape of two intake shapes namely, circular and elliptical shapes. In numerical computation, ANSYS CFX software was adopted. The result from numerical computations was an initial prediction to the tendency of the results. In order to verify the results, an experiment in a tank facility known as Circulating Water Channel (CWC) tank was conducted. In the experiment, measurements were performed at 2 (two) different locations which were at the center of propeller and at 0.7R of propeller blade. The results of the research found that the performance of the elliptical intake shape is more optimal compared to the circular one.

1. Introduction

Ship operation needs a large amount of energy which proportional to costs increase and pollution production. Therefore, energy efficiency in ship operation has been a great concern over the last decade [1]. Due to its importance, there are numerous research which have been performed related to ship energy efficiency such as the ones performed by Coraddu et. al [2], Armstrong et. al [3], Trodden et. al [4], Baldi et. al [5], and Mahmuddin [6].

Amongst equipment which consume energy aboard a ship, pumping system consumes approximately almost half of the electrical energy produced on a ship [7]. Thus, increasing the efficiency of pumping system will result in a significant increase of energy efficiency performance of a ship. One of the main pumps used in ship operation is the engine cooling pump. The pump is used to deliver water from open sea to the main engine as well as to discharge them after circulating inside the engine. In order to obtain an efficient engine cooling pumping system, Mrakovcic et. al [8] and Theotokatos et. al [9] performed modelling and simulation of the engine-cooling system. Su et. al [10] also conducted an investigation on a closed pump system for saving energy of marine power system.



Content from this work may be used under the terms of the [Creative Commons Attribution 3.0 licence](https://creativecommons.org/licenses/by/3.0/). Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.

Published under licence by IOP Publishing Ltd

2 (two) main categories of the engine cooling system are sea water system (open system) and central cooling system (using a combination of seawater and freshwater). The number of pumps used in an engine-cooling system depends on the type of the engine-cooling system chosen. However, central cooling-system is usually more complex so it needs more number of pumps compared to the sea water system. Another difference of those systems is that the central cooling system is usually installed in large size ships while small ships or boats usually use sea water or open system.

Especially in South Sulawesi area of Indonesia, the number of fishermen who use small ships or boats for fishing is quite common. These fishing boats usually adopt an open cooling system where water is flowed directly to cool the engine and then discharged outboard. However, in order to save space inside the ship, a special pump for engine cooling purpose is not installed inside the boat. Instead, they utilize the flow in front of the propeller to cool the engine. The system takes advantage of the high pressure flow behind propeller which is directed to circulate inside the engine. After circulating inside the main engine, the water is then discharged outboard. The system will be referred to Propeller Flow Cooling System (PFCS) in the present study. Using this system, not only initial and operational costs can be reduced but also some spaces inside the ship can be saved to store other essential equipment and loads.

Example of a fishing boat which implements the PFCS is shown in Fig. 1.

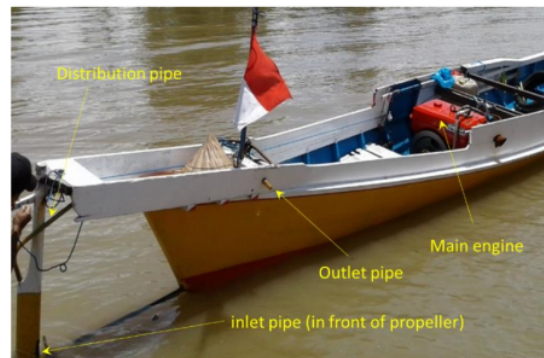


Figure 1. A fishing boat that implements Propeller Flow Cooling System (PFCS)

The performance of the (PFCS) highly depends on the propeller flow pressure. Therefore, it is important to determine and choose the optimal dimension and shape of the system components. As shown in Fig. 1, the main components of the PFCS are intake, distribution pipe, and outtake. The present study will evaluate and compare the performance between circular and elliptical intake shape. The shape of the intake is very significant in determining the flow capacity of the PFCS. In analysis and investigation, numerical methods using ANSYS CFX Software and direct measurement by experiment were performed.

2. Propeller Flow Water Cooling System

The main principle of the Propeller Flow Cooling System (PFCS) is using the pressurized flow in front of propeller to be directed and circulated in the main engine. In order to use the flow, an intake is placed in front of propeller. When the propeller rotates, some amount of water will get into the intake. Due to continuous propeller rotation, water will also continuously stream inside the distribution pipe which is directed to the main engine. The water will then circulate inside and cool the main engine. After the circulation, the hot water is then discharged outboard. The main components and concept of PFCS are illustrated in Fig. 2.

As shown in Fig. 2, the main components of the PFCS are intake, distribution pipe (hot water and cold water pipes), and outtake. In its application, several variations of PFCS can be found to be used

by fishermen on their boats. The variations are mainly on the position and shape of the intake. For the intake shape, the most common adopted shapes are circular and elliptical shapes. Thus, performing an investigation on the performance comparison of these intake shapes in order to determine the best intake shape is necessary.

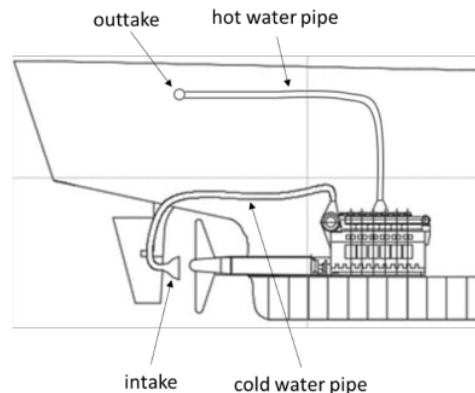


Figure 2. The main components and concept of PFCS

3. Research Methodology

For the analysis purpose in the present study, numerical analysis using software and an experiment in a water tank were conducted. The employed software is Ansys CFX software while the tank experiment was conducted in a tank facility known as Circulating Water Channel (CWC). The tank is located at Marine Structure System Laboratory, Marine Engineering Department, Engineering Faculty, Hasanuddin University, Indonesia.

The concept of the CWC is shown in Fig. 3.

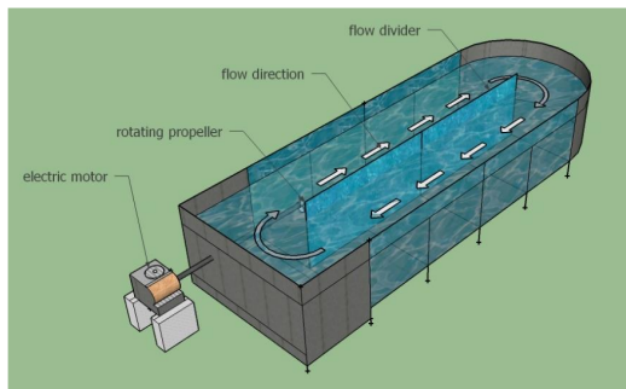


Figure 3. The sketch of CWC

As shown in Fig. 3, CWC is a tank where the water inside the tank can circulate horizontally which enables a convenient data collection for a certain period. The water circulation is made possible due to a rotating propeller connected to an electric motor. The motor is placed outside the tank as shown in Fig. 3. The speed of the flow is adjusted by regulating the speed of the motor (motor RPM) using an Automatic Voltage Regulator (AVR).

3 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

In the present study, the investigated intake was placed in front of the rotating propeller of the CWC to simulate the inlet pipe position of PFCS on a fishing boat described in the previous sections. There are 2 (two) shapes which were investigated which were circular and elliptical shapes. Moreover, there were 2 (two) different vertical positions where measurements were performed which were at the center of propeller (Case 1) and at 0.7 x Radius (0.7R) of propeller blade (Case 2).

More detail about intake shapes and vertical measurement positions are described below.

3.1. Shape description

The sketch and dimension of each shape are shown in Fig. 4 and Table 2.

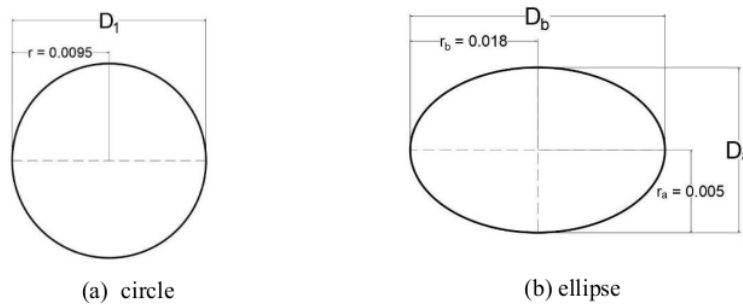


Figure 4. Shape dimension

Table 2. Dimension of shapes

Parameter	Intake Shape	
	Circle	Ellipse
Diameter	$D_1 = 0.019 \text{ m}$	$D_a = 0.01 \text{ m}, D_b = 0.036 \text{ m}$
Surface area		0.00028 m^2
Horizontal distance to propeller		0.1 m

3.2. Vertical position

In order to verify the tendency, beside at the center of propeller, the data collection using these shapes also was conducted for the case when the intake was placed at location of 0.7R of the propeller vertically. The sketch of the location where the intake was placed is shown in Fig. 5.

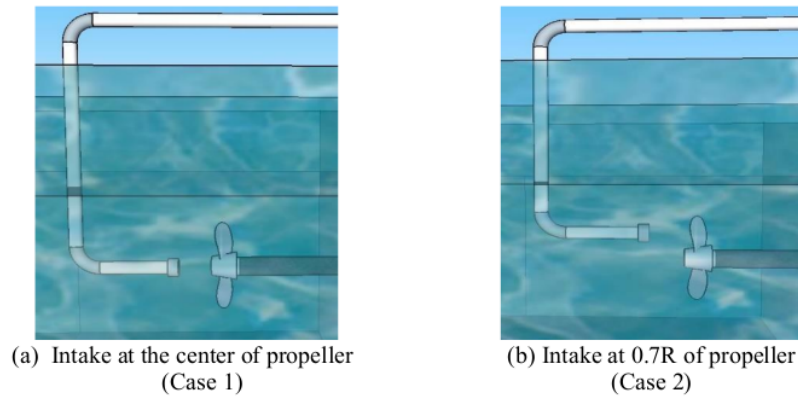


Figure 5. Vertical position of the intake

As shown in Fig. 5, there are 2 (two) cases of vertical intake placement. Case 1 is when the intake was placed exactly at the center of the propeller while Case 2 is when the intake was placed at 0.7R of propeller. The horizontal distance of intake to propeller in each case was same which equals to 0.1 m as described in Table 2.

4. Results Analysis and Discussion

In order to compare the performance, in the present study, a numerical simulation and an experiment were performed.

4.1. Ansys CFX simulation

Before conducting the experiment, a numerical analysis using ANSYS CFX software was performed. The main aim of the software analysis is to compare the flow velocity between circular and elliptical shapes. It become an initial prediction of the analysis results. However, only the case when the inlet pipe was placed at the center of the propeller (Case 1) which was simulated because the input velocity flow was just assumed for the software simulation. Thus, the second case will obtain the same result tendency.

In this analysis, the fluid source was considered to have a box shape. The domain of the intake and flow source were then meshed using unstructured meshes. The shape and their meshes are shown in Fig. 6.

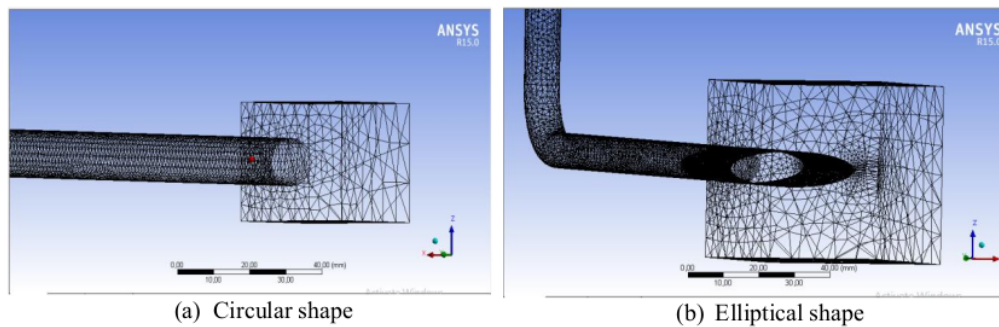


Figure 6. Domain meshing in ANSYS CFX

After the meshing was converged, the computation was run. The computation results are shown in Fig. 7.

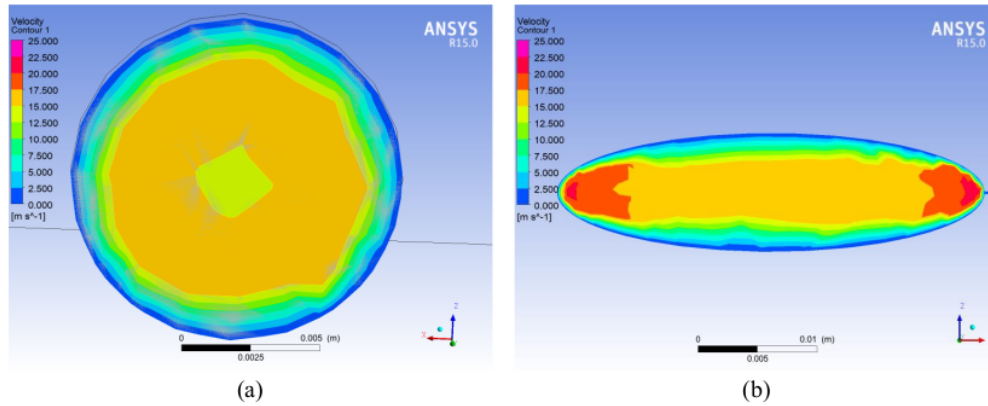


Figure 7. Velocity contour of circle (a) and ellipse (b) intake

The computation results shown in Fig. 7 clearly shows that the flow velocity for the case of elliptical shape is higher than the one in circular shape. The main difference of velocity is mainly around the corner area where the flow velocity is higher on that area. Around the corner area of elliptical shape, the flow velocity could reach 23 m/s while on circular shape only reached 18 m/s. The obtained results are due to flow separation on the corner area which makes the flow could pass through with higher velocity.

4.2. Experimental results

Based on the experimental setup described in the previous section, several direct measurements were conducted. The main measured parameters in the experiment were motor shaft revolution (motor RPM) and water capacity (Q). The first data collection was conducted for the case when the intake was placed in front of the center of propeller as shown in Fig. 5(a). The experimental results for this case are shown in Fig. 8.

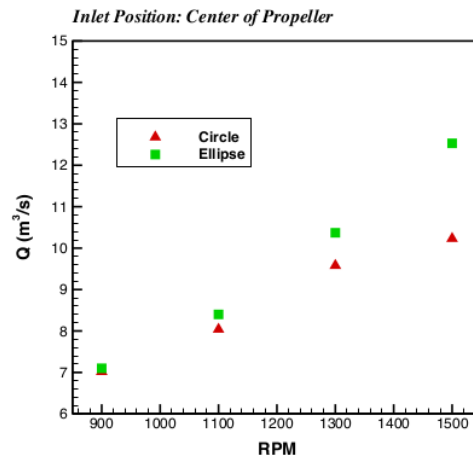


Figure 8. Flow capacity of circle and ellipse shape at the center of propeller position

In Fig. 8, the horizontal axis is motor revolution (motor RPM) while the vertical axis is the water flow capacity (Q) in m^3/s . The figure compares the flow capacity over a certain range of rpm for circular and elliptical shapes. The measured rpm is 900, 1100, 1300, and 1500. It can be seen from the figure that flow capacity is almost same for rpm equals to 900. However, the flow capacity for elliptical shape case is higher as the rpm become higher.

Moreover, to verify the tendency at a different location, the measurement was also performed at the location of 0.7R of propeller blade as illustrated in Fig. 5(b). The measurement results are shown in Fig. 9.

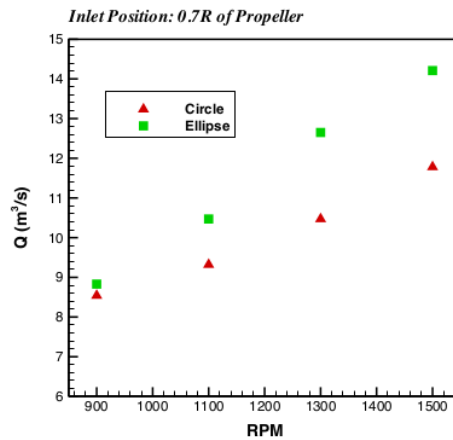


Figure 9. Flow capacity of circular and elliptical intake shapes at 0.7R of propeller blade position

From Fig. 9, the same tendency can be observed where elliptical intake shape has a better performance. The maximum flow velocity in circular shape is $11.8 m^3/s$ while in elliptical is $14.2 m^3/s$. Moreover, it can also be seen that the flow capacity difference between circular and elliptical shapes are larger than the one in the previous case. As a result, a higher maximum flow capacity can be observed for this case.

5. Conclusion

A numerical analysis and an experiment have been conducted to investigate the performance of 2 (two) intake shapes which are circular and elliptical shapes. The main measured parameters in the experiment were motor rotation and intake flow capacity. Measurements at two different positions of the intake shapes were also conducted which were when the intake at the center of propeller and when the intake is at 0.7R of propeller blade position. It was found that elliptical shape has a higher flow rate which can be attributed to the flow separation around the corner area of the elliptical shape. Moreover, it was also found the measurement at 0.7R of propeller blade position has higher flow capacity compared to when the intake was placed at the center of the propeller.

Acknowledgement

The authors would like to express gratitude to JICA C-BEST project of Engineering Faculty, Hasanuddin University for funding this research. The authors also express gratitude to Professor Jun Ando from Kyushu University for the fruitful discussion and supervision of the project.

References

- [1] E. K. Boulougouris, A. D. Papanikolaou, and A. Pavlou, "Energy efficiency parametric design tool in the framework of holistic ship design optimization," *Proc. Inst. Mech. Eng. Part M J. Eng. Marit. Environ.*, vol. 225, no. 3, pp. 242–260, 2011.
- [2] A. Coraddu, M. Figari, and S. Savio, "Numerical investigation on ship energy efficiency by Monte Carlo simulation," *Proc. Inst. Mech. Eng. Part M J. Eng. Marit. Environ.*, vol. 228, no. 3, pp. 220–234, 2014.
- [3] V. N. Armstrong and C. Banks, "Integrated approach to vessel energy efficiency," *Ocean Eng.*, vol. 110, pp. 39–48, 2015.
- [4] D. G. Trodden, A. J. Murphy, K. Pazouki, and J. Sargeant, "Fuel usage data analysis for efficient shipping operations," *Ocean Eng.*, vol. 110, pp. 75–84, 2015.
- [5] F. Baldi, H. Johnson, C. Gabriellii, and K. Andersson, "Energy and exergy analysis of ship energy systems - The case study of a chemical tanker," *Int. J. Thermodyn.*, vol. 18, no. 2, pp. 82–93, 2015.
- [6] Mahmuddin, F., "Analysis of Wind Energy Potential with a Mobile Floating Structure around Sulawesi and Maluku Islands of Indonesia", Proceeding of the 34th International Conference on Ocean, Offshore, and Arctic Engineering (OMAEO), St. John's, Canada, Vol. 9, 2015.
- [7] G. Kocak and Y. Durmusoglu, "Energy efficiency analysis of a ship's central cooling system using variable speed pump," *J. Mar. Eng. Technol.*, vol. 17, no. 1, pp. 43–51, 2018.
- [8] V. Medica, "Numeri ~ no modeliranje hladilnega sistema na motorju z notranjim zgorevanjem," vol. 50, pp. 104–114, 2004.
- [9] G. Theotokatos, K. Sfakianakis, and D. Vassalos, "Investigation of ship cooling system operation for improving energy efficiency," *J. Mar. Sci. Technol.*, vol. 22, no. 1, pp. 38–50, 2017.
- [10] C. L. Su, C. H. Liao, T. C. Chou, M. H. Chou, and J. M. Guerrero, "Variable flow controls of closed system pumps for energy savings in maritime power systems," *IEEE Ind. Appl. Soc. 52nd Annu. Meet. IAS 2016*, pp. 1–8, 2016.

ORIGINALITY REPORT

7%

SIMILARITY INDEX

%

INTERNET SOURCES

7%

PUBLICATIONS

%

STUDENT PAPERS

PRIMARY SOURCES

- 1** Zhigalov Maxim Viktorovich, Apryskin Victor Alexandrovich. "Comparison of the results of solving the problem of nano-beams contact interaction according to different theories of size-dependent behavior", IOP Conference Series: Materials Science and Engineering, 2020
Publication **3%**
- 2** Francesco Baldi, Stefano Moret, Kari Tammi, François Maréchal. "The role of solid oxide fuel cells in future ship energy systems", Energy, 2020
Publication **3%**
- 3** "Practical Design of Ships and Other Floating Structures", Springer Science and Business Media LLC, 2021
Publication **1%**
- 4** Heiu-Jou Shaw, Cheng-Kuan Lin. "Marine big data analysis of ships for the energy efficiency changes of the hull and maintenance **<1%**

evaluation based on the ISO 19030 standard",
Ocean Engineering, 2021

Publication

5

F Mahmuddin, N Puspitasari, M U Pawara.
"Economic Feasibility of a Small-Scale Wind
Turbine in Coastal Area of South Sulawesi",
IOP Conference Series: Earth and
Environmental Science, 2021

Publication

<1 %

Exclude quotes On

Exclude matches < 5 words

Exclude bibliography On