Journal of Advanced Research in
Numerical Heat Transfer

Journal homepage:
https://semarakilmu.com.my/journals/index.php/arnht/index
ISSN: 2735-0142

Water Inlet Behaviour in Engine Colling Piping on Indonesian Traditional Wooden Vessel Using CFD Simulation

Andi Mursid Nugraha Arifuddin[1,*], Wira Setiawan[1], Septian Tommy Pamungkas[1], Muhammad Rusydi Alwi[2], Suandar Baso[3], Mohammad Rizal Firmansyah[3]

[1] Department of Naval Architecture, Institut Teknologi Kalimantan, Jalan Soekarno Hatta KM. 15 Karang Joang, Balikpapan, Indonesia
[2] Department of Marine Engineering, Faculty of Engineering, Hasanuddin University, Jalan Poros Malino, Gowa, Indonesia
[3] Department of Naval Architecture, Faculty of Engineering, Hasanuddin University, Jalan Poros Malino, Gowa, Indonesia

ARTICLE INFO

Article history:
Received 15 July 2024
Received in revised form 16 August 2024
Accepted 17 September 2024
Available online 30 October 2024

Keywords:
CFD; Cooling Piping; Pressure; Velocity; Wooden Vessel

ABSTRACT

Traditional wooden boat builders in East Kalimantan make the main engine seawater cooling system by using the water currents created by the movement of the ship without the use of suction pumps. This system has been applied to several wooden ships of various sizes. However, the size of the water flow in the pipe is not known by the builders at each ship's speed. Thus, in this paper, an investigation is carried out on seawater cooling pipes on the behavior of fluids moving in the pipes. This study aims to determine the effect of ship speed on the flow of water that occurs in the cooling pipe (inlet and outlet water). The approach taken in this paper is a computer simulation based on Computational Fluid Dynamics (CFD) by analyzing the current flow (Va) of the water discharge in the cooling pipes at speeds of 1 - 9 knots. CFD can show very detailed results in analyzing fluid flow parameters in pipes. Based on the simulations performed, the average flow velocity has increased by around 14.73% for every 1 knot increase in speed. Meanwhile, the average flow rate will increase on the pipe by 14.78% for every 1 knot increase in speed. For the average pressure obtained, the vertical pipe will increase 25% for every 1 knot increase and the horizontal pipe shell will increase 19.86% for every 1 knot increase. Meanwhile, the recommended minimum ship speed is 2.5 knots to get the required cooling seawater flow rate.

1. Introduction

The use of sea water as an engine coolant on traditional ships is very widespread in Kalimantan. Cooling systems like this are used by fishermen because they are cheap and easy to install on their fleet of boats. Uniquely, an engine cooling system like this does not use a pump to suck water into a storage tank or engine. The inlet pipe is only installed behind the propeller where the pipe mouth faces the Forepeak (FP) to catch sea water when the ship moves forward which is then channeled to the main engine as in Figure 1. The sea water cooling system is used to suppress excessive use of fresh water and radiator water to maintain engine temperature when operating. To find out the

[*] *Corresponding author.*
E-mail address: andi.mursid@lecturer.itk.ac.id (Andi Mursid Nugraha Arifuddin)

https://doi.org/10.37934/arnht.25.1.7386

behavior of water flow in seawater cooling pipes, analysis was carried out using Computational Fluid Dynamics (CFD) simulation based on the Finite Element Method (FEM). In its application, CFD uses sophisticated computers and applied mathematics to model fluid flow situations [1].

Water cooling systems are critical to the efficient functioning of internal combustion engines, particularly in marine environments where ships rely heavily on these systems to remove excess heat generated during operation. Traditional wooden ships, which represent a unique part of maritime heritage, often use conventional engines for propulsion. However, the integration of modern engine cooling systems into these vessels requires careful consideration to ensure optimal performance and longevity. Traditional wooden ships have different structural and material properties that differentiate them from modern ships, influencing the design and implementation of modern engine cooling systems. Engine cooling is very important to prevent overheating and maintain engine efficiency and durability. The water inlet system, which is an important component of the cooling system, plays a vital role in facilitating effective coolant flow throughout the engine. Understanding the behavior of these inlet systems, especially in the context of traditional wooden vessels, is critical to designing efficient cooling solutions tailored to the unique characteristics of the vessel.

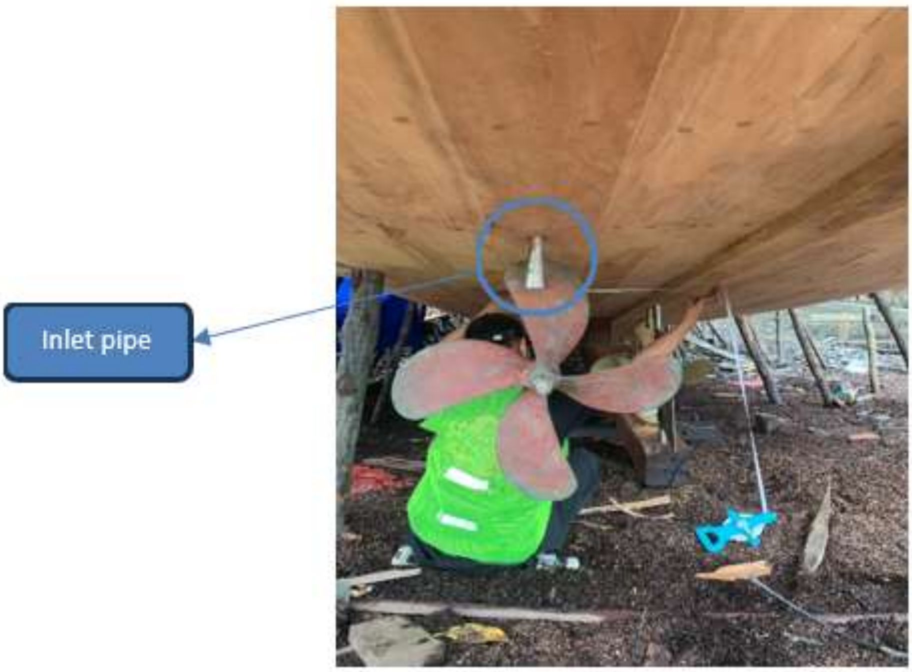

Fig. 1. Inlet pipe at the stern of a traditional wooden ship

The research written in this paper aims to look at fluid phenomena that occur in the main engine cooling pipe. The object of research is how the speed of the ship influences the speed of sea water flow that occurs around the ship's hull, the speed of sea water flow in pipes, the pressure value produced by sea water flow in pipes, the value of the Reynold number to determine the form of flow that occurs. in the pipe and the sea water discharge that occurs in the pipe for every increase in ship speed. The CFD method was chosen in this research because this method is considered very suitable for measuring fluid flow parameters. CFD can replace physical testing for homologous curve measurements and can be applied to low tide and flow velocity as well as rotational speed testing [2]. CFD can also shorten time and reduce expensive costs for each experiment, especially for industrial cases [3]. In the case of fluid flow analysis, CFD analysis is carried out to realize the phenomena caused by turbulent penetration and valve leakage and temperature distribution throughout the pipe [4]. With the reliability of this method, CFD Simulation has been widely applied in the field of process safety and loss prevention to gain new insights, improve existing models, and assess new hazardous scenarios [5].

Several studies related to the results of fluid movement analysis using CFD have been carried out by engineering researchers. Ferng (2008) carried out an analysis on pipes with various forms of connections to analyze the liquid fraction. The results show quite good flow parameter values [6]. Halanger *et al.,* [7] carried out an analysis of pipes that distribute gas to offshore buildings and produced parameter values for each modeled pipe line. Lin *et al.,* [8] analyzed the corrosion rate of pipes by first looking for hydrodynamic parameters using CFD. Analysis shows parameter values at 2 different pipe diameters. Schleder *et al.,* [9] modeled the pipeline route from the tank to the release point to distribute propane gas, the results showed clear parameter values and were used as a reference for the next research stage. Khan *et al.,* [10] did testing influence comparison CD Nozzel area area using CFD, the results show parameter values are almost the same with experiment.

Based on these various reviews, further analysis of the hydrodynamic parameters of the main engine cooling pipes for traditional wooden ships will be attempted using computer simulation. This traditional wooden ship uses sea water as a cooling agent. The analysis was carried out by applying the CFD method by carrying out a previous analytical approach, to obtain the values needed when carrying out computer simulations. CFDs are expected capable catch parameter values that occur in sea water flow in the pipe. Because CFDs are suitable For calculate and analyze complex and difficult system with manual calculations [11]. Additionally , on testing our experimental only can see element input and output fluid but in all CFD simulations element available [12]. So, in research expected Maximum data capture of sea water flow parameters in the pipe can visible and known.

2. Methodology

2.1 Ship Data

The principal main dimension data obtained from direct data collection at the Penajam Traditional Shipyard is described in Table 1. Meanwhile, the shape of the traditional wooden ship hull which was used as the research object can be seen in Figure 2.

Table 1
Principal main dimension

LoA	:	21.00	m
Lpp	:	18.80	m
Breath	:	4.00	m
Depth	:	2.00	m
Draugth	:	1.45	m
Vs	:	9.00	Knot
Engine Power	:	30.00	Hp

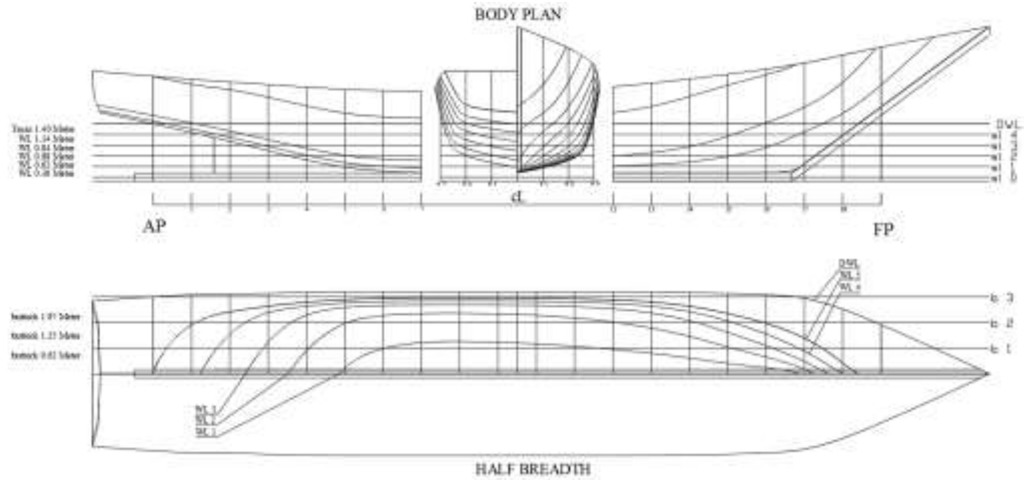

Fig. 2. Lines plan for traditional wooden ships

The main engine cooling system on this ship uses sea water. Sea water is not sucked in using a pump, but uses the water flow created by the movement of the ship and the push of the propeller at the stern of the ship. So, to get the value of the flow velocity entering the cooling pipe using the flow velocity equation due to the movement of the ship. An illustration of the main engine cooling pipe route can be seen in Figure 3.

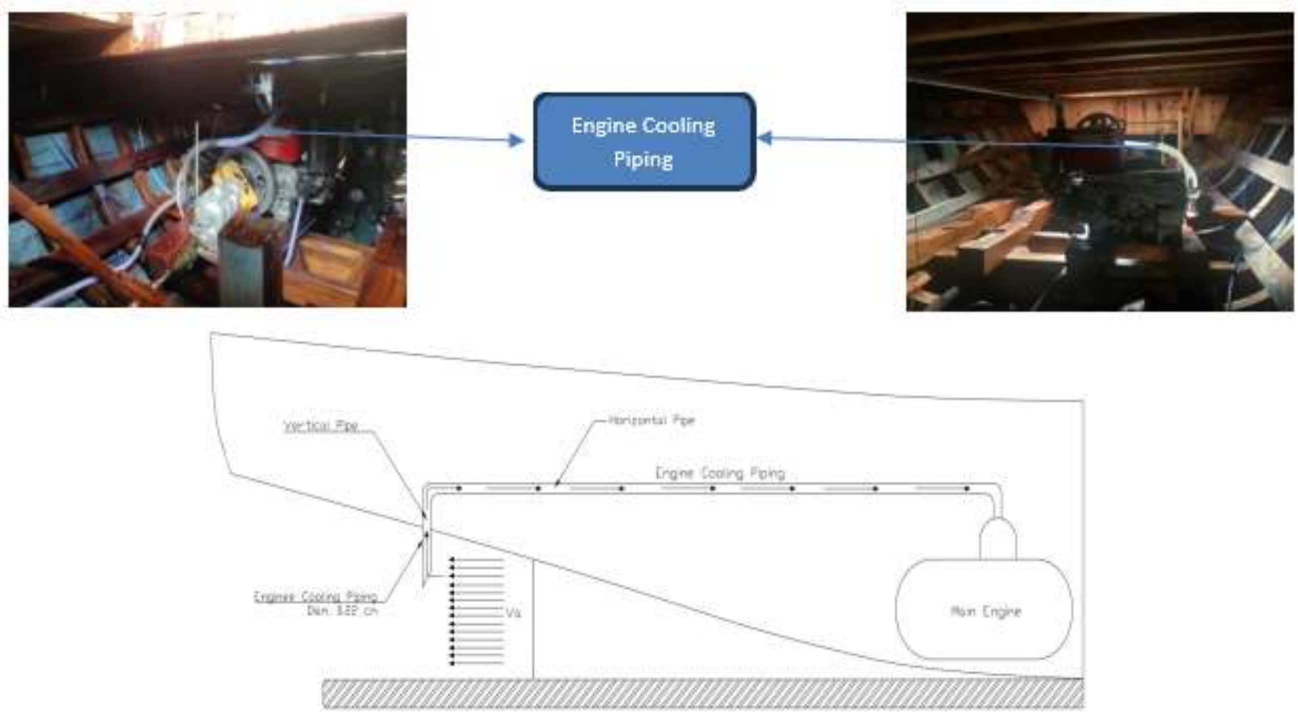

Fig. 3. Engine cooling piping system scenario

2.2 Modeling

The modeling carried out in this case uses the Ansys Fluent application which adopts the solver fluid flow analysis (CFX) numerical simulation method based on Computational Fluid Dynamics (CFD). At this stage, the pipe modeling for the ship's main engine cooling system is made based on pipe size data used on traditional ships as seen in Figure 4.

Fig. 4. Modeling of sea water cooling pipe

To facilitate the pipe analysis process, the pipe object is divided into 2 parts, namely vertical pipes and horizontal pipes as shown in Figure 5. Vertical pipes are pipes that are immersed in water and are installed through the hull at the stern of the ship. Meanwhile, the horizontal pipe is a pipe that is installed longitudinally from the stern to the main engine position.

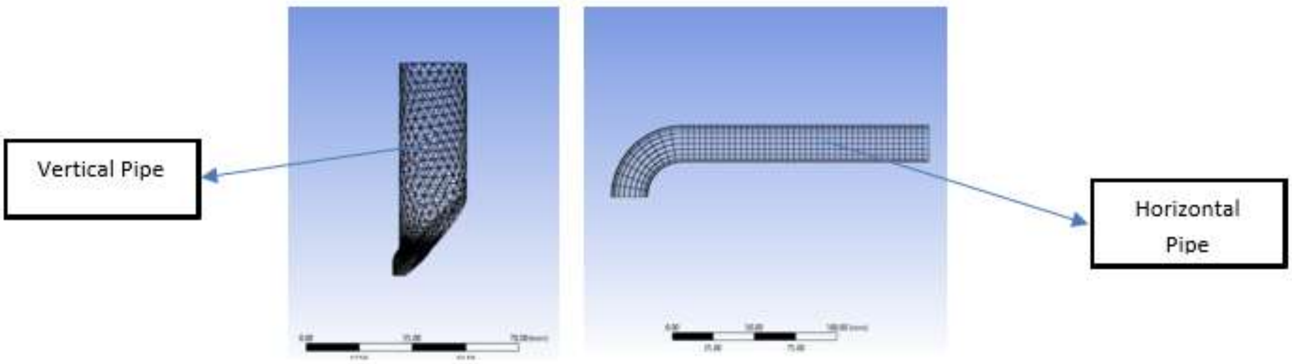

Fig. 5. Meshing application on cooling pipes

2.3 Speed of Advance (Va)

The presence of the ship's hull in front of the propeller changes the local average speed of the propeller. If the ship moves at speed V and the acceleration of the water in the propeller section will move less than the speed of the ship. The acceleration of the water moves at a speed Va, known as the Speed of Advance. The equations used are as follows Eq. (1) [13] :

$$Va = Vs\,(1 - WT) \tag{1}$$

where the wake fraction, WT , average speed of flow into the propeller, Va, and speed of advance of the hull, Vs.

2.4 Fluid Flow

Fluid flow is the form of a substance in the liquid phase and gas phase. Liquids will flow by themselves from higher places to lower places or from higher pressure to lower pressure. The amount of liquid that flows through the flow cross section per unit of time is called flow and is given the notation Q. Flow discharge is usually measured in the volume of liquid per unit of time, so the unit is cubic meters per second (m^3/s). In an ideal fluid, where there is no friction, the flow velocity V is the same at every point on the cross section. If the flow surface is perpendicular to the flow direction, the flow rate can be formulated using the following equation [14] :

$$Q = A1.V1 \quad (2)$$

where,
Q = Flow rate (m^3/s)
$A1$ = Field cross-sectional area (m^2)
$V1$ = Flow speed (m/s)

2.5 Reynolds Number (Re)

For help with the determination process type Genre the fluid that occurs in the seawater cooling pipe is required explanation related type Genre based on Re number of fluids flowing in the cooling pipe. As for the types Genre explained as following:

i) Laminar flow occurs when fluid particles move in parallel layers along the flow or parallel paths, which means there is no countercurrent. This laminar flow has a Reynolds number value of less than 2300 (Re < 2300).
ii) Turbulent flow occurs when fluid particles move at a speed and direction that changes over time so that it is difficult to observe, which means that a turbulent flow occurs. The value of the Renolds number is greater than 4000 (Re > 4000).
iii) Transition flow is a transition flow from laminar flow to turbulent flow. This transition state depends on fluid viscosity, speed and other things related to flow geometry where the Reynolds number value is between 2300 to 4000 (2300 < Re < 4000).

The equations used For get number Reynold Number as follows [15] :

$$\mathrm{Re} = \frac{Vm.\ D\ .\rho 1}{\mu 1} \quad (3)$$

where,
Re = Reynolds Number
$\rho 1$ = density (kg/m^3)
Vm = Average speed of flowing fluid (m/s)
D = inner diameter of the pipe (m)
$\mu 1$ = fluid dynamic viscosity (kg/ms)

2.6 Computational Fluid Dynamics

Computational Fluid Dynamics (CFD), is a branch of fluid mechanics that uses numerical methods and algorithms to calculate fluid flow modeling and mass transfer mechanisms. The advantages of CFD modeling are that it is able to model physical phenomena such as evaporation-condensation and surface tension, providing very accurate simulations of fluid flow, heat transfer and chemical reactions [16]. The Computational Fluid Dynamics (CFD) method, which is widely used in solving fluid-structure interaction problems, allows for economical, fast, and easy solutions in open channel flow calculations [17]. Generally, the fluid flow calculation process is solved using the momentum and continuity equations. The equations used are as follows.

Continuity equation [18] :

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0 \quad (4)$$

Continuity equation for incompressible flow [18]:

X direction: $$\rho(\frac{\partial u}{\partial \mathrm{t}}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z})=\rho g_x-\frac{\partial \rho}{\partial \mathrm{x}}+\mu(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}) \quad (5)$$

Y direction: $$\rho(\frac{\partial v}{\partial \mathrm{t}}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z})=\rho g_y-\frac{\partial \rho}{\partial y}+\mu(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}) \quad (6)$$

Z direction: $$\rho(\frac{\partial w}{\partial \mathrm{t}}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z})=\rho g_z-\frac{\partial \rho}{\partial z}+\mu(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}) \quad (7)$$

where u, v, and w are the x, y, and z components of velocity.

In current computer technology, Direct Numerical Simulation (DNS) simulation is not feasible for practical purposes in engineering. Therefore, simplification is needed. Most of the turbulent flow engineering simulations are performed using the Reynolds-Averaged Navier–Stokes (RANS) method [19]. The general equation as follows:

$$\rho\frac{D\bar{u}_i}{Dt} = \rho fi - \frac{\partial \bar{p}}{\partial_{xi}} + \frac{\partial}{\partial x_j}(\mu\frac{\partial \bar{u}_i}{\partial} - \rho\overline{u_i' u_j'}) \quad (8)$$

The Reynolds stress tensor or turbulent stresses, τ, is:

$$\tau ij = -\overline{\rho u_i' u_j'} \quad (9)$$

where u and j with the over bar indicate are two components that experience an increase in the average velocity. The turbulence model used here is the interpretable k-ε model. Among the turbulence models available in ANSYS Fluent software (e.g., k-ε, k-ω, Reynolds stress, SAS, DES, LES) [20]. All k-ε models relate Reynolds stress to the average velocity gradient [20].

$$\frac{\tau_{ij}}{\rho} = -\overline{u_i' u_j'} = 2V_T S_{ij} - \frac{2}{3}kS_{ij} \quad (10)$$

where, the turbulent kinetic energy, k, is defined as

$$k=\frac{1}{2}\overline{u_i' u_j'} \quad (11)$$

where V_T is the eddy kinetic velocity while S_{ij} is described as follows:

$$S_{ij}=\frac{1}{2}\left(\frac{\partial \bar{u}_i}{\partial x_j}+\frac{\partial \bar{u}_j}{\partial x_i}\right) \quad (12)$$

3. Results

CFD simulation of seawater flow in cooling pipes machine main boat wood traditional done using Ansys Fluent software. In simulation this, obtained mark speed Genre in the pipe (V inlet) and the resulting pressure value Genre that. Next, the debit value is calculated use simple using speed data sea water flow in the pipe. Simulation results Genre the fluid used using Ansys Fluent is shown in Figure 6. Next, we will discuss water inlet behavior that occurs as following:

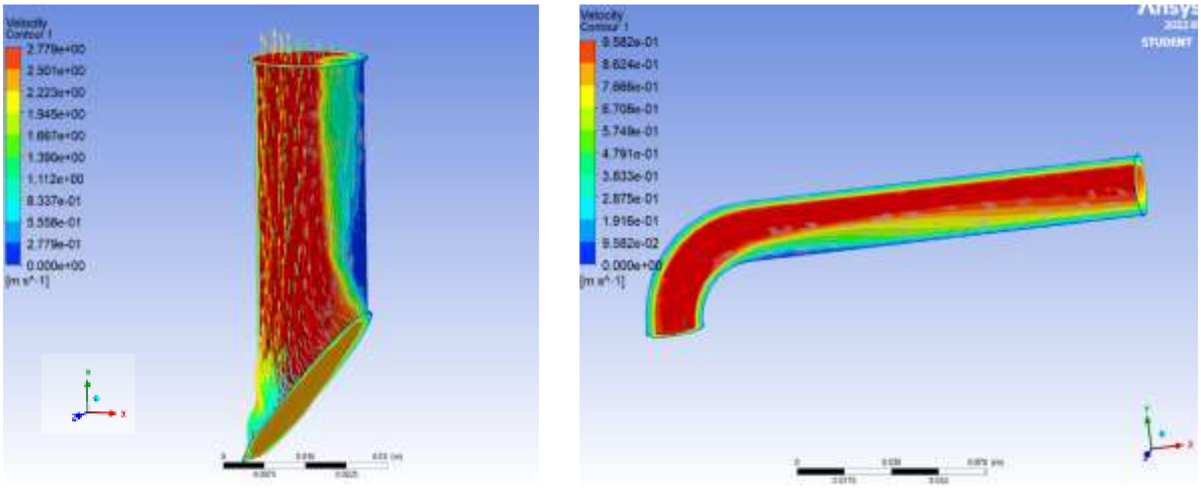

Fig. 6. Seawater flow contours in cooling pipes

3.1 Speed of Advance (Va)

To run a simulation of the flow of cooling water in a pipe, the Va value (flow speed) produced at each ship speed is needed. So, Eq. (1) is used. The results of the flow velocity calculations for each are shown in Table 2.

Table 2
Flow speed (Va) for each ship speed

No	Ship speed		Speed of advance (m/s)
	Knots	m/s	
1	0.5	0.26	0.21
2	1	0.51	0.43
3	1.5	0.77	0.64
4	2	1.03	0.85
5	2.5	1.29	1.07
6	3	1.54	1.28
7	3.5	1.80	1.50
8	4	2.06	1.71
9	4.5	2.32	1.92
10	5	2.57	2.14
11	5.5	2.83	2.35
12	6	3.09	2.56
13	6.5	3.34	2.78
14	7	3.60	2.99
15	7.5	3.86	3.21
16	8	4.12	3.42
17	8.5	4.37	3.63
18	9	4.63	3.85

To see the trend of increasing flow velocity for each speed, the distribution of Va for ship speed is distributed in a graph in Figure 7. In this graph, it can be seen that the increase in the VA value is directly proportional to the increase in speed so that the graph looks linear. From a speed of 1 – 9 knots, an average value of Va increases of 33.80% was obtained for each increase in ship speed of 1 knot.

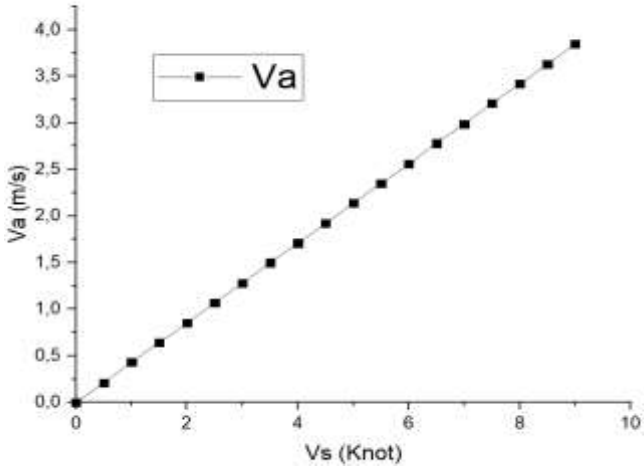

Fig. 7. The relationship between ship speed and water flow speed

The contour shape of the sea water flow that is formed when it hits the engine cooling inlet pipe is shown in Figure 8. In this figure, fluid flow is simulated when the ship is sailing at a speed of 8 knots.

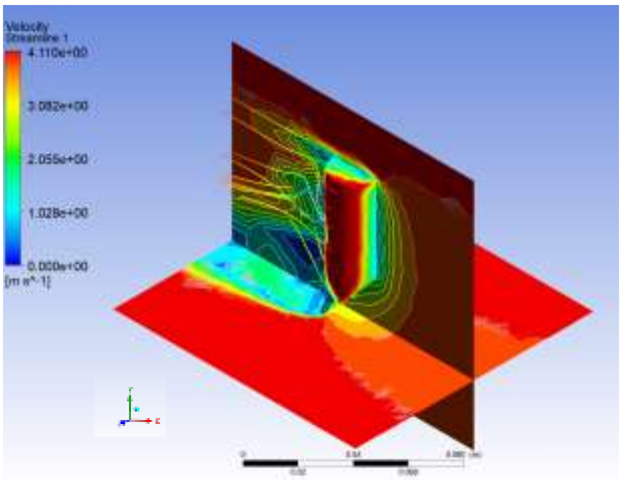

Fig. 8. Simulation of sea water flow at a ship speed of 8 knots

3.2 Reynolds Number

Next, the flow form phenomenon that occurs in the pipe will be known by calculating the Reynolds Number (Rn) value using equation 4. From the simulation results it is shown that there are 2 types of flow that occur in the pipe, both those installed vertically and those installed horizontally. The Rn value in the pipe obtained for each ship speed converted to flow speed can be seen in Table 3.

Table 3
Rn values in vertical and horizontal pipes for each flow speed

No	Va (m/s)	Vertical pipe		Horizontal pipe	
		V inlet (m/s)	Rn	V inlet (m/s)	Rn
1	0.21	0.47	151.03	0.97	309.71
2	0.43	0.96	306.62	1.44	462.22
3	0.64	1.44	460.83	1.93	616.42
4	0.85	1.92	615.77	2.41	771.36
5	1.07	2.41	771.89	2.90	927.48
6	1.28	2.90	928.41	3.39	1084.00
7	1.50	3.39	1084.18	3.87	1239.77
8	1.71	3.87	1239.89	4.36	1395.48
9	1.92	4.35	1393.52	4.84	1549.11
10	2.14	4.85	1550.53	5.33	1706.13
11	2.35	5.33	1706.57	5.82	1862.16
12	2.56	5.82	1863.08	6.31	2018.67
13	2.78	6.31	2019.56	6.80	2175.15
14	2.99	6.80	2176.00	7.29	2331.59
15	3.21	7.29	2332.70	7.78	2488.30
16	3.42	7.78	2489.44	8.27	2645.03
17	3.63	8.27	2646.38	8.76	2801.98
18	3.85	8.76	2803.34	9.25	2958.94

From the results of computer-based numerical experimental simulations, it is shown that the V inlet value that occurs in vertical pipes is lower when compared to the V inlet in horizontal pipes. This is due to the influence of gravity which is in the opposite direction to the flow of water in the vertical pipe. The difference in inlet V values on the two pipes causes the Rn values on the two pipes to be different. In vertical pipes the average increase in Rn value reaches 20.52% and in horizontal pipes the average increase in Rn value reaches 14.70%. A comparison of flow velocities in vertical and horizontal pipes can be seen in Figure 9. At a flow speed of 0 - 2.99 m/s, laminar flow occurs in the vertical pipe. Furthermore, at a flow speed of 3.21 – 3.85 m/s, transition flow occurs in the vertical pipe. Meanwhile, for horizontal pipes at speeds of 0 – 2.78 m/s laminar flow occurs and at speeds of 2.99 – 3.85 transition flows occur.

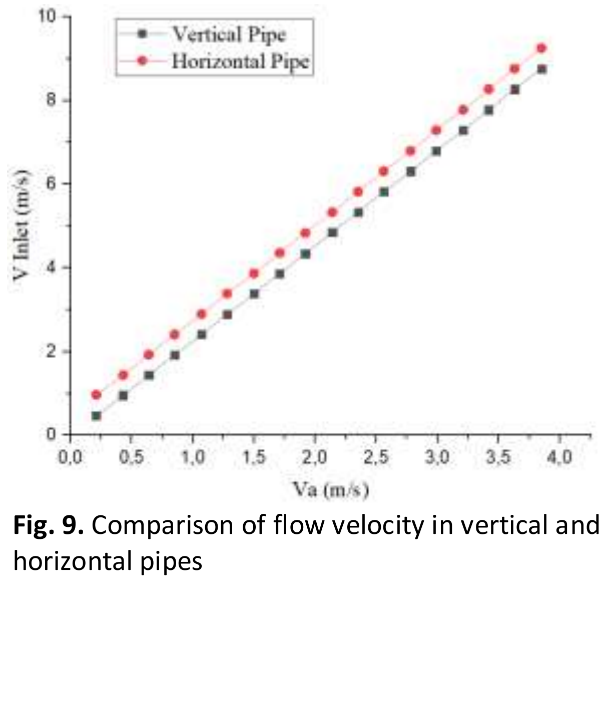

Fig. 9. Comparison of flow velocity in vertical and horizontal pipes

3.3 Debit

Furthermore, based on the known flow velocity in the pipe (V inlet), the character of the flow discharge (Q) that occurs in the pipe can be determined using equation 3. The discharge that occurs in the cooling water pipe is presented in a table as seen in Table 4. The discharge that occurs occurs will be evaluated with the minimum standards required for minimum discharge for cooling water pipes.

Table 4
Discharge in main engine cooling water pipe vs ship speed

No.	Vs (Knots)	Q (m 3 /h)	Minimum standard discharge (2 m 3 /h)
1	0.5	1.32	not analysis
2	1	1.98	not analysis
3	1.5	2.63	pass
4	2	3.30	pass
5	2.5	3.96	pass
6	3	4.63	pass
7	3.5	5.30	pass
8	4	5.97	pass
9	4.5	6.62	pass
10	5	7.29	pass
11	5.5	7.96	pass
12	6	8.63	pass
13	6.5	9.30	pass
14	7	9.97	pass
15	7.5	10.64	pass
16	8	11.31	pass
17	8.5	11.98	pass
18	9	12.65	pass

Based on the results of calculating the discharge in the cooling water pipe, information is obtained that if a ship sailing at a speed of 1 knot does not reach the minimum standard discharge. If the ship sails 1.5 knots to 9 knots, the discharge in the cooling pipe is sufficient to meet the minimum discharge standards required. In other words, when the ship is not moving, the sea water cooling system does not function.

3.4 Pressure

The pressure value displayed is the pressure value exerted by the fluid moving in the pipe. Based on the results of the computational simulation, the maximum pressure value for the vertical pipe and horizontal pipe was obtained. The pressure results that occur are also influenced by the speed of water flow in the pipe. The relationship between ship speed and the pressure value that occurs is shown in Table 5.

Table 5
Pressure values in sea water cooling pipes

No.	Vs (Knots)	Pressure (Pa)	
		Vertical pipe	Horizontal pipe
1	0.5	0.15	0.90
2	1	0.52	1.27
3	1.5	1.12	2.32
4	2	1.94	3.58
5	2.5	2.98	5.06
6	3	4.24	6.76
7	3.5	5.73	8.71
8	4	7.44	10.86
9	4.5	9.37	13.23
10	5	11.52	15.80
11	5.5	13.89	18.64
12	6	16.49	21.69
13	6.5	19.32	24.97
14	7	22.35	28.41
15	7.5	25.62	32.17
16	8	29.11	36.08
17	8.5	32.82	40.26
18	9	36.76	44.64

Based on the CFD simulation results, the pressure that occurs when the ship operates at a speed of 9 knots reaches 44.64 Pa for horizontal pipes and 36.76 Pa for vertical pipes. In general, the pressure value in horizontal pipes is greater when compared to vertical pipes. This is because the flow velocity in the pipe is also different, where the value in the horizontal pipe is greater. Next, to see the comparison level of pressure values in the two pipes, it is shown in the graph in Figure 10 below:

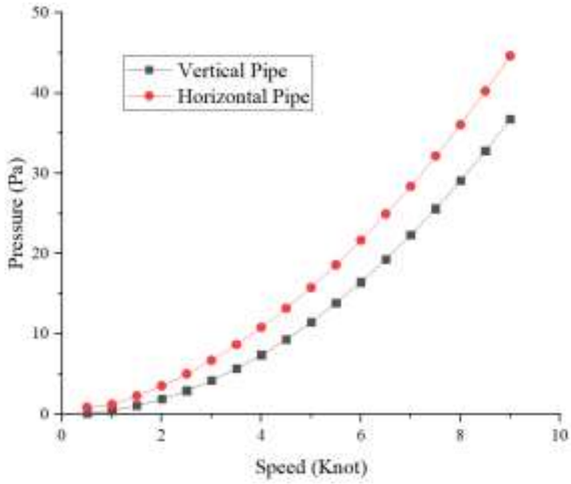

Fig. 10. Relationship between pressure vs ship speed

4. Conclusions

Analysis Genre fluid (sea water) in the cooling pipe machine main Wooden ship traditional has succeed done. The use of the cooling system applied is very unique Because For sucking sea water to machine main No use pump special. However, water passed distributed with utilise the flow of sea water created consequence movement ship (ship speed). From the results CFD based analysis computer obtained enhancement the average speed of pipe flow is 14.73% in vertical pipes and 14.78% in horizontal pipes for every increase speed ship 1 knot. Meanwhile, value enhancement speed the average sea water flow that occurs namely 19.86% for every increase speed 1 knot. For horizontal pipes get enhancement the average pressure value is 25% for every enhancement speed 1 knot. A must Pay attention to the use of the cooling system like This that is when speed boat low, then the cooling system No too works with good. Recommended boat operated at a minimum speed of 2.5 knots.

References

[1] Xia, Bin, and Da-Wen Sun. "Applications of computational fluid dynamics (CFD) in the food industry: a review." *Computers and electronics in agriculture* 34, no. 1-3 (2002): 5-24. https://doi.org/10.1016/S0168-1699(01)00177-6
[2] Jung, Jaeho, Byeonggeon Bae, and Je Yong Yu. "Homologous curve generation for reactor coolant pump of small modular reactor by testing and CFD analysis." *Nuclear Engineering and Design* 400 (2022): 112049. https://doi.org/10.1016/j.nucengdes.2022.112049
[3] Mirzaei, Mohamadali, Sønnik Clausen, Hao Wu, Mohammadhadi Nakhaei, Haosheng Zhou, Kasper Jønck, Peter Arendt Jensen, and Weigang Lin. "Investigation of erosion in an industrial cyclone preheater by CFD simulations." *Powder Technology* 421 (2023): 118424. https://doi.org/10.1016/j.powtec.2023.118424
[4] Kim, Sun-Hye, Jae-Boong Choi, Jung-Soon Park, Young-Hwan Choi, and Jin-Ho Lee. "A coupled CFD-FEM analysis on the safety injection piping subjected to thermal stratification." *Nuclear Engineering and Technology* 45, no. 2 (2013): 237-248. https://doi.org/10.5516/NET.09.2012.038
[5] Shen, Ruiqing, Zeren Jiao, Trent Parker, Yue Sun, and Qingsheng Wang. "Recent application of Computational Fluid Dynamics (CFD) in process safety and loss prevention: A review." *Journal of Loss Prevention in the Process Industries* 67 (2020): 104252. https://doi.org/10.1016/j.jlp.2020.104252
[6] Ferng, Y. M. "Predicting local distributions of erosion–corrosion wear sites for the piping in the nuclear power plant using CFD models." *Annals of Nuclear Energy* 35, no. 2 (2008): 304-313. https://doi.org/10.1016/j.anucene.2007.06.010
[7] Hallanger, Anders, Camilla Saetre, and Kjell-Eivind Frøysa. "Flow profile effects due to pipe geometry in an export gas metering station–Analysis by CFD simulations." *Flow Measurement and Instrumentation* 61 (2018): 56-65. https://doi.org/10.1016/j.flowmeasinst.2018.03.015
[8] Lin, Chih Hung, and Yuh Ming Ferng. "Predictions of hydrodynamic characteristics and corrosion rates using CFD in the piping systems of pressurized-water reactor power plant." *Annals of Nuclear Energy* 65 (2014): 214-222. https://doi.org/10.1016/j.anucene.2013.11.007
[9] Schleder, Adriana Miralles, Elena Pastor, E. Planas, and Marcelo Ramos Martins. "Experimental data and CFD performance for cloud dispersion analysis: The USP-UPC project." *Journal of Loss Prevention in the Process Industries* 38 (2015): 125-138. https://doi.org/10.1016/j.jlp.2015.09.003
[10] Khan, Sher Afghan, Abdul Aabid, Fharukh Ahmed Mehaboobali Ghasi, Abdulrahman Abdullah Al-Robaian, and Ali Sulaiman Alsagri. "Analysis of area ratio in a CD nozzle with suddenly expanded duct using CFD method." *CFD Letters* 11, no. 5 (2019): 61-71.
[11] Sukamta, Sukamta. "Computational fluid dynamics (CFD) and experimental study of two-phase flow patterns gas-liquid with low viscosity in a horizontal capillary pipe." (2021).
[12] Ghiasi, Pedram, Amar Salehi, Seyed Salar Hoseini, Gholamhassan Najafi, Rizalman Mamat, Balkhaya Balkhaya, and Fitri Khoerunnisa. "Investigation of the effect of flow rate on fluid heat transfer in counter-flow helical heat exchanger using CFD method." *CFD Letters* 12, no. 3 (2020): 98-111. https://doi.org/10.37934/cfdl.12.3.98111
[13] Molland, Anthony F. *Ship resistance and propulsion*. Cambridge university press, 2017. https://doi.org/10.1017/9781316494196
[14] Ghurri, A. *Dasar-Dasar Mekanika Fluida*. 2014.
[15] Zambrano, Héctor, Leonardo Di G. Sigalotti, Jaime Klapp, Franklin Peña-Polo, and Alfonso Bencomo. "Heavy oil slurry transportation through horizontal pipelines: Experiments and CFD simulations." *International Journal of Multiphase Flow* 91 (2017): 130-141. https://doi.org/10.1016/j.ijmultiphaseflow.2016.04.013
[16] Kumar, Aditya. "ANSYS Fluent-CFD analysis of a continuous single-slope single-basin type solar still." *Green Technologies and Sustainability* (2024): 100105. https://doi.org/10.1016/j.grets.2024.100105
[17] Soydan Oksal, N. Goksu, M. Sami Akoz, and Oguz Simsek. "Numerical modelling of trapezoidal weir flow with RANS, LES and DES models." *Sādhanā* 45, no. 1 (2020): 91. https://doi.org/10.1007/s12046-020-01332-2

[18] Munson, Bruce R., Alric P. Rothmayer, and Theodore H. Okiishi. *Fundamentals of fluid mechanics*. Wiley Global Education, 2012.

[19] Martins, Nuno MC, Nelson JG Carrico, Helena M. Ramos, and Didia IC Covas. "Velocity-distribution in pressurized pipe flow using CFD: Accuracy and mesh analysis." *Computers & Fluids* 105 (2014): 218-230. https://doi.org/10.1016/j.compfluid.2014.09.031

[20] Martins, Nuno MC, Alexandre K. Soares, Helena M. Ramos, and Didia IC Covas. "CFD modeling of transient flow in pressurized pipes." *Computers & Fluids* 126 (2016): 129-140. https://doi.org/10.1016/j.compfluid.2015.12.002