



Computational Fluid Dynamics Modelling of the Temperature Distribution in Power Plant's Cooling Canal

Rohmah Iftitah Sa'idatul Izzah¹, Harmin Sulistyanning Titah^{2*}, Shade Rahmawati³

^{1,2,3}Sepuluh Nopember Institute of Technology, Surabaya, West Java, Indonesia

Emails: Rohmahiftitah@gmail.com¹, harmin_st@its.ac.id^{2*}

ABSTRACT

Seawater used as a heat exchanger in the coal-fired power plant condenser system has a higher water temperature than the natural temperature of seawater. There was an increase in the temperature of the waters around the coal-fired power plant due to the discharge of boiling water reaching 6.50C. According to the regulations in force in Indonesia, the hot water temperature at the water discharge outlet should not be more than 20C of the natural temperature of seawater. The purpose of this study is to predict the pattern of temperature decrease in the temperature of the boiling water along the cooling line. The method used to predict temperature drop patterns along the cooling line uses a 3-dimensional CFD (Computational Fluid Dynamics) modelling approach with the Ansys Fluent 2023 RI application. The K-Epsilon turbulence model was chosen to describe the effect of flow turbulence on temperature changes. The modelling results were validated by comparing the actual measurements of the water temperature field data representing along the cooling line. The results showed that the performance of reducing the temperature of the water increased when passing through the cascade aeration downstream of the channel. The flow of water along the channel tends to be on the lower side, and downstream, the flow is on the left-middle side. The error calculation using MAPE resulted in 9.49%.

Keywords: Air Bahang, CFD, Modeling, Temperature, Cooling Line.

INTRODUCTION

Seawater used as a heat exchanger in the coal-fired power plant condenser system has a higher water temperature than the natural temperature of seawater. There was an increase in the temperature of the waters around the coal-fired power plant due to the discharge of hot water reaching 6.5°C. According to the regulations in force in Indonesia, the temperature of hot water at the outlet water discharge should not be more than 2°C of the natural temperature of seawater. Thermal power plants are generally located near surface water bodies, as economical

water sources are available for cooling systems (Chiasson, 2016). Power plants with open cycles have a high level of water use, about 40 (forty) times more than closed-cycle systems. This causes power plants with open-cycle cooling systems to be more sensitive to water supply and an increased risk of environmental damage (Lee et al., 2020).

Thermal pollution can be caused by hot or cold-water discharge, and the rate and rate of temperature change that deviates from normal conditions are important factors affecting marine organisms (Baag & Mandal, 2022). A common cause of thermal pollution is the discharge of water used as a coolant in industrial storage centres and power plants (Issakhov & Zhandaulet, 2019). When cooling water is returned to the marine environment (usually at higher temperatures), it decreases the availability of dissolved oxygen. Dissolved oxygen is essential for underwater life, and if it is lacking it can cause adverse effects such as fish death (Speight, 2020). Rising seawater temperatures also cause stratification of seawater, which occurs when an isolated layer of water forms at the top of the warm layer (epilimnion), separated by a cold layer (hypolimnion) (Huang et al., 2019). Changes in temperature can result in increased sensitivity to other pollutants. The impacts of rising temperatures include forced migration, fish deaths due to slowing metabolism, increased sensitivity to toxic substances, and loss of biodiversity. Warming seawater temperatures over the past few decades have resulted in mass coral bleaching events in many parts of the world (Black et al., 2015).

Cooling water channel installation is a reinforced concrete construction in the form of a channel which is part of the PLTU infrastructure consisting of a closed channel (box culvert), open channel (open channel), and outfall discharge outlet building to eliminate waste heat generated by various industrial processes (Darwis et al., 2023). A 3-dimensional CFD (Computational Fluid Dynamics) approach has been developed and shows effective results in designing sewer systems to reduce the discharge of pollutants into the receiving water body (Chen et al., 2013). CFD hydrodynamic simulation is one of the tools to plan the reconstruction and operation of structures in wastewater treatment at a small cost with efficient results (Patziger, 2021). Simulation of temperature distribution with different flow rates along the Irtysh River due to power plant activity using Ansys Fluent Software.

Based on the results of the numerical simulations obtained, the temperature distribution in the Irtysh River to the downstream of the river can be observed significantly. More research is needed to describe the distribution of hot water discharge to aquatic media. The numerical simulation results obtained will help minimize water damage due to power plants (Issakhov & Zhandaulet, 2019). CFDs can be an alternative to physical modelling in many areas of fluid dynamics, with the advantages of lower cost and greater flexibility. CFD can also be applied to environmental problems to simulate heat and mass transfer in rivers, lakes, oceans, atmospheres and porous media such as soil and rocks (Bates et al., 2005). Computational Fluid Dynamics (CFD) tools like Ansys Fluent are particularly suitable for such studies. They offer advantages over

physical modeling, such as lower costs and greater flexibility. CFD enables detailed simulations of heat and mass transfer in environmental contexts like rivers, lakes, and oceans, including open channel heat transfer modeling and temperature analysis (Blocken, 2015).

The purpose of this study is to predict the temperature decrease pattern of boiling water along the cooling line, which involves analyzing the distribution of hot water temperature using CFD modelling. This aims to ensure that the water discharged into the sea meets the quality standards specified in Government Regulation of the Republic of Indonesia Number 22 of 2021 Attachment VIII.

RESEARCH METHODS

The research was carried out at the coal-fired power plant consisting of seven generating units with a total generation capacity of 4,050 MW. The data used in the modelling were in the form of boiling water temperature taken at the condenser discharge (ST-DC) sampling point and discharge data. Boiling water enters the cooling line through 13 pipes that are assumed to have the same diameter of 3 meters. The water-cooling channel is a square-shaped open channel with concrete material, has a length of 2,500 meters, a width of 16 meters, and a maximum water depth of 4 meters.



Source: Google Earth, 2024

Figure 1. Research Location

The software used for CFD modeling is Ansys Fluent R1 2022. The Reynolds-averaged Navier-Stokes equation (RANS) is used to simulate thermal effects on rivers or bodies of water (Issakhov & Zhandaulet , 2019). The possibility of high-frequency fluctuations in steady flow will remain, and to account for this, an average time procedure is used, resulting in additional

requirements. These additional requirements need to be expressed as a countable amount for the solution approach. The κ - ϵ model used is a semi-empirical model based on the model transport equation for turbulent-kinetic energy ' κ ' and dissipation rate ' ϵ ' (Gandhi & Abraham, 2010). The κ - ϵ model (epsilon) is based on the equation of the turbulent kinetic energy transport model (κ) and the dissipation rate (ϵ) per unit mass (Tasar et al., 2021). The study used the κ - ϵ turbulence model (epsilon) to determine the flow characteristics in open channels (Kamel et al., 2014).

Simulation of river flow to determine the influence of the presence of baffle structures at the bottom of the river flow on the decrease in sedimentation speed along the river flow using the κ - ϵ (epsilon) model. The river flow simulation used two different turbulent models κ - ϵ (Tasar et al., 2021) (epsilon) and Reynolds Stress (RS). The κ - ϵ (epsilon) model has a lower average error percentage compared to the RS model. The difference in the total average percentage between the simulation results and the actual data from the κ - ϵ model (epsilon) is less than 25% so the numerical simulation carried out with the κ - ϵ model (epsilon) is more accurate (Usman et al., 2023). The area of water affected by the thermal effects generated by the power plant is simulated using a two-dimensional numerical model. This two-dimensional model was chosen because the horizontal change in water temperature is practically the same as the three-dimensional model (Issakhov & Zhandaulet, 2019).

Simulation of river flow with trials using low, medium, and high discharge data to determine the characteristics of the flow (Usman et al., 2023). CFD modelling validation was carried out by comparing the modelling water temperature measurement data with temperature measurement data in the field. The determination of location of field data collection is determined based on the point where it is possible to take samples, the presence of turns, straight channels, and plunges. Sampling was done directly and measured using a digital thermometer (Roy et al., 2022).

Table 1. Simulation Variations

Bahang Water Discharge		Min	Average	Max
Bahang Water Temperature		12 m3/s	20 m3/s	25 m3/s
Min	300C	A1	B1	C1
Average	370C	A2	B2	C2
Max	400C	A3	B3	C3

RESULTS AND DISCUSSION

The geometry is used to model the flow according to the actual geometry so that it can describe the actual flow. The geometry domain is then discrete into a tetrahedral finite element (triangular pyramid) through a spatial discrete network (mesh) (Simionescu et al. 2022). The meshing process carried out produced a tetrahedral mesh of 446,618 nodes and 2,260,481

elements. The quality of the mesh built is assessed based on mesh checking, resulting in a maximum skewness value of 0.83 (Acceptable) and an orthogonal quality of 0.17 – 0.9 (Acceptable – Very Good).

Table 2. Mesh Quality Recommendations

Skewness					
Excellent	Very Good	Good	Acceptable	Bad	Unacceptable
0 – 0.25	0.25 – 0.50	0.50 – 0.80	0.80 – 0.94	0.95 – 0.97	0.98 – 1.00
Orthogonal Quality					
Unacceptable	Bad	Acceptable	Good	Very Good	Excellent
0 – 0.001	0.001 – 0.14	0.15 – 0.20	0.20 – 0.69	0.70 – 0.95	0.95 – 1.00

Source: Ansys Inc, 2015

The type of solver used is pressure-based, which is commonly used in the case of incompressible flows, while density-based solvers are more suitable for compressible flows or have very high velocities (supersonic or hypersonic) (Singh & Mukhopadhyay, 2023). Steady solvers are used in these simulations because they are suitable for flow conditions that are stable or do not change over time, while transient solvers are more suitable for modelling flow changes over time, such as the presence of acceleration or deceleration of flow, changes in boundary conditions, or interactions between fluids and moving structures (Changjun et al., 2011). Scheme coupled is used in natural convection simulations because the pressure distribution and velocity generated by the fluid flow will affect the temperature distribution (Mangani et al., 2016).

Variation 1: temperature 300C, discharge 12, 20, and 25 m3/s

Variation 1 simulation was carried out at the lowest temperature of boiling water ever released into the waters during the 2019 – 2023 period, which was 30°C. The simulation results of Figure 1, Figure 2, and Figure 3 show that the temperature change pattern of hot water along the cooling line with an initial temperature of 30°C has the same temperature change pattern even though there is a difference in water discharge. Temperature changes from inlet to outlet between 30°C – 28°C. The temperature change is not too big because when the temperature of the water approaches the surrounding temperature, the temperature difference between the water and its environment becomes smaller.

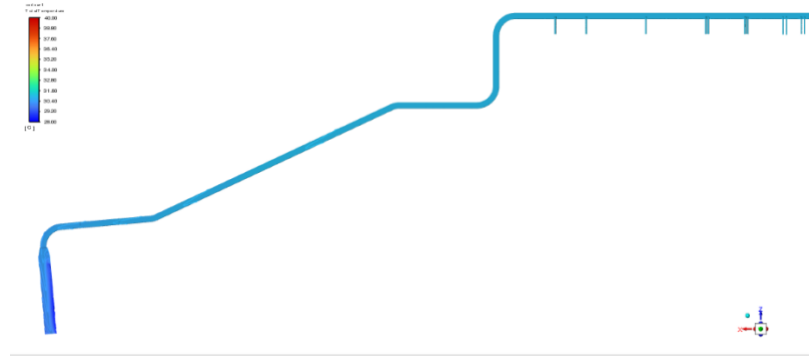


Figure 2. Simulation Results A1 Discharge 12 m3/s

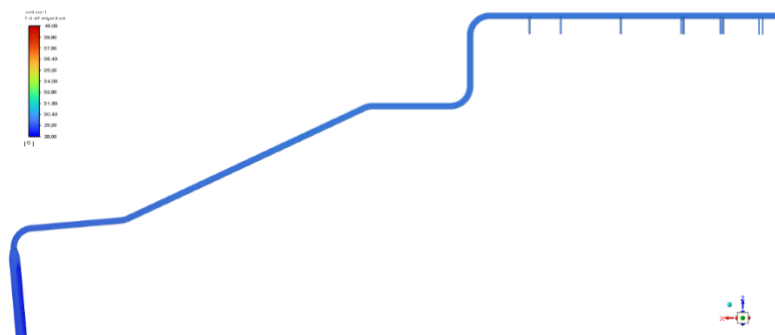


Figure 3. Simulation Results of B1 Discharge 20 m3/s

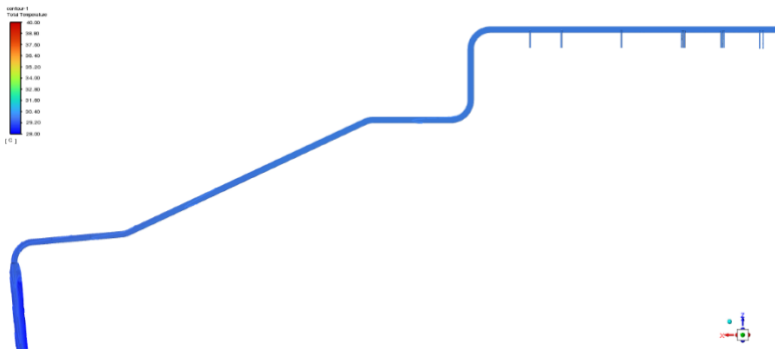


Figure 4. C1 Discharge 25 m3/s Simulation Results

The dominant temperature contour is 29°C which means that the temperature drop from the inlet is only 1°C and there is a slight blue contour on the right wall of the outlet depicting the lowest temperature of the hot water with a value of 28°C. The decrease in the temperature of the boiling water at the Suralaya PLTU was obtained as a result of a decrease in temperature every 1000 meters by 0.77°C (Purba, 2004). Temperature changes begin to occur at a distance of

100 meters from the last inlet. The temperature contours of the discharge variations B1 and C1 have a colour that depicts a lower temperature than the lowest discharge variation (A1).

Variation 2: temperature 370C, discharge 12, 20, and 25 m3/s.

The temperature of the boiling water from the condenser system generally ranges from 36 – 38°C so in the simulation, an average temperature of 37°C is obtained. Temperature changes with a maximum flow discharge occur at a distance of 550 meters, while smaller discharges occur at a distance of 950 meters from the last inlet point. The difference in the discharge of the hot water results in different contours; the farther the water flows from the inlet point, the greater the temperature change due to interaction with the surrounding environment. The time it takes for water to flow from a source to a specific point can also affect its final temperature (Liu et al., 2017).



Figure 5. A2 Discharge 12 m3/s Simulation Results

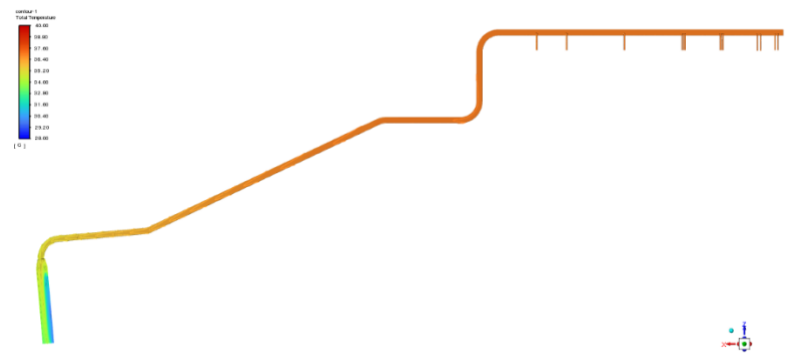


Figure 6. Simulation Results of B2 Discharge 20 m3/s

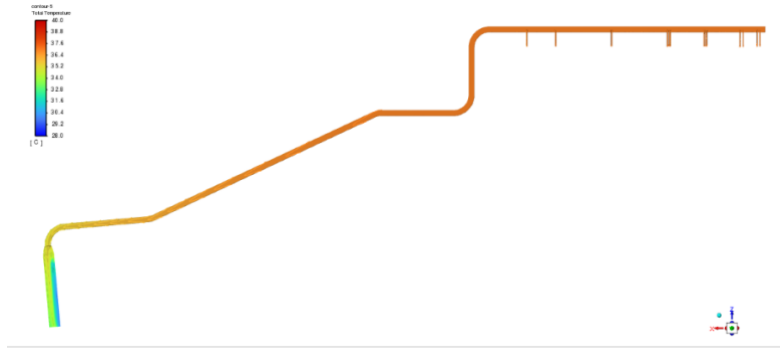


Figure 7. C2 Discharge 25 m3/s Simulation Results

Variation 3: temperature 40°C, discharge 12, 20, and 25 m3/s

Regulations regarding hot water release regulate the maximum temperature value allowed to be recorded at ST-DC or the inlet point in this study is 40°C. Temperature drops tend only to occur when approaching the outlet point, while temperatures along the flow tend to experience small temperature changes. The effect of discharge on temperature change of simulation variation 3 is clearly visible at outlet points such as in variation 1 and variation 2. Simulations with discharge variations at a boiling water temperature of 40°C also resulted in a greater discharge effect, leading to a higher drop in water temperature.

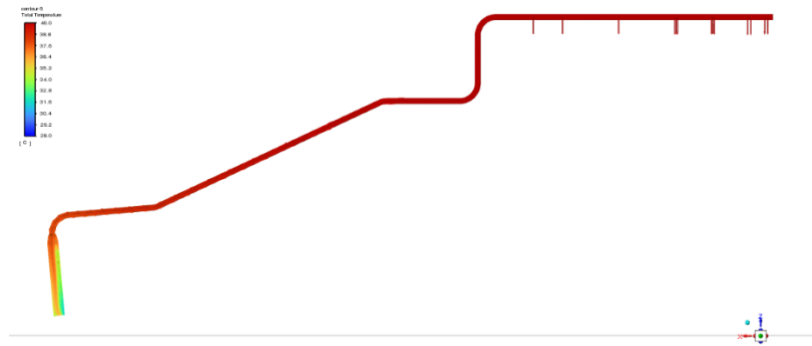


Figure 8. A3 Discharge 12 m3/s Simulation Results

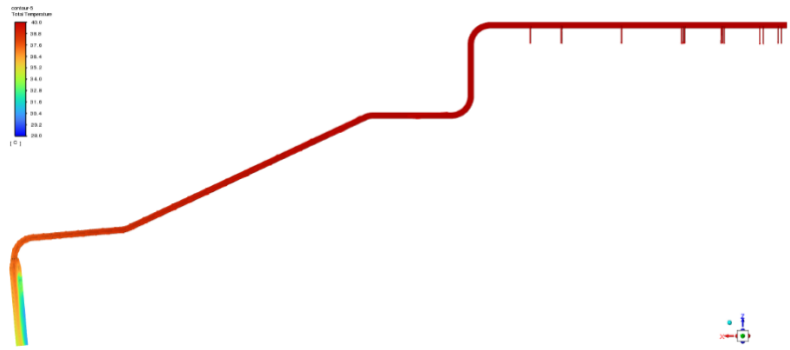


Figure 9. Simulation Results of B3 Discharge 20 m3/s

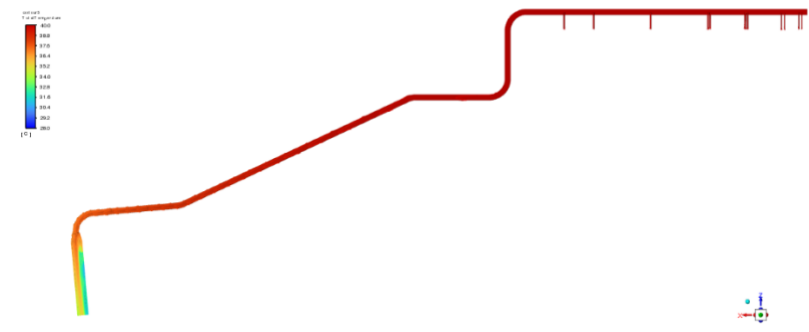
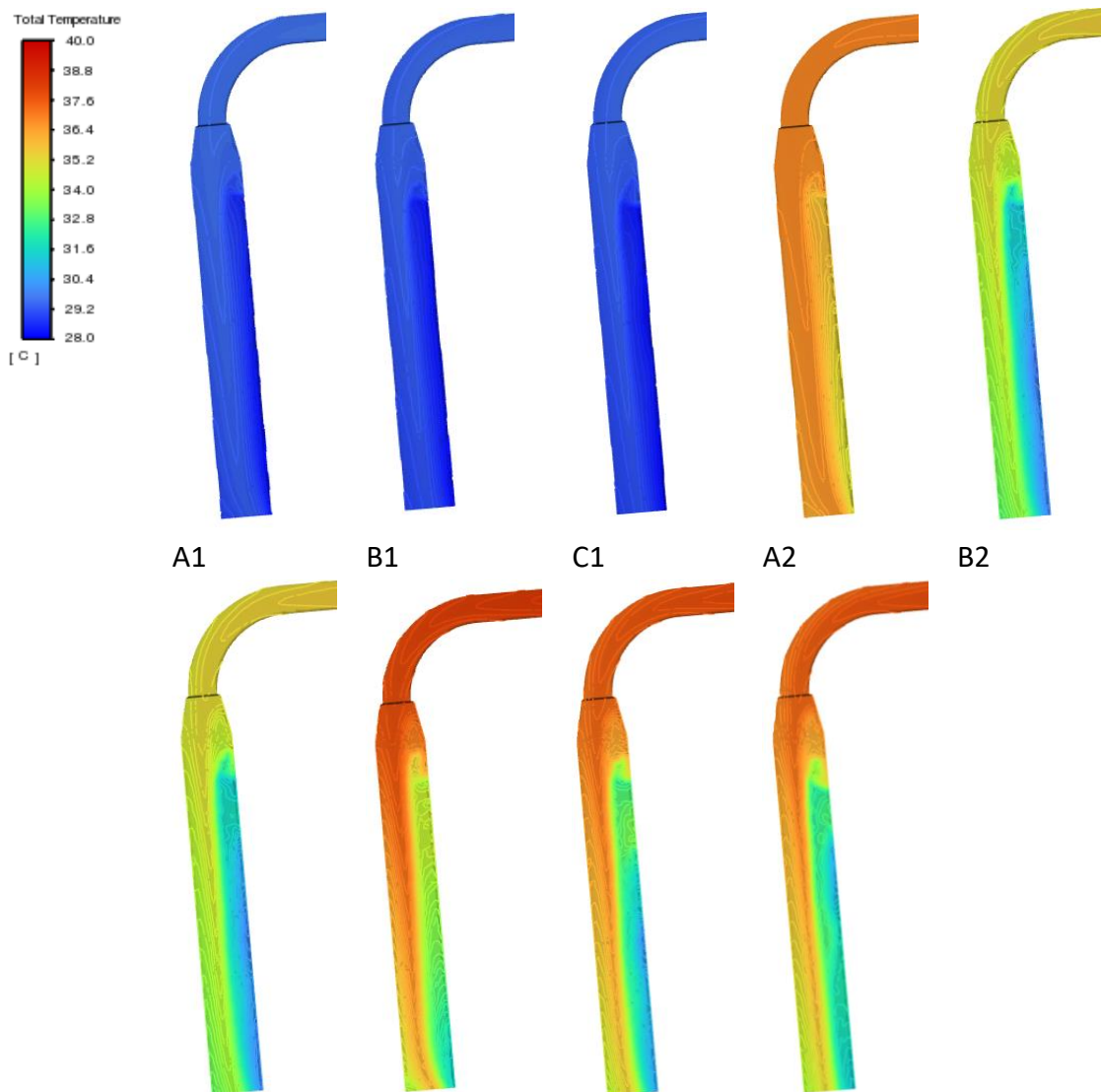


Figure 10. C3 Discharge 25 m3/s Simulation Results

The simulation results in Figures 2 to 10 show that the temperature change begins to be clearly visible when approaching the outlet, namely at the point of the channel that has a plunge. A decrease in water temperature can occur when water moves rapidly through a stream, causing water to stir and increase mixing. Stirring and mixing can cause the water to come into contact with the cooler air around it, resulting in heat transfer from water to air and a decrease in water temperature (Cheng & Constantinescu, 2018).

The simulation results show that with the same temperature, the simulation with a higher discharge experiences a decrease in water temperature at a shorter distance or at a faster time. Based on the results of the three simulation variations that have been carried out, at the maximum discharge, the temperature drop begins to be seen at a distance of 550 meters, while at a smaller discharge, the temperature decreases at a distance of 550 – 950 meters. Water discharge is closely related to the time and speed of water flow. Water discharge is the amount of water volume flowing through a flow cross-section in a certain period of time (m³/s), while flow velocity is the speed of water movement at a point in the flow (m/s). The flow speed changes as the discharge changes. The increase in water discharge will cause the flow speed to increase, so the time it takes to reach a certain point will be shorter.

The temperature drop pattern begins to be clearly visible when the water passes through the waterfall, which is a cascade aeration method. The water flows through a series of stairs. The turbulence formed from such flows can result in better mixing between water of different temperatures. Turbulence can cause water to break into small droplets, increasing the contact between water and air, and increasing the heat transfer rate from water to air. The process of cooling hot water through aeration to increase the area of contact with air and mixing can improve the process of reducing water temperature. Figure 11 shows the temperature contour at the outlet of the channel where the dominant temperature change occurs due to water passing through the waterfall. Based on these results, it was obtained that the cooling line showed the best effectiveness in reducing the temperature when the raw water input ranged from 30 to 37°C with a flow discharge of 20 – 25 m³/s.



C2 A3 B3 C3

Figure 11. Temperature Contour at Outlet

The simulation is validated using field data of the actual temperature value at the predetermined sampling point. The numerical simulation results and actual values analyzed using MAPE in Table 4 produce an error value of 11%, and the modelling that has been carried out includes the category that can predict well (good).

Table 3. MAPE Theory

No.	MAPE Value	Prediction
1	MAPE ≤ 10%	High
2	10% < MAPE ≤ 20%	Good
3	20% < MAPE ≤ 50%	Reasonable
4	MAPE ≥ 50%	Low

Table 4. MAPE calculation

Δ S1	Δ S2	Δ S3	Δ S4	Δ outlet	% error
Variation 1					
0,25005	0,24750	0,24823	0,26216	0,27559	0,25671
0,25005	0,24750	0,24865	0,26345	0,27603	0,25714
0,25005	0,24750	0,24865	0,26388	0,27647	0,25731
Variation 2					
0,01404	0,01355	0,02169	0,02987	0,04485	0,02480
0,01404	0,01355	0,02225	0,06023	0,09369	0,04075
0,02289	0,04265	0,10128	0,09599	0,09531	0,07162
Variation 3					
-0,06207	-0,06289	-0,05465	0,01214	0,04455	-0,02459
-0,06184	-0,06172	-0,05225	0,02418	0,07150	-0,01603
-0,06184	-0,06195	-0,05201	0,03044	0,07618	-0,01384
Total Error					9,48751

CONCLUSION

The decrease in the temperature of the boiling water in nine simulated variations with temperature and discharge differences generally occurred at a distance of 550 m from the last inlet. The water flow tends to be on the lower side, and in the downstream part, the flow tends to be on the left and middle so that the right part of the channel tends to be saturated. The active water temperature decreases when there is a stir in the form of a cascade aerator downstream. The cooling line's performance in lowering the boiling water temperature shows the best effectiveness if the input temperature ranges from 30 - 37°C with a flow discharge of 20 - 25

m³/s. Existing cooling channels tend to be less effective in lowering the temperature of hot water, so it is necessary to conduct a review to increase the decrease in water temperature so that it can prevent marine pollution due to higher water temperatures.

BIBLIOGRAPHY

- Baag, S., & Mandal, S. (2022). Combined effects of ocean warming and acidification on marine fish and shellfish: A molecule to ecosystem perspective. In *Science of the Total Environment* (Vol. 802). Elsevier B.V. <https://doi.org/10.1016/j.scitotenv.2021.149807>
- Bates, P. D., Stuart N. Lane, & Robert I. Ferguson. (2005). *Computational Fluid Dynamics: Applications in Environmental Hydraulics*.
- Black, J. G., Reichelt-Brushett, A. J., & Clark, M. W. (2015). The effect of copper and temperature on juveniles of the eurybathic brittle star *Amphipholis squamata* - Exploring responses related to motility and the water vascular system. *Chemosphere*, 124(1), 32–39. <https://doi.org/10.1016/j.chemosphere.2014.10.063>
- Blocken, B. (2015). Computational Fluid Dynamics for urban physics: Importance, scales, possibilities, limitations and ten tips and tricks towards accurate and reliable simulations. *Building and Environment*, 91, 219–245. <https://doi.org/10.1016/j.buildenv.2015.02.015>
- Changjun, L., Wenlong, J., & Xia, W. (2011). *Modeling and Simulation for Steady State and Transient Pipe Flow of Condensate Gas*. www.intechopen.com
- Chen, Z., Han, S., Zhou, F. Y., & Wang, K. (2013). A CFD Modeling Approach for Municipal Sewer System Design Optimization to Minimize Emissions into Receiving Water Body. *Water Resources Management*, 27(7), 2053–2069. <https://doi.org/10.1007/s11269-013-0272-9>
- Cheng, Z., & Constantinescu, G. (2018). Stratification Effects on Flow Hydrodynamics and Mixing at a Confluence With a Highly Discordant Bed and a Relatively Low Velocity Ratio. *Water Resources Research*, 54(7), 4537–4562. <https://doi.org/10.1029/2017WR022292>
- Chiasson, A. (2016). Waste Heat Rejection Methods in Geothermal Power Generation. In *Geothermal Power Generation: Developments and Innovation* (pp. 423–442). Elsevier Inc. <https://doi.org/10.1016/B978-0-08-100337-4.00015-2>
- Darwis, I., Dwiyanto, M., Prakasa, A., & Amansah, S. (2023). Analisis Penurunan Suhu Air Limbah PLTU Menggunakan Cooling Water Way pada PLTU Jeneponto 2X125MW Analysis of Reducing The Temperature of Steam Power Plant Waste Water Using Cooling Water Way PLTU Jeneponto 2X125 MW. In *Journal of Applied Civil and Environmental Engineering* (Vol. 3, Issue 2).
- Gandhi, B. K., & Abraham, B. (2010). *Investigation of Flow Profile in Open Channels using CFD*.
- Huang, F., Lin, J., & Zheng, B. (2019). Effects of thermal discharge from coastal nuclear power plants and thermal power plants on the thermocline characteristics in sea areas with different tidal dynamics. *Water (Switzerland)*, 11(12). <https://doi.org/10.3390/w11122577>

- Issakhov, A., & Zhandaulet, Y. (2019). Numerical simulation of thermal pollution zones' formations in the water environment from the activities of the power plant. *Engineering Applications of Computational Fluid Mechanics*, 13(1), 279–299. <https://doi.org/10.1080/19942060.2019.1584126>
- Kamel, B., Ilhem, K., Ali, F., & Abdelbaki, D. (2014). 3D simulation of velocity profile of turbulent flow in open channel with complex geometry. *Physics Procedia*, 55, 119–128. <https://doi.org/10.1016/j.phpro.2014.07.018>
- Lee, U., Chou, J., Xu, H., Carlson, D., Venkatesh, A., Shuster, E., Skone, T. J., & Wang, M. (2020). Regional and Seasonal Water Stress Analysis of United States Thermoelectricity. *Journal of Cleaner Production*, 270. <https://doi.org/10.1016/j.jclepro.2020.122234>
- Liu, Z., Yang, M., Wan, G., & Wang, X. (2017). The spatial and temporal variation of temperature in the qinghai-xizang (Tibetan) plateau during 1971-2015. *Atmosphere*, 8(11). <https://doi.org/10.3390/atmos8110214>
- Patziger, M. (2021). Improving Wastewater Treatment Plant Performance by Applying CFD Models for Design and Operation: Selected Case Studies. *Water Science and Technology*, 84(2), 323–332. <https://doi.org/10.2166/wst.2021.019>
- Roy, P., Rao, I. N., Martha, T. R., & Kumar, K. V. (2022). Discharge Water Temperature Assessment of Thermal Power Plant using Remote Sensing techniques. *Energy Geoscience*, 3(2), 172–181. <https://doi.org/10.1016/j.engeos.2021.06.006>
- Simionescu, S. M., Gogoase, D. E., CIRCUMARU, G., & Chihaia, R. A. (2022). Numerical Simulation of Water Flow through an Ecological River Intake. *Hidraulica*, 2.
- Singh, A., & Mukhopadhyay, S. (2023). Comparison of pressure-based and density-based solvers for scramjet modeling. *AIP Conference Proceedings*.
- Speight, J. G. (2020). Sources of water pollution. In *Natural Water Remediation* (pp. 165–198). Elsevier. <https://doi.org/10.1016/b978-0-12-803810-9.00005-x>
- Tasar, B., Unes, F., Gemici, E., & Varcin, H. (2021). *Numerical Simulation of Channel Flow Using Submerged Vane in River Arrangements*. 119–130. https://doi.org/10.24193/AWC2021_11
- Usman, M., Shahid, S., Ali, S., & Ullah, M. K. (2023). Numerical simulations of turbulent and flow characteristics of complex river reach in Pakistan. *Environmental Engineering Research*, 28(1). <https://doi.org/10.4491/eer.2021.369>

Copyright holder:

Rohmah Iftitah Sa'idatul Izzah, Harmin Sulistyning Titah, Shade Rahmawati (2024)

First publication right:

Asian Journal of Engineering, Social and Health (AJESH)

This article is licensed under:

