The Advanced Cooling System of Generator for Hydropower Plant

Abdulmuttalib A. Muhsen[1], Natalya Kizilova[2]

[1,2]Faculty of Power and Aeronautical Engineering, Warsaw University of Technology, Warsaw

[1]Haditha Hydropower Station, Ministry of Electricity, Haditha, Iraq

Abstract—**This paper documents heat and fluid flow characteristics of the counterflow heat exchangers with line-to-line flow channels embedded fins of a fractal air-water cooler for hydrodynamic power plant generator which was analyzed numerically using AnSys 2021 R1 Fluent software to simultaneously solve the continuity, momentum, and heat equations in a 3D computational domain, and the conformal mesh technique is applied to generate the grid for the conjugate heat flow in the cooling system. At the low Reynolds numbers, the laminar viscous flow model was used, while at higher Re numbers Re~1200 the transitional flow regime was studied on the k-epsilon turbulent model. The governing pressure drop was ~(35-45)x10^5 Pa. Numerical results revealed that the fractal geometry can improve the integrated performance of the cooling system. The pressure drop through the pipe channel of the cooler as decreases obviously when compared with that of conventional parallel channels. The fractal system showed a more uniform temperature distribution and relatively lower maximum surface temperature. Application of the fins could intensify the transfer of heat remarkably and lower the power of the pump in a quick fashion related to the conventional tube system.**

Keywords - fluid dynamics, cooling channel, convection, heat transfer, fractal network

I. Introduction

Hydropower is a conventional clean renewable source of energy that plays an important role in sustainable energy. Nowadays hydropower plants located in 150 countries generate ~29% of total energy in China, 9% in Brazil, 7% in the US and Canada, and 31% in the rest world [1]. In 2020 global hydropower generation increased 1.5% to reach 4,370 TWh, representing around 16.8% of the world's total electricity generation. It is expected to increase by ~ 3.1% each year for the next 25 years.

Efficient cooling of the hydropower plant is essential to maintain and keep the system in good operational conditions [2]. Hydropower generators' need a cooling system to dissipate heat from some of the heated areas such as the turbine seals, the oil-filled transformers, and lubrication bearings. A close loop circuit in which water supply passes through a tubular heat exchanger directly submerged into draft tube water was proposed in [3]. The hydropower plant must integrate a systematic cooling system such as a heat exchanger (Fig.1) [4]. The efficient design of the cooling system must provide fast uniform heat removal at low-pressure drops and with low acoustic emissions due to flow-induced vibrations of the solid structures [5, 6]. The thermal contact design between the fluid and solid structures has important applications in heat pumps with extraction and release and thermal energy storage [7, 8].

The objective of the paper is to determine the thermal performance of the fluid flow structure of the cooling system. To be precise, we analyze

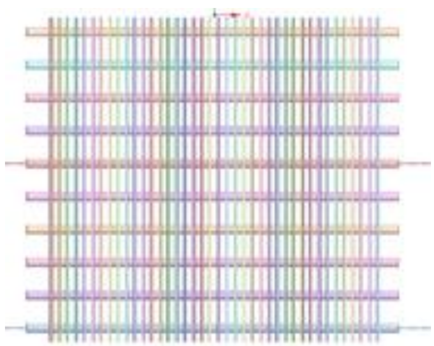

Figure 1. Pipe channel cooling system type.

the effect of the flow structure on the overall performance of the system.

II. Cooling Systems for Generators

A hydropower plant like that of Haditha hydropower plant (HPP) is one of the principal hydropower plants in Iraq. The plant has 6 units of Kaplan turbines capable generating of 110 MW of power each which generates a total capacity of 660MW [4]. The most essential part of the hydropower plant is the turbine which is attached to the electric generators that turn the generators by utilizing the energy of water flowing through and impacting on the turbine blades. The friction due to the rotation of the rotary components of the generators causes heat to be generated as well as the fluid friction also causes heat to be generated on the generator components. The occurrence of thermal losses on the generator unit at maximum load is much it is approximately 1000kW. The need for the cooling system in hydropower generators is required. All the generators in the Haditha hydropower plant have 12 radiators located in the stator hub of the generator that is an ideal isolated environment for the radiators to function properly.

In the conventional cooling systems (Fig. 2), the radial fan located at the stator hub makes the airflow which warms the fluid in return. The airflow is constant because the angular velocity of the hydropower unit is fixed. Cold fluid is circulated from the dam into the cooling system through the pipes. Consequently, the temperature data were recorded by the monitoring unit of the plant, as 15˚C to 27˚C of cooling systems inlet changes [8-10].

III. Analysis of the Cooling System

The cooling system for the hydropower plant generator consists of a series of tubes and fins which are housed on a stainless-steel frame.

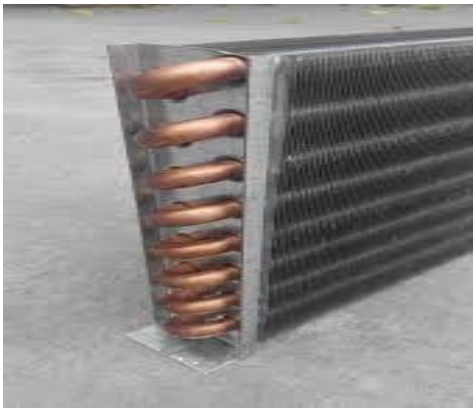

Figure 2. Conventional channel type cooling system.

These tubes contain a heated fluid that needs cooling. The fins are in contact with the walls of the tubes that are used to facilitate air to cool the tube through convection so that it can absorb the heat from the tubes. An engineering simulation software was used to model and design the tubes and fins of the cooling system in ANSYS 2021 R1, Workbench. The analysis was performed using Ansys FLUENT, which was used to compute equations using the Computational Fluid Dynamics (CFD) volumetric control approach (VCA). This VCA simplifies the complex differential equations into simple algebraic equations. The geometric modeling of the cooling system was done using the parametric approach in Solidworks software. Meshing was done using the structured grids as well as unstructured grids using the method and sizing controls to define the meshing properties of the model. The boundary conditions were defined using the name selection tool to indicate the inlet, outlet, walls, etc. in the Ansys modeler for the cooling system model. This file was imported into Fluent software in the Ansys workbench and has given the input values like mass flow rate, pressure, temperature, etc.

TABLE I. Dimensions and Materials for the Cooling System.

Overall Dimension	**Dimension of tube**	**Material details**	**Types of tube pattern**
Length=800mm	Outer diameter=20mm	ANSI1316L Stainless steel	Straight Parallel tubes
Width=140mm	Inner diameter=16mm		
Height=720mm			

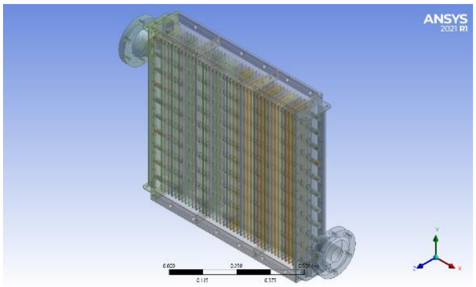

Figure 3. Geometrical model of the cooling system.

Realistic 3D geometry (Fig. 3) was used for the analysis of the heat transfer by the moving fluid in the cooling system sufficient to cool the powerplant generator for solving the mass, momentum, and energy equations. The mean pressure and temperature were defined at the inlet of the cooling system. The laminar flow models were used to solve the convergence of the equations setting the functions as wall standards [11]. The gravitational acceleration of 9.81 m/s2 in the downward flow direction was used. The study was done in a steady state to obtain the results. The inlet of the temperature is 25°C. The material of the pipe is ANSI1316L Stainless steel, and the fins are AlMg3 Aluminum Alloy was used to determine the rate of heat transfer between the two materials using convection [12-14].

After the model is designed, a thermal simulation of the cooling system is to be run. The general conditions played role in the study with the computational domain. Fig. 1 shows the model of the cooling system in Ansys CFD.

IV. Mathematical Model Formulation

The following equations based on the balance of mass, momentum, and energy have been used [15]:

- Mass conservation:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0\ , \tag{1}$$

- Momentum Eq.:

$$\rho\left(\frac{\partial(uu)}{\partial x}+\frac{\partial(uv)}{\partial y}+\frac{\partial(uw)}{\partial z}\right)=-\frac{\partial p}{\partial x}+\left(\begin{array}{l}\frac{\partial}{\partial x}\left(\mu\frac{\partial u}{\partial x}\right)+\\ +\frac{\partial}{\partial y}\left(\mu\frac{\partial u}{\partial y}\right)+\frac{\partial}{\partial z}\left(\mu\frac{\partial u}{\partial z}\right)\end{array}\right), \tag{2}$$

$$\rho\left(\frac{\partial(uv)}{\partial x}+\frac{\partial(vv)}{\partial y}+\frac{\partial(vw)}{\partial z}\right)=-\frac{\partial p}{\partial x}\left(\begin{array}{l}\frac{\partial}{\partial x}\left(\mu\frac{\partial v}{\partial x}\right)+\\ +\frac{\partial}{\partial y}\left(\mu\frac{\partial v}{\partial y}\right)+\frac{\partial}{\partial z}\left(\mu\frac{\partial v}{\partial z}\right)\end{array}\right), \tag{3}$$

$$\rho\left(\frac{\partial(uw)}{\partial x}+\frac{\partial(vw)}{\partial y}+\frac{\partial(ww)}{\partial z}\right)=-\frac{\partial p}{\partial x}+\frac{\partial}{\partial x}\left(\mu\frac{\partial w}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu\frac{\partial w}{\partial y}\right)+\frac{\partial}{\partial z}\left(\mu\frac{\partial w}{\partial z}\right). \tag{4}$$

- Energy Eq.:

Energy in the fluid domain:

$$\rho_f\left(\begin{array}{l}\frac{\partial\left((c_p)_f uT\right)}{\partial x}+\frac{\partial\left((c_p)_f vT\right)}{\partial y}+\\ +\frac{\partial\left((c_p)_f wT\right)}{\partial z}\end{array}\right)=\frac{\partial}{\partial x}\left(k_f\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(k_f\frac{\partial T}{\partial y}\right)+\frac{\partial}{\partial z}\left(k_f\frac{\partial T}{\partial z}\right), \tag{5}$$

Energy in the fluid domain:

$$=\frac{\partial}{\partial x}\left(k_f\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(k_s\frac{\partial T}{\partial y}\right)+\frac{\partial}{\partial z}\left(k_s\frac{\partial T}{\partial z}\right). \tag{6}$$

The boundary conditions for computations were taken as follows [16, 17]:

TABLE II. BOUNDARY CONDITIONS.

Inlet	Pressure-inlet
Outlet	Zero pressure
Dividers	Convective, no-slip criteria
Liquid	Water (H_2O)
Solid Material	Stainless steel Aluminum Alloy
Energy Condition	ON
Computational model	K-epsilon, feasible, standard wall treatment
Solution Method	Second-order

V. NUMERICAL PROCEDURE

CFD simulations were performed using a finite-volume package with a pressure-based solver, node-based gradient evaluation, the Eqs. (2)-(6) are constant values used in the simulation together with the SIMPLE algorithm for pressure–velocity coupling, and a second-order upwind scheme for momentum and energy equations [18]. In grid generation, tetra grids are adopted for conformal meshing between the fins and the tube. The independence of the solution with respect to the grid size was checked by inspecting the values of the mean pressure, maximum temperature, temperature differences between the inlet and outlet, the coefficient of heat transfer for convection between the wall of the tube and the fins [19-25].

The mesh was carried out on the geometry to ensure there is conformal meshing between the tubes and the fins using the tetrahedral method. The cell grids vary across the model to achieve medium coarse mesh; the smallest number was $6x10^6$.

Convergence is achieved when the residuals for the mass and momentum equation are smaller than 10^{-4}, and the residual of the energy equation is less than 10^{-11}.

TABLE III. RESULT OF THE OUTLET OF THE COOLING SYSTEM ANALYSIS.

Parameters	**Units**
Maximum Velocity	3.5m/s^2
Outlet Maximum pressure	0.9Mpa
Temperature	22.4°C

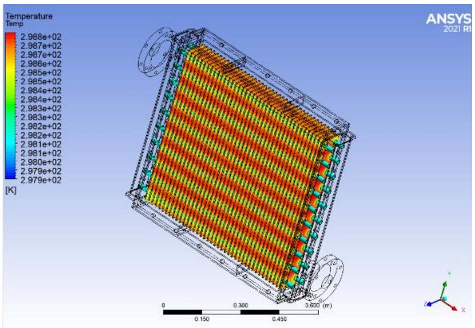

Figure 4. Temperature distribution in the cooling system.

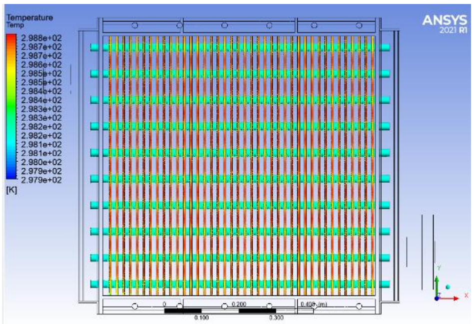

Figure 5. Temperature of the fins and the tubes of the cooling system.

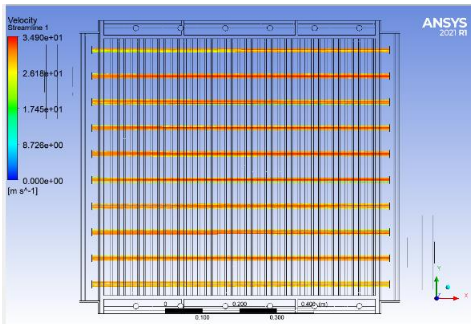

Figure 6. Streamline velocity of the cooling system.

VI. RESULTS AND DISCUSSION

From the CFD analysis of the cooling system of the power plant generator with the variations in tubes, we found the distributions of temperature (Figs. 4, 5), velocity (Fig. 6), and static pressure along the channels and fins.

Numerical simulations revealed the temperature differences. Some numerical results are summarized in Table III. Conclusion

From this study, the flow and heat transfer characteristics in the cooling structure for the system, parallel flow is investigated using CFD in the current study, and the optimization procedure for the leading influencing factors including the level number.

We select Stainless steel material for the tubes and Aluminum Alloy material for fins then analyzed in Ansys and enhance the optimized material temperature value. Assuming that the cooling is system is isolated from the environment and contrasting the results the maximum heat transfer occurred at the parallel flow of the cooling system.

It may be assumed that no heat transfer is taking place in between model and surroundings. It is concluded that tubes and fins need modification to increase the efficiency of the heat transfer flow, water in Parallel flow tubes gives 22°C temps at the outlet which is lowered from 25°C.

REFERENCES

[1] Renewables (2021). *Global status report*. Available at: https://www.ren21.net/wp-content/uploads/2019/05/GSR2021_Full_Report.pdf

[2] Kumar, K., & Saini, R. P. (2022). A review on operation and maintenance of hydropower plants. *Sustainable Energy Technologies and Assessments*, *49*, 101704.

[3] Shah, S. K., & Shrestha, R. (2015). Design Modification of Cooling Water System for Hydropower Plants [A Case Study of Middle Marsyangdi Hydropower Plant in Nepal]. *Global Journal of Research In Engineering*, *15*(6), 090699.

[4] Muhsen, A. A., Szymański, G. M., Mankhi, T. A., & Attiya, B. (2018). Selecting the most efficient maintenance approach using AHP multiple criteria decision making at haditha hydropower plant. *Zeszyty Naukowe Politechniki Poznańskiej. Organizacja i Zarządzanie.*

[5] Hamadiche, M., Kizilova, N., & Gad-el-Hak, M. (2009). Suppression of absolute instabilities in the flow inside a compliant tube. Communications in *Numerical Methods in Engineering, 25*(5), 505-531.

[6] Kizilova N., Hamadiche M., & Gad-el-Hak M. (2011). Flow stabilization in compliant ducts: from nature-made to human-made. *International Journal Numerical Methods and Applications*, *6*(1), 1-86.

[7] Irshad, M., Kaushar, M., & Rajmohan, G. (2017). Design and CFD analysis of shell and tube heat exchanger. International *Journal of Engineering Science and Computing, 7*(4), 6453-6457.

[8] Kaliappan, A. S., & Mothilal, T. (2018). Thermal analysis of shell and tube heat exchanger. International *Journal of Pure and Applied Mathematics, 119*(12), 14299-14306.

[9] Mohanty, S., & Ahmed, S. U. (2018). Performance prediction of Counter flow Heat Exchanger by using CFD technique. *International Journal of Engineering Development and Research, 6*(2).

[10] Patil, A. S., & Farkade, H. S. (2017). Advances in Design and Development of Heat Exchangers: A Review. *International Research Journal of Engineering and Technology (IRJET), 4*(05).

[11] Sethi, C. K. (2017). CFD Analysis on Effectiveness of a Plate Type Heat Exchanger Using Sea Water and Engine Oil. International *Journal of Advanced Mechanical Engineering, 12*(1), 191-198.

[12] Shrivastava, D., & Mishra, R. (2017). CFD Analysis of Heat Transfer for Tube-In-Tube Heat Exchanger. *International Journal of Recent Trends in Engineering & Research (IJRTER), 3*(08).

[13] Kishan, R., Singh, D., & Sharma, A. K. (2020). CFD Analysis of Heat Exchanger Models Design Using Ansys Fluent. *International Journal of Mechanical Engineering and Technology, 11*(2).

[14] Miao, Q., Wang, Z., & Wang, L. (2017). A Failure Analysis of Air Heat Exchanger Based on Modal Calculation and Thermal-Structural Interaction Simulation. *International Journal of Materials, Mechanics and Manufacturing, 5*(3).

[15] Miao, Q., Wang, Z., & Wang, L. (2017). A Failure Analysis of Air Heat Exchanger Based on Modal Calculation and Thermal-Structural Interaction Simulation. *International Journal of Materials, Mechanics and Manufacturing, 5*(3).

[16] Jayachandriah, B., & Rajasekhar, K. (2014). Thermal analysis of tubular heat exchangers using ANSYS. *International Journal of Engineering Research, 3*(SP 1), 21-25.

[17] Rao, J. B. B., & Raju, V. R. (2016). Numerical and heat transfer analysis of shell and tube heat exchanger with circular and elliptical tubes. *International Journal of Mechanical and Materials Engineering, 11*(1), 1-18.

[18] *Fluent User's Maual.* (2021). ANSYS, Inc.

[19] Dincer, I., & Rosen, M. A. (2012). *Exergy: energy, environment and sustainable development.* Newnes.

[20] Wu, S. Y., Yuan, X. F., Li, Y. R., & Xiao, L. (2007). Exergy transfer effectiveness on the heat exchanger for finite pressure drop. *Energy, 32*(11), 2110-2120.

[21] Ujiie, R., Arlitt, R., & Etoh, H. (2006). Application of computational fluid dynamics (CFD) on ventilation-cooling optimization of electrical machines. *Review Energy Technologies Generation, Transmission and Distribution of Electric and Thermal Energy*, 17-22.

[22] Doost, A. K., & Majlessi, R. (2015). Heat transfer analysis in the cooling system of hydropower's generator. *Open Journal of Applied Sciences, 5*(03), 98.

[23] Doost, A. K., & Majlessi, R. (2015). Heat transfer analysis in the cooling system of hydropower's generator. *Open Journal of Applied Sciences, 5*(03), 98.

[24] Şahin, A. Ş., Kılıç, B., & Kılıç, U. (2011). Design and economic optimization of shell and tube heat exchangers using Artificial Bee Colony (ABC) algorithm. *Energy Conversion and Management, 52*(11), 3356-3362.

[25] Cengel, Y. A., & Ghajar, A. J. (2007). *Heat and mass transfer. A practical approach.*