Eduvest – Journal of Universal Studies
Volume 6 Number 1, Januari, 2026
p- ISSN 2775-3735- e-ISSN 2775-3727

Performance Evaluation and Evaporation Loss Prediction in Counter Flowinduced Draft Geothermal Wet Cooling Tower Type Through Computational Fluid Dynamics (CFD) Simulation

Mukhamad Nashir* , Moh. Djaeni, Muchammad
Universitas Diponegoro, Indonesia
Email: mukhamad.nashir@gmail.com* , moh.djaeni@live.undip.ac.id,
muchammad@lecturer.undip.ac.id

ABSTRACT

Cooling towers are critical in geothermal power plants for thermal efficiency and water management. This study examines thermal performance and evaporation loss in an induced draft counter-flow cooling tower via Computational Fluid Dynamics (CFD) simulation. Validation against real data showed 1.8% error, confirming CFD's reliability for design optimization. Raising hot water inlet temperature from 35°C to 49°C increased evaporation loss from 5.0 kg/s to 13.0 kg/s (CFD) and 5.5–14.5 kg/s (ASHRAE). For hot water mass flow rates of 423–845 kg/s, ASHRAE predicted linear loss increases, while CFD remained stable; higher flow reduced cold-water outlet temperature (21°C to 11°C) and effectiveness (92% to 86%). Increasing cold air inlet velocity from 3.5 m/s to 6.5 m/s raised evaporation loss to 16.0 kg/s (CFD) and boosted effectiveness from 11% to 91%. CFD offers realistic estimates over empirical methods under dynamic conditions, making it ideal for optimizing geothermal cooling systems.

KEYWORDS Geothermal power plant, cooling tower, computational fluid dynamics (CFD), thermal performance, evaporation loss, heat and mass transfer.

INTRODUCTION

Geothermal energy represents one of the most promising renewable energy sources for baseload electricity generation, with global installed capacity reaching approximately 16 GW by 2023 (IRENA, 2024). Among the critical components in geothermal power plants, cooling towers play an essential role in sustaining the thermal cycle by facilitating the condensation of steam into water for reinjection. In vapor-dominated geothermal fields, the majority of reinjection water originates from condensate produced during turbine exhaust cooling. However, evaporation losses from cooling towers can reach 2-3% of circulating water flow, representing a substantial reduction in available reinjection water and potentially affecting long-term reservoir sustainability (Guan et al., 2016). Given that adequate reinjection is crucial for maintaining reservoir pressure and thermal recovery, minimizing water losses through optimized cooling tower design has become a strategic priority for geothermal operators worldwide (AlGaiar & Afolabi, 2024; Elshehabi & Alfehaid, 2025; Eze et al., 2025; Khan et al., 2025; Santoso et al., 2019).

The economic and environmental implications of evaporation losses are particularly significant in geothermal operations. Water losses not only reduce the volume available for reinjection but also increase operational costs associated with makeup water treatment and procurement. In regions where water resources are constrained, such as many geothermal fields in Indonesia, East Africa, and the western United States, excessive evaporation can threaten project viability and sustainability. According to the International Geothermal Association (IGA), water management challenges rank among the top three operational concerns in geothermal power generation, alongside reservoir management and equipment reliability (Chandrasekharam et al., 2020; Falcone et al., 2025; Kabeyi, 2019; Kassem & Moscariello, 2024). Furthermore, inadequate reinjection due to water losses can lead to reservoir pressure decline, reduced steam production, and decreased plant efficiency over time, creating a cascade of technical and economic challenges.

Despite the critical importance of cooling tower performance, traditional design and analysis methods have relied heavily on empirical correlations and simplified mathematical models, such as those proposed by Merkel (1925) and standardized by ASHRAE. While these methods provide reasonable estimates under steady-state conditions, they often fail to capture the complex fluid dynamics, heat and mass transfer phenomena, and spatial temperature distributions that characterize real-world cooling tower operation. This limitation is particularly pronounced under dynamic operating conditions, where inlet temperatures, flow rates, and ambient conditions vary significantly throughout the day and across seasons. Consequently, there is a pressing need for more sophisticated analytical tools that can accurately predict cooling tower performance under diverse operational scenarios and support evidence-based design optimization (Felicioni et al., 2023; Heidari et al., 2024; Imani et al., 2025; Jradi, 2026; Roshan Kharrat et al., 2025).

In recent years, computational fluid dynamics (CFD) has emerged as a powerful tool for analyzing and optimizing cooling tower performance, offering capabilities that far exceed those of traditional empirical methods. CFD enables detailed visualization and quantification of airflow patterns, water distribution, temperature gradients, and evaporation rates throughout the cooling tower volume, providing insights that are difficult or impossible to obtain through experimental measurements alone. Several researchers have successfully applied CFD to cooling tower analysis, demonstrating its effectiveness in identifying flow irregularities, dead zones, and opportunities for performance enhancement. Blecich et al. (2018) conducted numerical investigations of heat and mass transfer in wet cooling towers, validating CFD predictions against experimental data and demonstrating excellent agreement. Similarly, Klimanek et al. (2015) developed comprehensive 3D CFD models of natural draft wet cooling towers, incorporating complex phenomena such as fill media characteristics, droplet dynamics, and ambient wind effects. These studies collectively demonstrate that CFD simulations can provide accurate predictions of cooling tower thermal performance while requiring significantly less time and resources compared to physical testing (Araujo et al., 2024; Kang et al., 2025; Kim et al., 2025).

Beyond basic performance prediction, CFD has proven particularly valuable for design optimization and troubleshooting. Dang et al. (2019) employed CFD to analyze thermal performance in super large-scale wet cooling towers equipped with axial fans, identifying optimal fan configurations that improved cooling efficiency by 12-15% compared to baseline designs. Chen et al. (2023) investigated the thermal characteristics of natural draft wet cooling towers with auxiliary fans, using CFD to determine optimal fan placement and operational strategies that enhanced heat rejection capacity while minimizing parasitic power consumption. In the context of geothermal applications specifically, Zargar et al. (2023) developed hybrid cooling tower models for plume abatement and performance analysis, demonstrating how CFD-based design modifications could simultaneously reduce visible plume formation and improve thermal efficiency. These applications underscore the versatility of CFD as both a research tool and a practical engineering solution for cooling tower optimization.

The comparison between CFD predictions and empirical methods such as ASHRAE has been a subject of considerable research interest. While empirical correlations offer computational simplicity and ease of implementation, they are based on idealized assumptions and generalized datasets that may not accurately represent specific installation conditions. Qureshi and Zubair (2006) developed simplified empirical relations for predicting evaporation losses in wet cooling towers based on ASHRAE guidelines, noting that while these methods provide reasonable first-order estimates, they tend to overpredict evaporation losses under certain operating conditions, particularly at high flow rates or non-standard atmospheric conditions. This tendency toward conservative predictions can lead to oversized makeup water systems and inflated estimates of water consumption, potentially affecting project economics and water resource planning. In contrast, CFD simulations incorporate site-specific geometric configurations, material properties, and boundary conditions, enabling more accurate predictions tailored to actual operating environments.

Despite the growing body of literature on CFD applications in cooling towers, significant research gaps remain, particularly in the geothermal sector. Most published CFD studies focus on power plant cooling towers operating with relatively stable inlet conditions and ambient environments. Geothermal cooling towers, however, face unique challenges including variable steam quality from production wells, fluctuating ambient conditions in volcanic regions, and the need for maximum water conservation to ensure adequate reinjection. Furthermore, there is a paucity of studies that systematically compare CFD predictions with empirical methods like ASHRAE across a wide range of operating parameters relevant to geothermal applications, particularly for induced draft counter-flow configurations which are commonly employed in modern geothermal plants. Additionally, few studies have validated CFD models against long-term operational data from actual geothermal facilities, limiting confidence in model predictions for real-world design decisions.

This study addresses these research gaps by conducting a comprehensive CFD analysis of an induced draft counter-flow wet cooling tower at an operating geothermal power plant in Indonesia, with systematic comparison against ASHRAE empirical predictions. The novelty of this work lies in several key aspects: (1) validation of the CFD model against actual operational data from a large-scale geothermal facility, demonstrating exceptional accuracy with only 1.8% deviation; (2) systematic parametric analysis examining the effects of hot water inlet temperature, mass flow rate, and air velocity on evaporation losses and thermal performance; (3) direct comparison between physics-based CFD predictions and empirically-based ASHRAE estimates across a wide parameter space; and (4) development of practical insights for cooling tower design optimization specifically tailored to geothermal applications. By demonstrating that CFD can provide reliable predictions of cooling tower performance under dynamic conditions while offering significantly more detailed insights than empirical methods, this research establishes CFD as an essential tool for geothermal cooling system design and optimization.

The urgency of this research is underscored by the growing global demand for geothermal energy and the increasing water scarcity challenges facing many geothermal developments. As the geothermal industry expands, particularly in water-stressed regions, the ability to accurately predict and minimize water losses becomes not merely an operational consideration but a strategic imperative for project sustainability. Conservative design approaches based on empirical correlations can lead to oversized systems with excessive capital costs, while underestimation of water losses can compromise long-term operational viability. This study provides validated tools and methodologies that enable geothermal operators and designers to make informed decisions regarding cooling system configuration, thereby optimizing the balance between capital cost, operational efficiency, and water conservation. Furthermore, the demonstrated capability of CFD to predict evaporation losses with high accuracy enables more precise water resource planning, potentially expanding the geographic scope of viable geothermal development to regions where water availability is constrained.

The objectives of this study are threefold: (1) to develop and validate a comprehensive CFD model of an induced draft counter-flow cooling tower representative of geothermal applications, demonstrating model accuracy through comparison with operational data; (2) to conduct systematic parametric analyses examining the effects of key operating variables—specifically hot water inlet temperature, water mass flow rate, and air inlet velocity—on evaporation losses and thermal performance metrics including approach, range, and effectiveness; and (3) to compare CFD predictions with ASHRAE empirical method estimates across the parameter space, identifying conditions under which each method provides accurate predictions and highlighting situations where simplified empirical approaches may lead to significant errors. Through these objectives, this research aims to provide both fundamental insights into cooling tower thermal-hydraulic behavior and practical guidelines for design optimization in geothermal power generation systems.

RESEARCH METHOD

This study focuses on a geothermal power plant located in West Java, Indonesia, recognized as the oldest geothermal field in the country, operating at a capacity of 235 MW and supplying electricity to the Java-Madura-Bali grid. The plant employs a dry steam system and utilizes a wet cooling tower as its primary cooling component. The availability of reinjection water represents a strategic concern for operational sustainability, with condensate serving as one of the main sources. Consequently, investigating water losses and the performance of the cooling system is of significant interest.

The research framework begins with a comprehensive literature review and field survey to gain an in-depth understanding of the cooling system in geothermal power plants. The study utilizes both primary data, including DCS records, log sheets, and heat and mass balance reports, and secondary data such as EPCC drawings, data sheets, manuals, and performance test reports. Based on these datasets, a geometric model of the cooling tower is developed and subjected to mesh quality assessment. If the mesh quality does not meet the required standards, refinements are applied until satisfactory conditions are achieved. Subsequently, appropriate governing equations, material properties, and boundary conditions are defined for the simulation process.

The next phase involves computational analysis using CFD methods. The simulation is executed with proper solution settings and numerical controls, followed by validation through a grid independence test. If the validation criteria are not met, parameter adjustments and iterative computations are performed until reliable results are obtained. Upon successful validation, the findings are interpreted to evaluate cooling tower performance, particularly concerning evaporation losses.

CFD Model

This study employs an existing cooling tower model operating within a geothermal power plant, featuring a multi-cell induced draft counter flow configuration. The selection of this model is based on its widespread application in power plant cooling systems, ensuring that the analysis accurately reflects real operational conditions. The cooling tower serves as a critical component in the condensation process, converting steam into water for reinjection, thereby playing a pivotal role in sustaining plant operations. Detailed technical specifications of the cooling tower, including dimensions, capacity, and operational parameters, are presented in Table 1 and serve as the foundation for geometric modeling and boundary condition definition in the CFD simulation.

Tabel 1. Technical data of the model

Tower Model	**3 x FRP Cooling tower**
Type	Multi-cell induced draft counter flow cooling tower
Design & Operation Condition	
Total circulating water, m3/hr	8.810
Circulating water per cell. m3/hr	2.936,67
Hot (inlet) water temperature, °C	42,7
Cold (outlet) water temp, °C	26
Wet bulb temperature inlet, °C	20
Dry bulb temperature, °C	21,1
Relative humidity	91%
Atmospheric pressure, mbar	850
Tower pump head, meter	8,11
Drift loss, % circulating water flow	0,005
Evaporation loss (at design cond)	2,35
Design seismic load, G	0,467
Tower exposure	Outdoor
Elevation, mASL	1.508,6
Noise level	Max. 85 db +/- 2db @ 1 m away from mechanical equipment. 85 db +/- 2 db @ point 2 m away from basin, 2m height

Although selected model of cooling tower consists of three cells, this study models only a single cell, which is considered representative of the entire system. This approach is commonly adopted in CFD modeling to reduce computational time and resource consumption without compromising the validity of the simulation results. Several factors justify this technique: the geometric design of each cell is identical, including internal configuration, materials, shapes, and dimensions of components such as fans, structural elements, stacks, and fillers; fluid flow distribution is uniform for both water and air;

operational conditions are consistent, encompassing fan speed, water distribution valve settings, and the absence of sedimentation or structural damage; and environmental conditions are homogeneous, including wet bulb temperature, relative humidity, wind speed, and wind direction. These uniformities ensure that a single cell can serve as a valid representation of the entire cooling tower.

3D Model, Boundary Condition, and Key Parameter

In Computational Fluid Dynamics (CFD), boundary conditions are essential for defining the physical constraints at the limits of the computational domain, thereby governing how the fluid interacts with its surroundings. Accurate specification of these conditions is critical to ensure numerical stability and convergence, as well as to produce results that reflect real-world behavior. Without appropriate boundary conditions, the simulation may fail to converge or yield inaccurate predictions of flow and thermal characteristics.

For this study, three primary boundary conditions are applied: inlet, outlet, and wall. The inlet condition specifies the characteristics of the incoming airflow and water spray, including velocity, temperature, and humidity, which are derived from operational data collected on-site. The outlet condition governs the discharge of air from the cooling tower, typically defined by pressure or mass flow constraints to maintain continuity within the system. Wall conditions represent the physical surfaces of the cooling tower structure, where no-slip conditions are imposed for airflow, and heat transfer interactions are modeled based on material properties as presented in Figure 2.

The reference values for these boundary conditions are based on field measurements recorded on September 20, 2022, which serve as the base case for the simulation. These measurements include fan speed, water flow rate, ambient wet bulb temperature, and relative humidity. Iterative computations are performed using these base case parameters, and the results are subsequently validated by comparison with additional operational data obtained from the same geothermal plant. This approach ensures that the simulation framework accurately reflects real operating conditions and provides a reliable basis for analyzing evaporation losses and cooling tower performance. Key paramters that selected to be initial condition is described in Tabel 2.

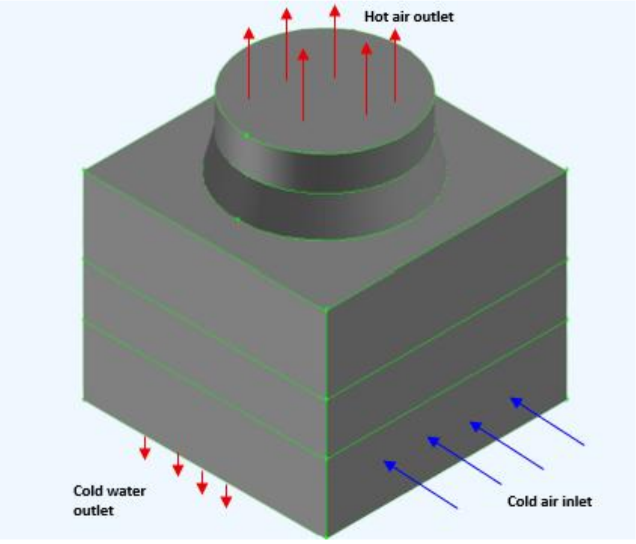

Figure 1. Model and boundary conditions

Tabel 2. Key parameter of boundary conditions

Domain	Boundary condition	Temp. (degC)	Pressure (Pascal)	Mass flow (kg/s)	Velocity (m/s)
Hot water inlet	Pressure inlet	45,2	150.000	1805	1,16
Cold air inlet	Mass flow inlet	18,3	101.325	407.23	3,20
Cold water outlet	Outflow	24,45	-	-	-
Hot air outlet	Pressure outlet	-	-	-	-

Meshing

In Computational Fluid Dynamics (CFD), meshing refers to the process of discretizing the geometric domain into smaller elements (mesh or grid) to enable numerical analysis. These elements form the computational framework that the CFD solver uses to resolve the governing fluid flow equations, such as the Navier–Stokes equations. The meshing stage is critically important because it directly influences the accuracy, stability, and efficiency of the simulation. A finer mesh generally improves solution accuracy by capturing detailed flow and thermal gradients; however, it also increases computational cost and memory requirements. Conversely, poor mesh quality can lead to numerical instability and convergence difficulties, compromising the reliability of the results.

For the selected cooling tower model (Figure. 3), a full hexahedral mesh configuration was employed to ensure high-quality elements and minimize numerical diffusion. The final mesh consists of 21,072 cells, 73,757 faces, and 25,583 nodes, providing an adequate balance between resolution and computational efficiency. This mesh density was determined through preliminary tests to achieve grid independence while maintaining reasonable iteration time. The choice of a structured hexahedral mesh is particularly advantageous for complex geometries such as cooling towers, as it enhances solver stability and accuracy in predicting airflow patterns, water distribution, and heat transfer phenomena within the system.

Mesh Quality Assessment

Evaluating mesh quality is a critical step in CFD modeling, as it directly influences numerical accuracy, convergence behavior, and overall reliability of the simulation. In this study, three primary parameters were employed to assess mesh quality: skewness, orthogonal quality, and aspect ratio. Skewness measures the degree of distortion of mesh elements from their ideal shape; lower values indicate minimal distortion and better element quality. The average skewness obtained for the developed model was 0.05, which, according to the ANSYS Fluent guidelines, falls within the excellent category on the mesh quality spectrum. This result confirms that the mesh elements exhibit negligible geometric distortion, thereby ensuring accurate interpolation and stable numerical performance.

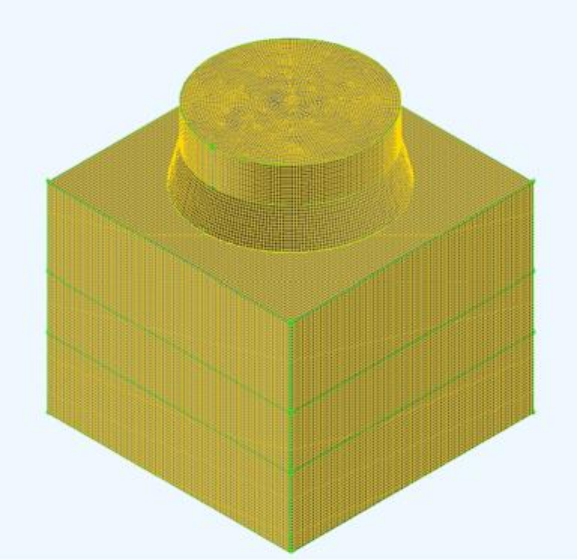

Figure 2. Meshed model of cooling tower with single cell

Orthogonal quality, which evaluates the alignment of mesh elements relative to the flow direction and neighboring cells, is another critical indicator of mesh integrity. The minimum orthogonal quality achieved in this model was 0.92, classified as very good by ANSYS standards. This high value demonstrates that the mesh elements maintain near-ideal orientation, reducing the likelihood of numerical errors associated with poor alignment. Although aspect ratio was also monitored to ensure proportionality of cell dimensions, the consistently favorable skewness and orthogonal quality values indicate that the mesh meets stringent quality requirements. Collectively, these metrics validate the robustness of the meshing process and provide confidence in the accuracy and stability of subsequent CFD computations. Both skewness and orthogonal spectrum can be observed in table 3 and table 4 below.

Table 3. Skewness mesh metric spectrum (www.ansys.com)

Excellent	Very Good	Good	Acceptable	Bad	Unacceptable
0-0,25	0,35-0,5	0,50-0,80	0,80-0,94	0,95-0,97	0,98-1,00

Table 4. Orthogonal mesh metric spectrum (www.ansys.com)

Excellent	Very Good	Good	Acceptable	Bad	Unacceptable
0-0,001	0,001-0,14	0,15-0,20	0,20-0,69	0,70-0,95	0,95-1,00

Grid Independence Analysis

To optimize the simulation process, particularly for complex models with multiple parameter variations, it is essential to perform a grid independence assessment. This procedure ensures that the numerical results are not significantly influenced by the mesh density, thereby achieving an optimal balance between computational efficiency and solution accuracy. Without such an evaluation, simulations may either consume excessive computational resources due to unnecessarily fine meshes or produce unreliable results from coarse discretization.

The most widely accepted approach for this evaluation is the Grid Convergence Index (GCI) method, which provides a quantitative measure of discretization error across different mesh resolutions. The GCI is formulated based on Richardson extrapolation principles and is expressed as:

$$GCI = \frac{1{,}25e_a}{r-1}\,.100\%$$

Where, e_a is approximated relative error and r is grid refinement ratio.

$$e_a^{21} = \left|\frac{\emptyset_1 - \emptyset_2}{\emptyset_1}\right|$$

$\emptyset$ is representatif variable, in this case can be define as cold water – outlet temperature, $\emptyset_1$ is the value of mesh number 1 dan $\emptyset_2$ is the value after refinement.

$$r = \frac{h_{coarse}}{h_{fine}}$$

h is representative mesh size, derived form equation as follow:

$$h = \left[\frac{1}{N}\sum_{i=1}^{N} \Delta V\right]^{1/3}$$

where N is the number of cell and ΔV is volume of cell.

Setting Parameter of CFD Simulation

Continuous Phase Model

In CFD simulations of closed-wet cooling towers, air is modeled as the continuous phase. This model is formulated based on conservation equations for four fundamental aspects: mass, momentum, thermal energy, and water vapor concentration (Gan, 2001). These governing equations are employed to describe the overall airflow behavior within the simulation domain, including the distribution of velocity, temperature, and relative humidity.

The continuous phase model can be expressed through the following governing equations for steady-state, incompressible flow conditions:

$$\nabla\,.\left(\rho\bar{V}\phi - \Gamma_\phi\,\nabla\,\phi\right) = S_\phi + S_\phi^{-p}$$

where ϕ is variable of stream such as mean velocity $\bar{V}$ (m/s), mean enthalpy and mean concentration of water vapor, ρ is density of air (kg/m3), Γ_ϕ is diffusion coeficient (Ns/m2), S_ϕ is source of continue phase and S_ϕ^{-p} is additional source due to interaction between air and water droplet.

Discrate Phase Model

The dispersed phase in this system consists of spherical water droplets suspended in air as the continuous phase. The motion of each droplet is calculated using particle dynamics equations that account for drag force, gravity, relative momentum change, and pressure gradients. Mathematically, the droplet trajectory is determined by the following formulation:

$$\rho_\mathrm{p}\frac{dV_\mathrm{p}}{dt} = \left(\frac{3}{4}\right)\frac{\rho C_D\left|V - V_\mathrm{p}\right|}{d_p}(V - V_\mathrm{p}) + g(\rho_\mathrm{p} - \rho) + \left(\frac{1}{2}\right)\rho\frac{d}{dt}(V - V_\mathrm{p}) + \frac{\partial P}{\partial r_p}$$

In this equation, V dan V_p represent the instantaneous velocities of air and water droplets (m/s), C_D denotes the drag coefficient, d_p is the droplet diameter (m), ρ_p refers to the droplet density (kg/m³), P indicates the static pressure (Pa), dan r_p corresponds to the droplet trajectory (m).

Turbulence Model

Defining the viscous model in fluid flow computation is a critical step, as fluid viscosity significantly influences velocity distribution, pressure fields, and heat transfer within the flow domain. The viscous model captures essential effects such as fluid friction, turbulence, and mixing, which govern flow behavior in the modeling process. In this simulation, the Standard k–ε turbulence model was selected over other viscous models due to its widespread applicability and proven effectiveness for flows with relatively low velocity gradients. The Standard k–ε model belongs to the Reynolds-Averaged Navier–Stokes (RANS) approach, which simplifies turbulence calculations by averaging flow variables over time, thereby reducing computational complexity while maintaining sufficient accuracy for engineering applications.

Performance Indicators of Cooling Tower Systems

Researchers have extensively examined the thermal performance of cooling towers through the development of mathematical models (Hajidavalloo et al., 2010). In 1925, Merkel introduced the thermal evaluation theory for cooling towers, which later became widely recognized and applied (Merkel et al., 1925; Reuter et al., 2012; Naik et al., 2017). Similar to the simplified assumptions in Merkel's method, the effectiveness–Number of Thermal Units (e-NTU) approach was also developed to analyze cooling tower thermal performance (Jaber et al., 1989; Stabat et al., 2004). Furthermore, comprehensive comparisons and in-depth analyses of three mathematical models—the Merkel method, e-NTU method, and Poppe method—have been conducted (Kloppers et al., 2005).

Key performance indicators for cooling towers include range, approach, and effectiveness. The range refers to the temperature reduction between the hot water entering the cooling tower and the cooled water exiting the system. A larger range indicates that the cooling tower effectively reduces water temperature, reflecting superior thermal performance. The approach is defined as the difference between the ambient air wet-bulb temperature and the cold-water outlet temperature. These parameters are complemented by effectiveness, expressed as the ratio of the actual range to the ideal range—the difference between the cooling water inlet temperature and the wet-bulb temperature. Higher effectiveness values signify improved cooling tower performance.

H_2O Loss in Cooling Tower

Despite its advantages over other heat exchanger technologies, wet cooling towers require a substantial water supply, and a portion of this water is inevitably lost during the cooling process. Water losses occur through several mechanisms, including evaporation, blowdown, and drift. To maintain the water mass balance within the cooling tower, it is necessary to replenish these losses with an additional supply known as make-up water.

Evaporation loss is strongly influenced by both environmental conditions and operational requirements. Yuan and Wei (2021) conducted experiments on a natural draft wet cooling tower (NDWCT) and concluded that an increase in dry-bulb temperature leads to higher evaporation losses. Conversely, higher relative humidity reduces evaporation loss, which is accompanied by an increase in the temperature of water discharged into the cooling tower basin.

Yuan and Wei (2020) employed a modified Merkel method and concluded that an increase of 1 °C in ambient air temperature reduces evaporation loss by approximately

26.65 tons per hour (equivalent to about 0.157% of the circulating water). Furthermore, a 1% increase in relative humidity (RH) decreases evaporation loss by around 5.63 tons per hour (approximately 0.033% of the total circulating water). In comparison, the ASHRAE Handbook for HVAC Applications states that evaporation loss typically ranges from 0.1% to 0.3% of the water mass per °F, or approximately 0.18% for each 1 °C change in temperature.

RESULT AND DISCUSSION

Effect of hot-water inlet temperature

Figure 6 demonstrates that both the ASHRAE method and the CFD simulation exhibit a consistent upward trend in evaporation losses with increasing hot water inlet temperature. At 35 °C, the evaporation loss estimated by ASHRAE is 5.5 kg/s, whereas the CFD simulation reports a slightly lower value of 5.0 kg/s. This discrepancy becomes more pronounced at elevated temperatures; for instance, at 41 °C, ASHRAE predicts 8.5 kg/s compared to 7.8 kg/s from CFD. At the highest temperature analyzed (49 °C), the ASHRAE method indicates an evaporation loss of 14.5 kg/s, while CFD yields 13.0 kg/s. These quantitative differences suggest that the ASHRAE approach tends to provide higher estimates than the simulation-based method, which may have implications for design accuracy and operational efficiency in thermal systems.

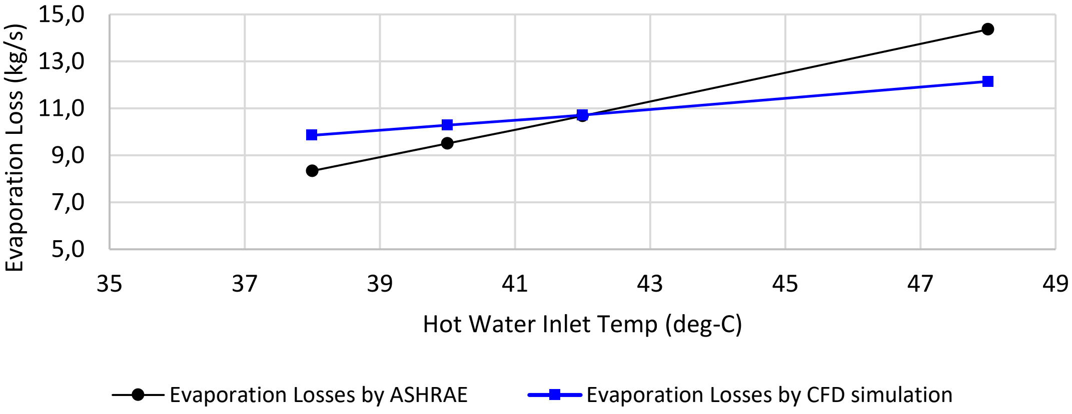

Figure 3. Relationship between hot water inlet temperature and evaporation loss

Figure 3 depicts the effect of hot water inlet temperature on the cold-water outlet temperature and system effectiveness in a wet cooling tower. Within the inlet temperature range of 38 °C to 42 °C, the cold-water outlet temperature increases from approximately 19.5 °C to 21.0 °C, while the system effectiveness decreases from 95% to 85%.

In wet cooling towers, the cold-water outlet temperature is strongly influenced by the ambient air wet-bulb temperature, as cooling occurs through partial evaporation of water into the air. As the hot water inlet temperature rises, the thermal load to be dissipated by the cooling tower also increases. If environmental conditions (particularly air temperature and humidity) remain constant, the tower's ability to reduce water temperature is constrained by the approach temperature, defined as the difference between the cold-water outlet temperature and the wet-bulb temperature. Consequently, an increase in hot water inlet temperature results in a higher cold water outlet temperature, as the system cannot maintain the same degree of cooling due to limitations in heat transfer capacity and ambient conditions.

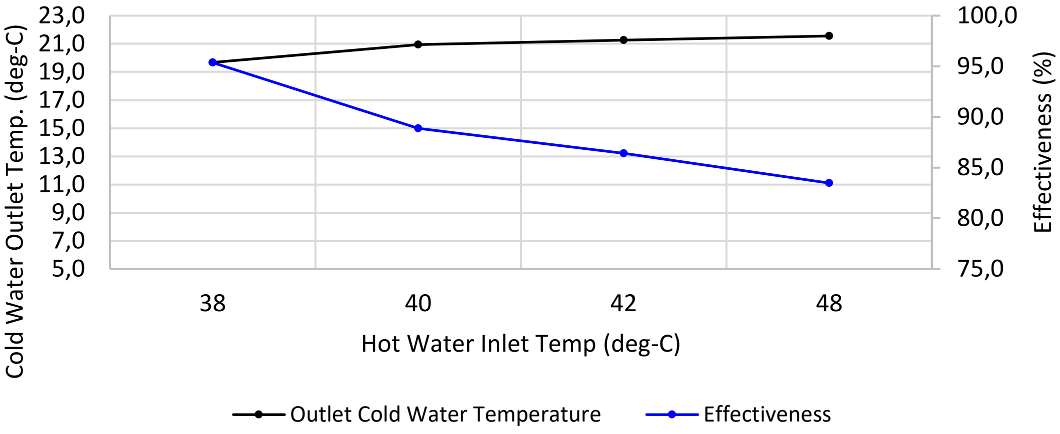

Figure 4. Relationship between hot water inlet temperature and cold-water outlet temperature

The reduction in system effectiveness from 95% to 85% indicates a diminished capability of the cooling tower to approach the wet-bulb temperature at higher inlet temperatures. Assuming a constant wet-bulb temperature, the increase in cold water outlet temperature leads to a lower effectiveness value. This decline may be attributed to several factors, including an evaporation rate that does not proportionally match the increased thermal load, limitations in airflow rate, or reduced performance of the fill media in supporting heat and mass transfer.

Effect of hot-water inlet mass flow rate

Figure 5 illustrates the relationship between hot water mass flow rate (kg/s) and evaporation loss, analyzed using two approaches: the ASHRAE method and CFD simulation. The horizontal axis represents the mass flow rate of hot water within the range of 400 to 850 kg/s, while the vertical axis denotes the evaporation rate in kg/s. Results indicate that the ASHRAE approach produces a linear correlation, where an increase in hot water flow directly raises the amount of evaporated water. In contrast, CFD simulation shows that the evaporation rate remains relatively constant despite increasing mass flow, with only minor fluctuations.

This discrepancy suggests that the ASHRAE method assumes a strictly proportional relationship between hot water flow and evaporation loss, without accounting for saturation effects or mass transfer limitations at the water surface. Conversely, CFD simulation incorporates more detailed physical phenomena, including temperature distribution, turbulence, and fluid-surface interactions, thereby providing a more realistic prediction of actual operating conditions. These findings underscore the importance of employing physics-based approaches such as CFD in complex systems, where empirical methods may lead to overestimation of energy and mass losses.

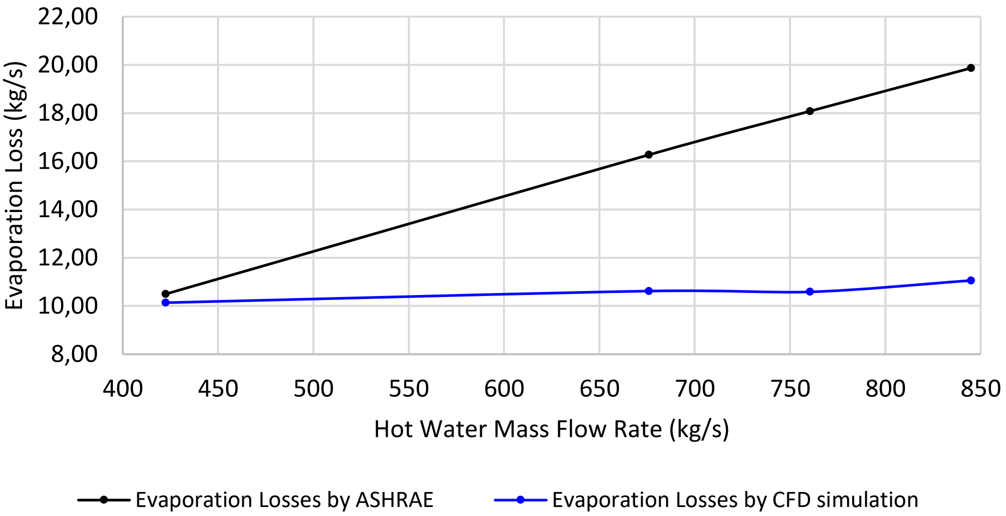

Figure 5. Relationship between hot water mass flow rate and evaporation loss

Figure 6 illustrates the influence of hot water mass flow rate on two primary performance indicators of the heat exchanger system: the cold-water outlet temperature and the system's thermal effectiveness. The horizontal axis represents the hot water mass flow rate ranging from 423 to 845 kg/s, while the left vertical axis shows the cold-water outlet temperature (approximately 17.5 °C to 19 °C) and the right vertical axis indicates system effectiveness (approximately 92% to 86%). The data reveal that increasing the hot water mass flow rate leads to a gradual rise in the cold-water outlet temperature, signifying an increase in heat transfer to the cooling fluid, accompanied by a decline in system effectiveness.

This phenomenon can be explained by energy balance and heat transfer dynamics. As the hot water flow rate increases, the total thermal energy entering the system also rises. However, given the fixed cooling capacity of the tower, the higher flow rate reduces the residence time of water within the tower. Although the larger water volume carries more energy, the system's ability to approach the wet-bulb temperature diminishes, resulting in lower effectiveness. These findings highlight the operational limitations of wet cooling towers under high thermal loads and underscore the importance of optimizing flow conditions to maintain efficiency.

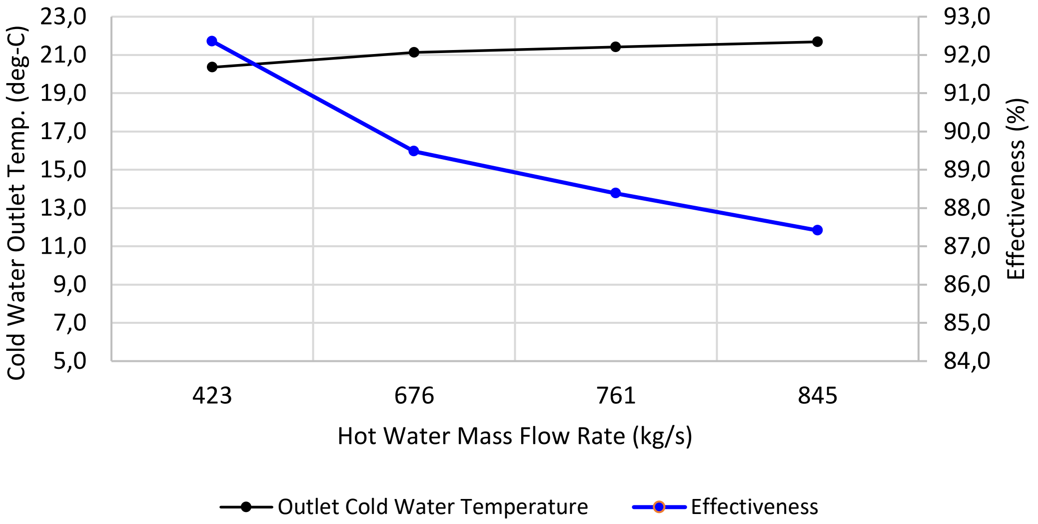

Figure 6. Relationship between hot water mass flow rate and cold-water outlet temperature

Model Visualization during Performance Test

Figure 7 presents the results of a numerical simulation in the form of H_2O mass fraction contours within a fluid domain during a performance test. These contours illustrate the distribution of water vapor concentration in the system, with mass fraction values ranging from 0.015 to 0.125. Blue regions correspond to the lowest mass fraction values, whereas red regions indicate the highest.

The pronounced color variations across certain areas signify significant concentration gradients, which can be interpreted as the result of mixing processes, thermal interactions, or localized temperature differences within the system. Notably, the upper region of the domain exhibits higher H_2O mass fractions (green to yellow), likely representing zones of vapor accumulation due to flow dynamics or thermal effects. This observation suggests that a portion of water vapor is carried out of the system, indicating the occurrence of evaporation loss.

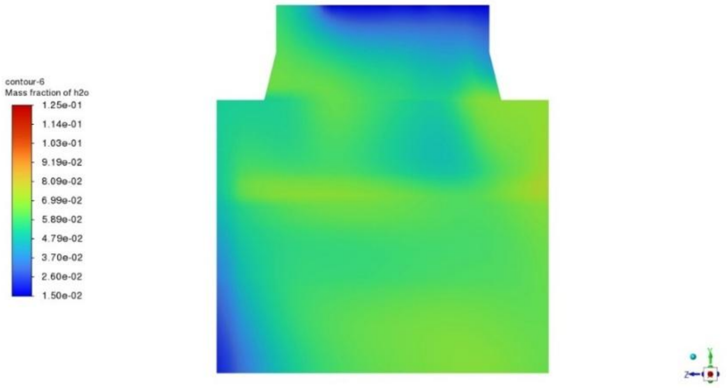

Figure 7. Contour of H_2O mass fraction at performance test condition

The highest temperature (≈40 °C) is concentrated in the upper channel region (dark red), while the lowest temperature (≈11 °C) occurs at the bottom (blue). The pronounced vertical gradient indicates upward heat transfer, likely driven by convection and density-induced fluid movement. Heat accumulation at the top suggests potential stagnation zones, whereas the more uniform temperature pattern in the middle and lower regions reflects improved mixing, possibly due to turbulence and droplet-air interactions within the rain and fill zones. This condition can be clearly observed in the visualization shown in Figure 8.

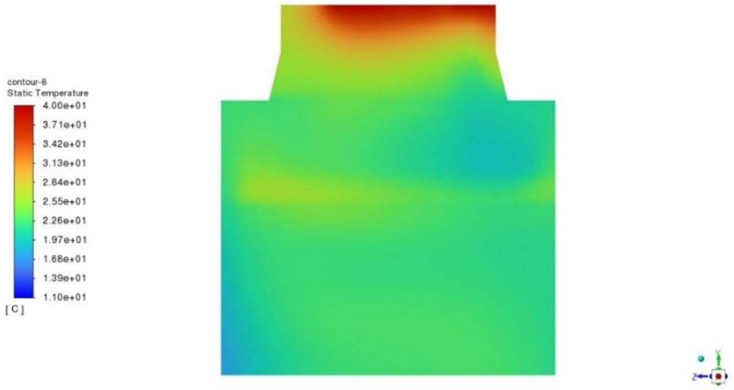

Figure 8. Contour of static temperature at performance test condition

The distribution of H_2O mass, coupled with temperature distribution within the cooling tower, is closely related to the flow patterns occurring in the domain. The velocity contours reveal significant gradients, with red and orange regions indicating high-velocity zones, most likely located along the main flow paths or near the inlet. Conversely, blue and green regions represent stagnant or low-velocity areas, which may indicate flow resistance, recirculation zones, or inefficiencies in fluid distribution. This pattern is critical to analyze as it can influence heat transfer effectiveness, pressure losses, and overall operational stability of the system. Figure 9 clearly illustrates this visualization.

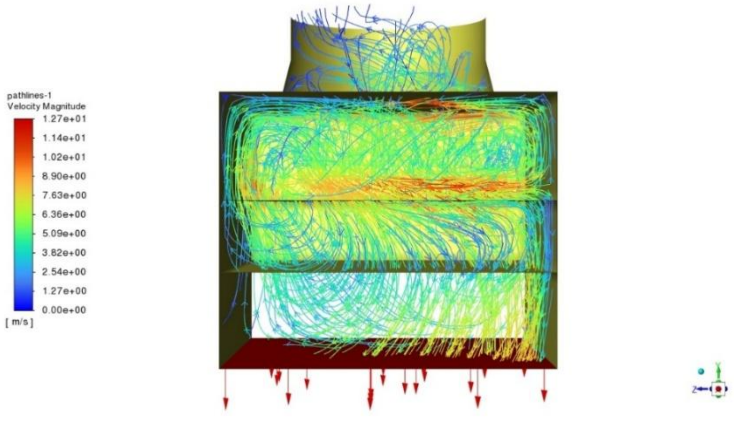

Figure 9. Pathlines for magnitude velocity at performance test

CONCLUSION

This study reveals that elevating hot water inlet temperature from 35°C to 49°C substantially increases evaporation loss—from 5.5 kg/s to 14.5 kg/s (ASHRAE) and 5.0 kg/s to 13.0 kg/s (CFD)—with ASHRAE providing conservative estimates and CFD offering realistic predictions by capturing detailed physics. For hot water mass flow rates rising from 423 kg/s to 845 kg/s, ASHRAE shows linear loss increases, while CFD indicates stable values due to mass transfer limitations; higher flows also lower cold-water outlet temperature (21°C to 11°C) and effectiveness (92% to 86%), reflecting reduced residence time. Validated CFD modeling of an induced draft counter-flow geothermal wet cooling tower achieved 1.8% error against real data, affirming its superiority for design optimization. For future research, integrating machine learning with CFD could enable real-time predictive control of evaporation losses under variable geothermal conditions, enhancing water conservation in water-scarce regions.

REFERENCE

AlGaiar, M. M., & Afolabi, J. (2024). Enhancing geothermal wells integrity: Key considerations and challenges from drilling to production. *Day 2, Wednesday, May 22, 2024*.

Araujo, S. D. O., Lages, A. S., Dutra, J. C. C., Primo, A. R. M., Michalewicz, J. S., Da Costa, J. A. P., Ochoa, A. A. V., Tomás, A. C. C., & Alcântara, S. C. S. (2024). Comparative study of cooling tower fills: Experimental analysis and CFD simulation of an alternative fill. *Journal of the Brazilian Society of Mechanical Sciences and Engineering, 46*(7), 412.

Blecich, P., Senčić, T., Wolf, I., & Bonefačić, I. (2018). Numerical investigation of heat and mass transfer inside a wet cooling tower. *Technical Journal, 12*, 131–138. https://doi.org/10.31803/tg-20171017145907

Chandrasekharam, D., Lashin, A., Al Arifi, N., Al-Bassam, A. M., & Chandrasekhar, V. (2020). Geothermal energy for sustainable water resources management. *International Journal of Green Energy, 17*(1), 1–12.

Elshehabi, T., & Alfehaid, M. (2025). Sustainable geothermal energy: A review of challenges and opportunities in deep wells and shallow heat pumps for transitioning professionals. *Energies, 18*(4), 811.

Eze, V. H. U., Eze, E. C., Alaneme, G. U., & Bubu, P. E. (2025). Recent progress and emerging technologies in geothermal energy utilization for sustainable building heating and cooling: A focus on smart system integration and enhanced efficiency solutions. *Frontiers in Built Environment, 11*, 1594355.

Falcone, G., Beardsmore, G., Conti, P., Kastl, S., Mijnlieff, H., Nádor, A., Ussher, G., Brommer, M., Griffiths, C., & Tulsidas, H. (2025). *8 years on: Incremental impact of worldwide implementation of the United Nations Framework Classification for Geothermal Energy Resources*. Proceedings of the 50th Workshop on Geothermal Reservoir Engineering, Stanford University.

Felicioni, L., Gaspari, J., Veselka, J., & Malík, Z. (2023). A comparative cradle-to-grave life cycle approach for addressing construction design choices: An applicative case study for a residential tower in Aalborg, Denmark. *Energy and Buildings, 298*, 113557.

Guan, Z., Hooman, K., & Gurgenci, H. (2016). Dry cooling towers for geothermal power plants. In *Alternative energy and shale gas encyclopedia*. https://doi.org/10.1002/9781119066354.ch32

Heidari, R., Dioguardi, E., Sethuvenkatraman, S., & Braslavsky, J. H. (2024). Evaluating advanced HVAC control benefits in operational buildings using historic data—A case study. *Applied Thermal Engineering, 252*, 123611.

Imani, E., Dawood, H., Williams, S., & Dawood, N. (2025). Physics-based and data-driven retrofitting solutions for energy efficiency and thermal comfort in the UK: IoT-validated analysis. *Buildings, 15*(7), 1050.

Jradi, M. (2026). Unlocking the potential of digital twin technology for energy-efficient and sustainable buildings: Challenges, opportunities, and pathways to adoption. *Sustainability, 18*(1), 541.

Kabeyi, M. J. B. (2019). Geothermal electricity generation, challenges, opportunities and recommendations. *International Journal of Advances in Scientific Research and Engineering, 5*(8), 53–95.

Kang, Z., Zhou, X., Yan, D., & An, J. (2025). Development and evaluation of cooling tower performance prediction using physics-informed neural networks. *Energy and Buildings*, 116101.

Kassem, M. A., & Moscariello, A. (2024). Advancing sustainable energy: A systematic review of geothermal-powered district heating and cooling networks. *International Journal of Sustainable Energy, 43*(1), 2417436.

Khan, F., Raza, A., Mahmoud, M., Patil, S., & Al Shafloot, T. (2025). A comprehensive review of geothermal reservoirs: Current understanding and future outlook. *Arabian Journal for Science and Engineering*, 1–25.

Kim, J. H., Kim, Y. S., Jo, H. G., Urabe, E., Mun, J., Shin, Y., Park, Y., & Park, C. S. (2025). Accuracy, generalizability, and extrapolation ability of physics-based, data-driven, and hybrid models for real-life cooling towers. *Building and Environment, 274*, 112756.

Roshan Kharrat, R., Perfetto, G., Ingaramo, R., & Mutani, G. (2025). Optimizing building performance with dynamic photovoltaic shading systems: A comparative analysis of six adaptive designs. *Smart Cities, 8*(4), 127.

Santoso, R., Hoteit, H., & Vahrenkamp, V. (2019). Optimization of energy recovery from geothermal reservoirs undergoing re-injection: Conceptual application in Saudi Arabia. In *SPE Middle East Oil and Gas Show and Conference*.

Yuan, W., Sun, F., Liu, R., Chen, X., & Li, Y. (2021). Effect of change factors on evaporation loss based on cold end system of natural draft counter-flow wet cooling towers. *Journal of Thermal Science and Technology, 16*(2), JTST0015. https://doi.org/10.1299/jtst.2021jtst0015

Zargar, A., Kodkani, A., Vickers, B., Flynn, M. R., & Secanell, M. (2023). A hybrid cooling tower model for plume abatement and performance analysis. *Applied Thermal Engineering, 219*, 119593. https://doi.org/10.1016/j.applthermaleng.2022.119593