사물인터넷융복합논문지 Vol. 11, No. 3, pp. 29-41, 2025 https://doi.org/10.20465/KIOTS.2025.11.3.029

Analysis and Optimization of Multi-Field Coupling Characteristics of Flow Field in Evaporative Cooling Tower Based on FLUENT

Gu Haiqin[1], Byung-Won Min[2*]
[1]Ph.D. Student, Department of IT Engineering, Mokwon University; Teacher,Nantong Institute of Technology
[2]Professor, Department of Game Software Engineering, Mokwon University

FLUENT 기반의 증발 냉각탑의 유동장 다중 물리장 연계 특성 분석 및 최적화에 관한 연구

고해진[1], 민병원[2*]
[1]목원대학교 IT공학과 박사과정; 중국난통이공학원 교수, [2]목원대학교 게임소프트웨어공학과 교수

Abstract This study focuses on the evaporative cooler employed in the dry dust removal system of a steel plant. Finite element models of the evaporative cooling tower and atomizing nozzles were developed, and the internal flow field of the tower was analyzed numerically using Fluent. Two nozzle arrangement schemes were investigated through CFD(Computational Fluid Dynamics) simulations, with a comprehensive analysis conducted on flow field characteristics, heat and mass transfer behavior, and ash deposition risk. The results indicate that Scheme II exhibits superior overall performance, characterized by well-developed turbulent mixing, effective control of high-temperature zones, and a spatially reasonable gradient distribution of H_2O mass fraction. A multi-parameter coupling analysis method integrating flow field, temperature field, species field, and particle field was established, providing a theoretical basis for the optimization design of evaporative cooling towers. The findings support the three-dimensional modeling and real-time monitoring of flue gas separation systems, and lay a foundation for the integration of 5G-enabled smart manufacturing and the application of digital twin technology in production process simulation, thereby contributing to the advancement of intelligent manufacturing in the steel industry.

Key Words : Basic oxygen furnace, dry dust removal, evaporative cooling tower, atomizing nozzle, CFD

요 약 본 연구는 제철소의 건식 집진 시스템에 사용되는 증발식 냉각기를 대상으로 한다. 증발식 냉각탑과 분무 노즐의 유한 요소 모델을 구축하고, Fluent를 이용하여 냉각탑 내부의 유동장을 수치적으로 분석하였다. 두 가지 분무 노즐 배치 방안을 CFD 시뮬레이션을 통해 조사하였으며, 유동장 특성, 열 및 물질 전달 거동, 그리고 재비산 축적 위험에 대한 종합적인 분석을 수행하였다. 연구 결과, 제2안이 전반적인 성능에서 우수한 것으로 나타났으며, 이는 잘 발달된 난류 혼합 특성, 고온 영역의 효과적인 제어, 그리고 H_2O 질량 분율의 공간적 구배 분포로 특징지어진다. 본 연구는 유동장, 온도장, 조성장, 입자장의 다중 파라미터 연계 해석 방법을 확립하였으며, 이는 증발식 냉각탑 최적 설계의 이론적 기초를 제공한다. 이 연구 결과는 배연가스 분리 시스템의 3차원 모델링 및 실시간 모니터링에 기여하며, 5G 기반 스마트 제조와 디지털 트윈 기술의 생산 공정 시뮬레이션 적용을 위한 기반을 마련하는 데 중요한 역할을 한다. 이는 제철 산업의 지능형 제조 발전에 실질적인 기여를 할 것으로 기대된다.

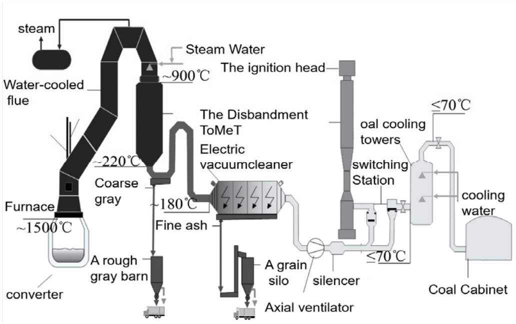
주제어 : 전로; 건식 집진, 증발식 냉각탑, 분무 노즐, CFD

*교신저자 : 민병원(minfam@mokwon.ac.kr)
접수일 2025년 04월 17일 수정일 2025년 05월 31일 심사완료일 2025년 06월 13일

1. Introduction

Since the implementation of China's 13th Five-Year Plan, the nation has prioritized environmental protection alongside economic development. Regulatory authorities across various industrial sectors have successively introduced and enforced a series of standards and guidelines aimed at promoting low-pollution, low-emission production practices. In response to increasingly stringent environmental regulations, dry dedusting systems have attracted growing attention due to their superior performance in energy efficiency and emission control, prompting many steel enterprises to retrofit their existing wet dedusting systems with dry alternatives. In the steelmaking process, gas recovery constitutes the highest share of energy consumption—approximately 54%—followed by steam recovery (21%) and steam consumption (6%) [1][2].

The steelmaking process produces a substantial volume of high-temperature flue gas. A typical dry dedusting system consists primarily of flue gas purification and gas recovery units. The high-temperature flue gas first enters an evaporative cooler for initial cooling and coarse particulate removal, and then proceeds to an electrostatic precipitator for fine particle capture. The purified gas that meets quality standards is directed to the gas holder via a switching station, while the substandard portion is discharged through flare combustion via a chimney. A detailed process flow is illustrated in Figure 1. The evaporative cooling tower is a critical component in the dry dedusting system of the converter, primarily functioning to rapidly reduce the temperature of high-temperature flue gas and achieve preliminary coarse particulate removal through spray atomization. By facilitating phase-change heat transfer between water and flue gas, the system significantly lowers the gas temperature to within the allowable operational range of downstream equipment. Concurrently, it enhances turbulence within the flue gas flow, promoting the settling and removal of large particulate matter.

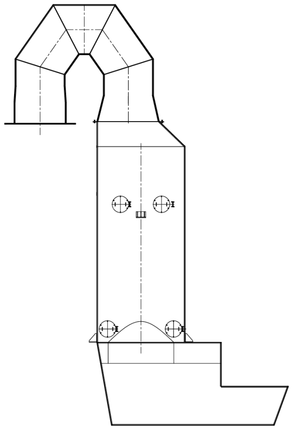


[Fig. 1] Process flow diagram of dry dedusting system for converter

In the evaporative cooling tower, particulate matter from high-temperature flue gas often deposits on internal surfaces, causing ash buildup. This reduces heat transfer efficiency and disrupts flow, weakening the spray system's cooling and atomization. As a result, the dedusting process becomes less efficient and less stable. Ash accumulation also speeds up equipment wear, raises maintenance costs, and may cause unexpected shutdowns. Poor cooling due to ash increases the load on downstream electrostatic precipitators, reducing their effectiveness and risking gas recovery safety. Therefore, controlling ash buildup is essential in full-dry converter dedusting systems and should be a key focus in nozzle design and CFD analysis [3].

2. Overview of Evaporative Cooling Tower

The evaporative cooling device primarily consists of six components: the main equipment body, ladder platform, nozzles, water system, air system, and control system. One of the key technologies in the evaporative cooling system is the evaporative cooler, which is designed to precisely cool flue gas through mist spraying[4]. The amount of water sprayed is adjusted according to the heat content of the flue gas, with the sprayed water completely vaporizing into steam. A schematic diagram of the structure of the evaporative cooler is shown in Figure 2.

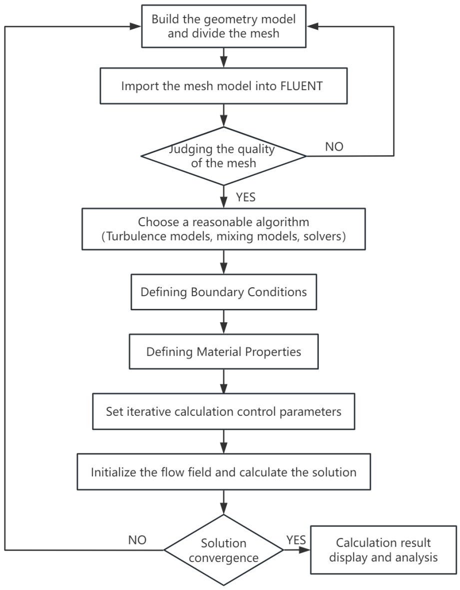


[Fig. 2] Schematic Diagram of Evaporative Cooler Structure

The evaporative cooling tower utilizes a distributed arrangement of nozzles to spray mist. Under the action of the mist, the liquid droplets in the spray are broken into micron-sized particles, which increases the contact surface area between the droplets and the flue gas, enhancing the heat and mass transfer between them. This process rapidly cools the flue gas, reducing the temperature of the high-temperature, dust-laden flue gas from 800–1000°C to 240–260°C. The arrangement and tilt angle of the nozzles in the evaporative cooler affect the atomization effect of the flue gas[5]. An improper tilt angle may cause droplets to collide with the tower walls, reducing cooling efficiency, and improper handling may quickly lead to the formation of scaling on the tower walls. Therefore, this study, based on Fluent, investigates the effects of different numbers of nozzles and installation angles of the nozzles on parameters such as velocity, temperature, H2O mass distribution, droplet diameter variation, droplet evaporation time, ash deposition on the tower walls, and outlet temperature through numerical simulations, leading to specific conclusions[5].

3. Introduction to CFD

Computational Fluid Dynamics (CFD) is a tool for numerical simulation and analysis of fluid dynamics problems using electronic computers and discretized numerical methods. The basic approach to solving CFD problems is as follows: physical quantities in the time domain are discretized using mathematical methods, represented by differential and difference equations, and the corresponding solutions to the multivariable linear equations derived from the transformed differential equations are obtained using a computer[6]. Data simulation and modeling are then performed using FLUENT, and the computational process is shown in Figure 3.



[Fig. 3] Calculation flow chart of FLUENT[5]

3.1 Model Assumptions

The reliability of the Fluent analysis results depends on appropriate simulation assumptions.

Although the composition and temperature of the flue gas at each stage in the converter dry dust removal system can vary, in this simulation analysis, the flue gas is treated as an incompressible model, and the effect of the spray initiation point on the overall flow field of the flue gas is neglected[7].

In this simulation, high-temperature dust-laden flue gas is modeled as the continuous phase, while the atomized droplets from the nozzles are treated as the discrete phase. As a result, the interaction between the droplets and the flue dust, as well as the influence of the droplet volume fraction on the continuous phase of the high-temperature flue gas, are ignored. The basic equations of the particle tracking model are as follows[8]:

A. Particle Continuity Equation (Mass Conservation)

$$\frac{d\rho_k}{dt}+\frac{\vartheta}{\vartheta x_j}(\rho_k v_{ij})=S_k$$

Nomenclature:

ρ_k: Density of the particulate phase (kg/m³)

v_{ij}: Velocity component of particles in the x_j direction (m/s)

S_k: Source term for particle mass (accounting for evaporation/ condensation effects)

B. Particle Momentum Equation

$$\frac{\vartheta}{\vartheta t}(\rho_k v_{ii})+\frac{\vartheta}{\vartheta x_j}(\rho_k v_{ii} v_{jj})= \rho_k g_i+\frac{\rho_k(v_i-v_{ii})}{\tau_{ik}}+v_i S_k+F_{kM}$$

Nomenclature:

v_{ii}: Velocity component of particles in the x_i direction (m/s)

g_i: Gravitational acceleration component in the x_i direction (m/s^2)

τ_{ik}: Particle relaxation time (s)

F_{kM}: Additional forces (e.g., Magnus force, Saffman lift force)

C. Particle Energy Equation

$$\frac{\vartheta}{\vartheta t}(\rho c_p T)+\frac{\vartheta y}{\vartheta x_j}(\rho v_j c_p T) =n_k(Q_t-Q_k-Q_{tk})+c_p T S_k$$

Nomenclature:

c_p: Specific heat capacity of particles (J/(kg·K))

T: Particle temperature (K)

Q_t: Heat exchange between particles and surrounding medium (W)

Q_k: Internal energy change of particles (W)

3.2 Simulation Data

In this simulation, the droplets ejected from the nozzle are treated as the discrete phase, while the flue gas is treated as the continuous phase.

The object of this simulation is the flue gas purification and dust removal system of a converter at a steel plant in Hebei Province, China. This study specifically analyzes the impact of different nozzle distributions and installation angles on the cooling performance of the equipment. The equipment parameters are provided in Table 1, based on preliminary data collection and analysis.

〈Table 1〉 Equipment Parameters of EC Unit

Item	Parameter
Flue duct diameter	2300 mm
EC tower diameter	4200 mm
EC tower inlet flue gas volume	325,000 m³/h
Flue duct inlet gas velocity	21.74 m/s
EC tower inlet gas temperature	1050°C / 1323 K
EC tower flue gas composition	CO–50%, N_2–30%, CO_2–20%

In the evaporative cooler of the dry dust removal system, the arrangement of nozzles is a key factor in ensuring efficient atomization, uniform cooling, and stable system operation. The following factors need to be comprehensively considered:

A. The nozzle arrangement must ensure that the atomized water covers the entire tower cross-section, avoiding the formation of "dry zones" or "wet zones" which could lead to uneven cooling of the flue gas. The atomization cone angles of adjacent nozzles should partially overlap to ensure there are no dead zones. The injection angle should be adjusted based on the tower shape to prevent droplets from directly spraying onto the tower wall, which may cause scaling or corrosion.

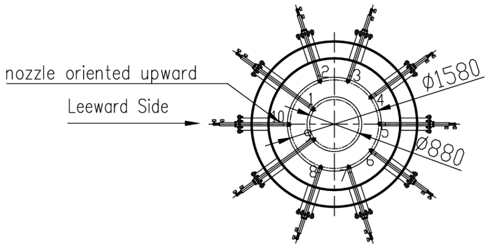
B. The nozzle injection velocity should be coordinated with the flue gas velocity to ensure the droplets remain in the tower for sufficient time to evaporate (typically 0.5–2 seconds).

C. The nozzle arrangement should be linked with temperature sensors to dynamically adjust the spray volume. The nozzle positions should be easy to maintain and work in conjunction with a filtration system to prevent particulate blockages.

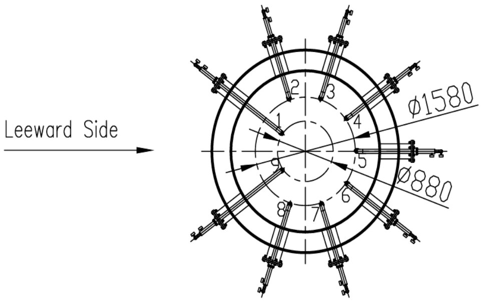
Based on preliminary data collection and analysis of actual operating conditions, combined with technical consultations with the nozzle manufacturer regarding product performance specifications and atomization characteristics, and taking into account the structural features and process requirements of the evaporative cooling tower, two nozzle layout schemes were designed[9]. Considering the above factors and actual operating conditions, the specific parameters of the EC tower gas-mist nozzles are provided in Table 2, based on communication with the nozzle manufacturer, using steam as the atomization gas source, the droplet atomization size is taken as 260 μm. To achieve better evaporation effects in practical applications, engineers designed two sets of nozzle distribution patterns, with the specific arrangements shown in Figures 4 and 5.

〈Table 2〉 EC Tower Gas-Mist Nozzle Parameters

Item	Parameter
Total water consumption of EC tower gas-mist nozzles	27.5 m^3/h (Single nozzle flow rate: 0.76 kg/s)
Atomization angle of EC tower gas-mist nozzles	40°
Water temperature of EC tower gas-mist nozzles	45°C / 318 K
Nozzle distribution and insertion length in flue duct	See below (Pattern 1/Pattern 2)
Nozzle tilt angle	0°
Droplet atomization particle size	260 μm



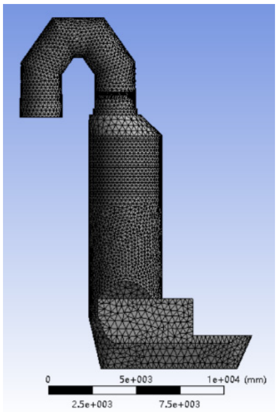
[Fig. 4] Nozzle Arrangement Pattern 1
(10 Nozzles / One Nozzle Facing Up)



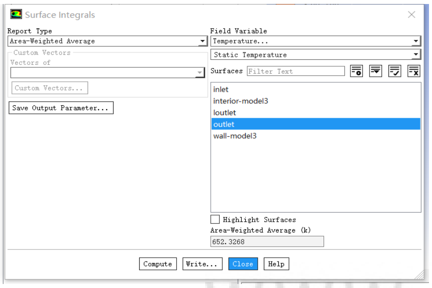
[Fig. 5] Nozzle Arrangement Pattern 2
(9 Nozzles / All Facing Down)

3.3 3D Modeling, Mesh Generation, Boundary Conditions

Based on the above equipment parameters, a 3D model of the evaporative cooler was created, followed by mesh generation. In CFD simulations, the quality of the mesh directly affects the accuracy, convergence, and computational efficiency of the results[10]. In the study of ash accumulation in evaporative cooling towers, the simplified model reduces computational complexity by neglecting the interactions between droplets and particulate matter as well as the influence of droplets on the continuous flue gas phase. While this simplification may lead to an incomplete understanding of the ash formation mechanisms, it facilitates rapid simulation and parameter optimization[11]. Future research should incorporate experimental data for validation and adjustment to enhance the model's accuracy and engineering applicability. High-quality meshes more accurately reflect flow characteristics, while low-quality meshes may lead to simulation failure or incorrect conclusions[12]. Based on previous experience, ANSYS Meshing was used to generate structured meshes, and the mesh quality was checked using Fluent's Mesh Quality panel. Subsequently, boundary conditions and solver settings were configured to provide guidance and support for subsequent flow field analysis. Figure 6 shows the mesh generation results, and Figure 7 shows the boundary conditions setup.

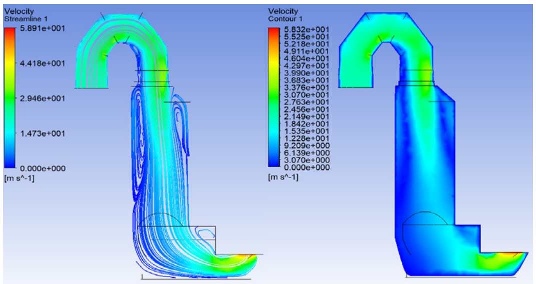


[Fig. 6] Mesh Generation Results



[Fig. 7] Boundary Conditions Setup

To verify the simulation, a preliminary simulation was conducted with the gas-mist nozzles inactive, analyzing the velocity streamlines and velocity contour maps of the EC cross-section. The results showed that the Fluent tool can function properly with the initial parameter settings, as shown in Figure 8.



[Fig. 8] EC Tower Cross-Section Velocity Streamlines / Velocity Contour Map (Gas-Mist Nozzles Inactive)

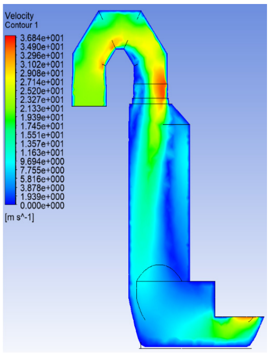
3.4 Selection of Atomized Droplet Diameter

In the flow field analysis of the evaporative cooler tower (EC tower), the choice of atomized droplet diameter directly impacts the evaporation efficiency, flow field uniformity, and system energy consumption. The core principles for selecting the droplet diameter include maximizing evaporation efficiency (where droplet evaporation time is proportional to the square of the droplet diameter), flow field compatibility, and balancing pressure drop with energy consumption[8]. Based on engineering experience and the actual average droplet diameter of the gas-mist nozzles in this project, a droplet diameter of 260μm was used for the simulation.

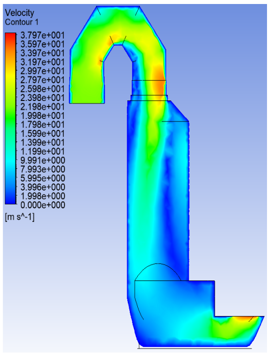
4. Simulation Results Analysis

4.1 Velocity Contour Simulation Results Analysis

In the CFD flow field analysis of the Evaporative Cooling Tower, velocity contour analysis is one of the core diagnostic tools. Its significance lies in its direct impact on flow characteristics, heat and mass transfer efficiency, and equipment performance. Velocity contours visually display the distribution of velocity magnitudes at various points in the flow field using color gradients. This can be used to analyze the uniformity of droplet flow, identify low-speed or high-speed regions, and prevent flow dead zones that could lead to local temperature rises or particle deposition[13]. Additionally, it can represent momentum transfer efficiency, verifying the intensity of interaction between flue gas and atomized droplets. Velocity contours can also serve as a basis for structural optimization, guiding modifications to the EC tower's structure, especially the dimensions of the inlet and outlet[9]. Figures 9 and 10 show the simulation results of velocity contours under different nozzle configurations.



[Fig. 9] Velocity Contour Simulation Results for Scheme 1



[Fig. 10] Velocity Contour Simulation Results for Scheme 2

Based on the CFD velocity contour simulation results for the different spray gun configurations, the following conclusions can be drawn:

〈Table 3〉 Velocity Distribution Comparison

Parameter	Scheme 1	Scheme 2	Difference Analysis
Maximum Velocity	36.84 m/s	37.97 m/s	Spray jet in Scheme 2 is stronger or more compact.
Low-Speed Region (〈5 m/s)	Larger at the bottom (5.8 m/s decrease)	Reduced at the bottom (6.0 m/s increase)	New configuration improves bottom flow stagnation.
Velocity Gradient	Steeper gradient (36.8 → 0 m/s)	Smoother gradient (37.9 → 0 m/s)	New configuration allows better flow development, reducing shear forces.

A. Spray Jet Core Area

Scheme 1: High-speed areas (〉30 m/s) are concentrated in the middle, which may lead to "jet penetration," resulting in insufficient droplet dispersion.

Scheme 2: High-speed areas are more distributed, but the peak speed is slightly higher, which requires checking for potential excessive droplet fragmentation (smaller particle size, leading to faster evaporation).

B. Bottom Low-Speed Area

Scheme 1: Bottom velocities are generally 〈10 m/s (minimum 0 m/s), which may cause particle deposition or local hot spots.

Scheme 2: The minimum velocity is increased to 1.998 m/s, showing that the new configuration enhances bottom flow by adjusting the spray angle/height.

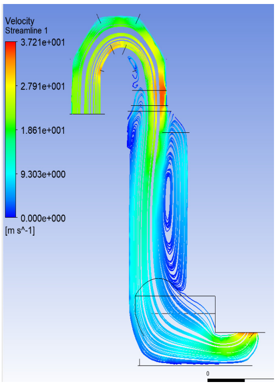
C. Wall Boundary Layer

Both figures show a sharp drop in velocity near the walls (〈2 m/s), but the boundary layer in Figure 10 is slightly thinner (1.998 vs. 1.939 m/s), indicating that the new configuration may have optimized the wall flow. However, further improvement is possible: to further reduce the boundary layer, increasing wall roughness or adding guide vanes could be explored.

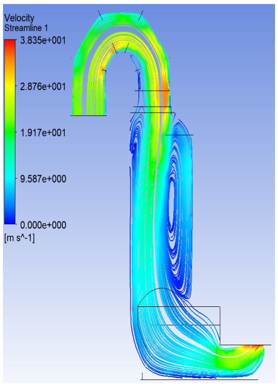
From the analysis above, the following conclusions can be drawn: The new configuration performs better in improving bottom flow and boundary layer performance, but attention must be paid to the energy consumption and wear risks in the high-speed region.

4.2 Analysis of Velocity Streamline Simulation Results

In the CFD flow field analysis of the Evaporative Cooling Tower (EC Tower), streamline analysis is a key method for revealing the three-dimensional flow structure, diagnosing complex vortices, and optimizing the design. Unlike velocity contour maps (which show scalar distribution), streamline maps track the movement trajectory of fluid micro-units, visually presenting core issues such as backflow zones, dead zones, droplet trajectory prediction, and separation point identification. When combined with the Discrete Phase Model (DPM), streamlines can predict whether droplets are entrained in vortices[14]. If the streamlines sharply curve downstream of the spray gun, it indicates that the droplets may hit the wall without fully evaporating, the detailed analysis results can be seen in Figures 11 and 12.



[Fig. 11] Velocity Streamline Simulation Results for Scheme 1



[Fig. 12] Velocity Streamline Simulation Results for Scheme 2

To evaluate the aerodynamic performance of the two nozzle distribution configurations (Scheme A and Scheme B) within the evaporative cooling tower (EC tower), a detailed comparative analysis of the streamline velocity field was conducted. The engineering implications derived from this analysis are outlined as follows.

A. Velocity Distribution Characteristics

Table 4 presents a comparative assessment of key parameters derived from the velocity streamline results for both nozzle arrangements.

〈Table 4〉 Comparative streamline parameters between Scheme 1 and Scheme 2

Parameter	Scheme 1	Scheme 2	Difference and Engineering Implications
Low-velocity region	Broader area 〈 9.30 m/s	Slightly reduced area 〈 9.59 m/s	Scheme B moderately improves low-velocity zones. However, near-zero velocity regions persist at the bottom in both cases, indicating a need for flow-guiding or purging mechanisms.
Velocity gradient transition	Smoother gradient (37 → 0 m/s)	Steeper gradient (38 → 0 m/s)	Scheme B exhibits stronger jet momentum and elevated shear stress. While this enhances mixing, excessive turbulent kinetic energy (recommended threshold: k 〈 0.5 m^2/s^2) should be avoided.

B. Flow Structure Diagnosis and Performance Evaluation

a. Jet Core Region

In Scheme 1, streamlines in the central region form a concentrated and elongated jet core, indicating potential for excessive penetration and insufficient droplet dispersion. The divergence of streamlines downstream of the nozzles suggests increased turbulence and mixing. However, this may induce secondary droplet breakup, reducing droplet size and potentially leading to premature evaporation.

Conversely, Scheme 2 shows a broader distribution of high-velocity regions with a slightly higher peak velocity. This configuration enhances mixing uniformity, although further evaluation is needed to prevent over-fragmentation of droplets and ensure optimal heat and mass transfer.

b.Recirculation and Vortex Identification

Both configurations exhibit closed streamline loops near the bottom of the tower (velocity ≈ 0 m/s), confirming the existence of recirculation zones prone to particle accumulation. These dead zones necessitate the integration of deflector cones or purging air systems. Furthermore, adjusting the nozzle inclination angle (e.g., downward by 15°) could help disrupt symmetric vortex formation and promote flow uniformity.

In Scheme 2, streamline intersections in the upper region are notably fewer, implying reduced large-scale vortex intensity and, consequently, lower pressure losses.

c. Wall Flow Attachment Characteristics

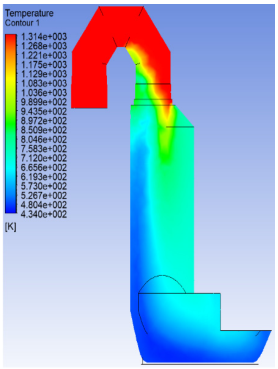
In Scheme 1, partial detachment of near-wall streamlines—particularly near the outlet—indicates potential flow separation, which could elevate local thermal resistance. Scheme 2 demonstrates improved streamline adherence along the wall surfaces; however, high-velocity streamlines near the wall (especially on the right side) may induce erosion, necessitating further structural reinforcement or flow diffusion measures.

Scheme 2 demonstrates superior performance in terms of turbulent mixing and vortex suppression, with potential benefits for thermal uniformity and operational efficiency. However, attention must be paid to the increased jet velocity, which may raise energy consumption and wall erosion risks. In contrast, Scheme 1 offers gentler flow gradients, rendering it suitable for low-pressure-drop applications, although recirculation mitigation strategies are required.

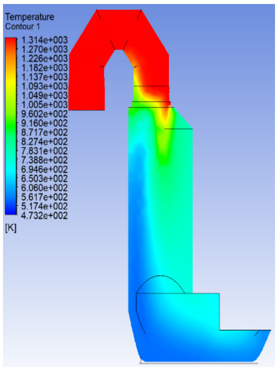
4.3 Temperature Contour Analysis

In the CFD flow field analysis of the Evaporative Cooling Tower (EC Tower), temperature contour analysis serves as a critical tool for evaluating thermodynamic performance, optimizing heat transfer efficiency, and identifying potential design deficiencies[15]. The temperature distribution directly reflects the evaporation efficiency of the system. Specifically, low-temperature regions (blue zones) that coincide with spray zones indicate effective heat absorption and evaporation of droplets. Conversely, localized high-temperature zones (red regions) suggest potential flow stagnation areas or insufficient spray coverage, which may lead to thermal stress-induced cracking of tower materials—particularly when temperatures exceed 300 °C.

Furthermore, banded high-temperature regions in the temperature contour map often signify excessive spacing between nozzles, pointing to an opportunity for layout optimization. Temperature contours support key design decisions by: (1) verifying whether the cooling efficiency meets the design targets, (2) identifying regions of overheating or insufficient evaporation, and (3) providing guidance for nozzle arrangement and tower structural modifications. Figures 13 and 14 present the temperature distribution results corresponding to the two proposed nozzle configurations.



[Fig. 13] Simulated temperature contour results under spray layout Scheme 1.



[Fig. 14] Simulated temperature contour results under spray layout Scheme 2.

〈Table 5〉 Comparison of temperature distribution characteristics between spray layout Scheme 1 and Scheme 2.

Parameter	Scheme 1	Scheme 2	Interpretation of Differences
Maximum Temperature	1314 K (1041 °C)	1314 K (1041 °C)	Identical inlet temperatures, consistent with design specifications.
Minimum Temperature	434 K (161 °C)	473 K (200 °C)	Scheme II demonstrates a higher minimum temperature, which may be attributed to inadequate droplet evaporation or uneven spray distribution.
Axial Temperature Gradient	880 K (1314 → 434 K)	841 K (1314 → 473 K)	Scheme II exhibits a slightly attenuated temperature gradient; further investigation is required to determine whether the upward displacement of the low-temperature zone results in insufficient heat exchange at the bottom.
High-Temperature Region (〉1000 °C) Proportion	Significant (concentrated near the inlet)	Slightly reduced (more localized)	Scheme II offers marginally improved control over high-temperature flue gas; however, potential risks of localized overheating warrant further monitoring.

In Scheme 1, the low-temperature zone (〈600 K) is more widely distributed, with the minimum temperature (434 K) located near the bottom of the tower. This may be attributed to excessive cooling caused by droplet settling, which increases the risk of corrosion or wet ash accumulation at the bottom region. In contrast, Scheme 2 exhibits an upward shift of the low-temperature zone (minimum 473 K), indicating more efficient droplet evaporation in the middle and upper sections of the tower. However, the relatively higher bottom temperature suggests a potential deficit in cooling performance in that area.

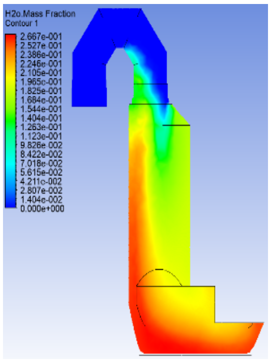
In Scheme 1, the high-temperature zone (〉1000 °C) extends toward the middle of the tower, implying that the spray jets may not fully intercept the core region of the high-temperature flue gas. By comparison, Scheme 2 shows a more localized high-temperature region concentrated near the inlet, suggesting that the spray arrangement is more effective in capturing the high-temperature gas immediately upon entry.

The temperature field in Scheme 1 exhibits significant stratification along the axial direction, possibly due to an excessive vertical spacing between spray layers. Scheme II demonstrates a more continuous thermal transition; however, localized “hot spots” (e.g., on the right side) are observed, indicating potential areas of incomplete mixing or inadequate spray coverage.

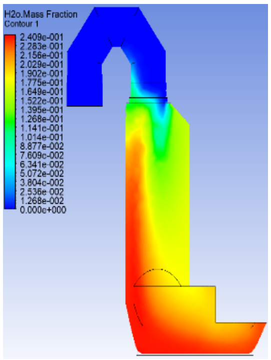
In summary, Scheme 2 demonstrates superior performance in terms of high-temperature gas control and thermal uniformity, although attention should be given to localized overheating regions.

4.4 In-Depth Analysis of H_2O Mass Fraction Distribution

The H_2O mass fraction distribution serves as a crucial metric for evaluating phase-change efficiency, mass transfer characteristics, and the optimization of flow dynamics. A comparative analysis of the two nozzle arrangements based on H_2O distribution is summarized below:



[Fig. 15] Mass fraction distribution of H_2O – Simulation result for Scheme 1



[Fig. 16] Mass fraction distribution of H_2O – Simulation result for Scheme 2

〈Table 6〉 Mass fraction distribution of H_2O between spray layout Scheme 1 and Scheme 2.

Parameter	Scheme 1	Scheme 2	Interpretation
Maximum H_2O mass fraction	0.2667 kg	0.2409 kg	Scheme II exhibits higher overall humidity, indicating increased moisture content.
Minimum H_2O mass fraction	0.1404 kg	0.1268 kg	Both schemes feature dry regions, suggesting areas of insufficient evaporation.
High-humidity region (〉0.2 kg/kg)	Widely distributed (from inlet to mid-section)	Concentrated in upper-middle region	Scheme II presents a more concentrated high-humidity zone, reflecting improved gas encapsulation.
Gradient transition	Smooth	Steep	Scheme II displays a steeper gradient, implying accelerated droplet evaporation rates.

In Scheme 1, the high-H_2O region (depicted in red) extends toward the tower bottom, indicating strong droplet penetration. However, the sharp decline in H_2O mass fraction near the bottom suggests inadequate evaporation in that region. In Scheme IIR, the high-humidity region is more concentrated in the upper–middle section of the tower, indicating that droplets stay longer in the core evaporation zone, where evaporation is more effective. Thus, Scheme 2 outperforms in evaporation symmetry and gas encapsulation.

Water vapor plays a critical role in evaporative cooling towers by facilitating both rapid cooling and particulate removal. Through phase change and latent heat absorption, it effectively reduces the temperature of high-temperature flue gas, while also promoting the agglomeration and settling of particulate matter, thereby enhancing dedusting efficiency[16]. However, uneven water distribution may lead to the formation of localized dry or wet zones, resulting in non-uniform cooling, flow instability, and increased risks of ash deposition and scaling. Therefore, optimizing nozzle configuration and atomization parameters to ensure uniform water distribution is essential for achieving efficient operation, suppressing ash accumulation, and minimizing maintenance costs.

5. Conclusion

This study conducted a comprehensive CFD numerical simulation based on Fluent to analyze the coupled flow characteristics, heat and mass transfer behavior, and ash deposition risk of an evaporative cooling tower under two different spray nozzle configurations. The interaction mechanisms between key flow dynamics and phase-change processes were elucidated. The results indicate that Scheme II exhibits superior overall performance: its streamline structure demonstrates enhanced turbulent mixing, more effective control of high-temperature flue gas (with the high-temperature zone concentrated near the inlet and a 5% reduction in axial temperature gradient), and a more reasonable spatial distribution of H_2O mass fraction (with the high-humidity zone concentrated in the core evaporation area). However, issues such as low velocity (<2 m/s) and low humidity ($H_2O \approx 0$ kg/kg) at the tower bottom still require optimization, potentially through adjusting the inclination angle of lower-layer nozzles or adding guiding structures.

In contrast, Scheme 1 features a more uniform axial temperature transition (temperature difference of 880 K) and broader evaporation coverage. However, it suffers from notable lateral non-uniformity (12% higher humidity on the left side compared to the right) and an overcooling issue at the bottom (434 K), which may increase the risk of corrosion. Notably, both configurations exhibit elevated particle concentrations near the wall regions (locally >0.2 kg/m^3), providing a clear direction for optimizing anti-deposition design.

The multi-parameter coupling analysis method established in this study—integrating flow field, temperature field, species field, and particle field—offers a theoretical basis for the refined design of industrial-scale evaporative cooling towers. Future work will incorporate transient simulations and material erosion experiments to further verify the long-term operational reliability of the nozzle arrangements.

Future experimental validation efforts should focus on empirically evaluating the key assumptions of the CFD model. A small-scale evaporative cooling tower test platform can be constructed, utilizing high-speed imaging and laser particle sizing systems (e.g., PDA) to measure droplet distribution and atomization quality under different nozzle configurations. Coupled with thermocouple arrays and infrared thermography, these results can be compared with the temperature fields and H_2O mass fraction distributions predicted by the CFD model. Additionally, by setting different flue gas compositions, temperatures, and particle concentrations, the distribution and deposition rate of ash within the tower can be monitored. Surface analysis methods such as SEM or EDS can then be used to evaluate wall surface corrosion and particle accumulation, providing validation data for the particle field simulations.

In the context of digital twin development, the CFD model established in this study can serve as a physically driven core, operating in conjunction with real-time data from industrial sites. Real-time input of sensor data—such as flue gas temperature, humidity, and spray pressure—into the CFD platform allows for dynamic updates of boundary conditions and source terms, enabling continuous self-calibration of the model. Additionally, by leveraging cloud computing platforms and 3D visualization engines, simulation results can be mapped onto a digital twin interface to support virtual nozzle layout optimization, fault prediction, and energy efficiency assessment. The integration of AI algorithms for trend forecasting and anomaly detection further enables real-time diagnosis and optimization of control strategies. This provides a reliable foundation for full-process simulation, remote monitoring, and energy management in 5G-enabled smart factories in the steel industry.

References

[1] Lan F, Fang X, Zhou G N. Design analysis of evaporative cooling device in converter gas dry

system [J]. Industrial Heating, 2025, 54(02): 42-45.

[2] Liu C P, Ma G Y, Yuan L, et al. Comprehensive energy-saving technologies in steelmaking [J]. Metallurgical Energy, 2017, 36(01): 3-10.

[3] Shi L, Hao D B, Wang H B. CFD-based simulation and application of atomizing nozzles in evaporative cooling tower [J]. Heavy Machinery, 2024(04): 6-12.

[4] H. Montazeri, B. Blocken, J.L.M. Hensen,Evaporative cooling by water spray systems: CFD simulation, experimental validation and sensitivity analysis,Building and Environment,Volume 83,2015,129-14

[5] Cai X J, Yuan Z L. Study on turbulence diffusion model of fine particulate matter in demisters [J]. Journal of Central South University (Science and Technology), 2018, 49: 290-299.

[6] Gu H Q,Min B W. Flow field simulation and structural optimization design of cyclone separator based on Fluent[J].Journal of Internet of Things and Convergence. [Internet]. 2024, 10(5): 73-85.

[7] Zhang Z, Xie Z L. Numerical simulation of fluid-solid two-phase flow [J]. Journal of Chemical Industry and Engineering (China), 2019, 52(01): 1-12.

[8] Montazeri H., Blocken B., Hensen J. L. M. CFD analysis of the impact of physical parameters on evaporative cooling by a mist spray system [J]. Applied Thermal Engineering, 2015, 75: 608-622.

[9] Montazeri H., Blocken B., Hensen J. L. M. Evaporative cooling by water spray systems: CFD simulation, experimental validation and sensitivity analysis[J]. Building and Environment , 2015, 83: 129-141.

[10] Cucumo M. A., Ferraro V., Galloro A., Gullo D., Kaliakatsos D., Nicoletti F. Computational fluid dynamics simulations to evaluate the performance improvement for air-cooler equipped with a water spray system[J]. Tecnica Italiana – Italian Journal of Engineering Science , 2019, 63(2-4): 158-166.

[11] Vala K. V., Makwana M. A., Nukasani S. Validation of evaporative cooling system using CFD analysis [J]. International Journal of Current Microbiology and Applied Sciences , 2019, 8(03): 393-399.

[12 Zhou N., Chen F., Cao Y., Chen M., Wang Y. Experimental investigation on the performance of a water spray cooling system [J]. Applied Thermal Engineering , 2017, 112: 1117-1128.

[13] Marek Wasilewski and Lakhbir Singh Brar. "Performance analysis of the cyclone separator with an ovel clean air inlet installed on the roof surface" in Powder Technology, Vol. 428, pp. 118, 2023.

[14] Long, H. M., Ding, L., Zhao, H. X., et al. Research progress on CO emission reduction in flue gas of typical iron and steel production processes [J]. Iron & Steel, 2023, 58(08): 1-12.

[15] He, K. Study on energy-saving and efficiency-increasing potential of energy regulation and process configuration optimization in metallurgical system [D]. Beijing: University of Science and Technology Beijing, 2019.

[16] Long, H. M., Ding, L., Zhao, H. X., et al. Research progress on CO emission reduction in flue gas of typical iron and steel production processes [J]. Iron & Steel, 2023, 58(08): 1-12. 1.

고 해 진(Gu Haiqin) [정회원]

- September 2008- July 2012, Taiyuan University of Technology, Bachelor's Degree in Mechanical Design manufacture and Automation.
- September 2012- December 2015, Graduated with a Master's degree in Mechanical Engineering from Taiyuan University of Technology.
- June 2021 - Present, has been pursuing Ph.D. in Intelligent Fusion in IT at Mokwon University, Daejeon, Korea.

〈관심분야〉
Computer software development, mechanical Engineering, robot control technology

민 병 원(Byung-Won Min) [정회원]

- He received M.S. degree in computer software from Chungang University, Seoul, Korea in 2005.
- He received Ph.D. degree in the dept. of Information and Communication Engineering, Mokwon University, Daejeon, Korea, in 2010.
- He is currently a professor of Mokwon University since 2010.

〈관심분야〉
digital communication systems, Big Data