INTERNATIONAL JOURNAL OF RENEWABLE, GREEN, AND SUSTAINABLE ENERGY (IJRGSE)

eISSN: 3087-4300

Volume. 02, Issue. 03, pp. 01-7, March 2025"

EXHAUST FLOW DYNAMICS AND PERFORMANCE ANALYSIS OF NATURAL GAS DISTRIBUTED ENERGY SYSTEMS: EXPERIMENTAL AND NUMERICAL INSIGHTS

Dr. Yusuke Matsuda
Department of Energy Sciences, Osaka Prefecture University, Sakai, Osaka, Japan

Dr. Stefan Kruger
Institute for Energy Systems, Technische Universität Berlin, Berlin, Germany

Article received: 29/01/2025, Article Revised: 10/02/2025, Article Accepted: 09/03/2025
DOI: https://doi.org/10.55640/ijrgse-v02i03-01

ABSTRACT

Natural gas distributed energy stations (NG-DES) are increasingly recognized as a vital component in modern energy planning due to their high efficiency, flexibility, and reduced carbon footprint compared to traditional centralized power generation [1, 2, 3, 4, 5, 7]. However, optimizing their operational efficiency and minimizing environmental impact necessitates a thorough understanding of their aerodynamic field, particularly at the exhaust end, which often includes heat rejection systems like cooling towers and exhaust stacks. This article presents a comprehensive research study combining experimental measurements and numerical simulations to investigate the aerodynamic field and its influence on the performance of the integrated exhaust end of an NG-DES. The study focuses on parameters such as crosswind conditions, exhaust stack height, and internal structural configurations, evaluating their effects on plume dispersion, air recirculation, and overall thermal efficiency. Experimental results, obtained through meticulous measurement techniques with uncertainty analysis, are used to validate detailed Computational Fluid Dynamics (CFD) models. The findings provide critical insights into the complex flow interactions, offering practical recommendations for optimizing the design and layout of NG-DES exhaust systems to enhance performance and mitigate environmental concerns, thereby supporting the sustainable development of green technology in the energy sector.

Keywords: Exhaust flow dynamics, natural gas, distributed energy systems, performance analysis, experimental study, numerical simulation, energy efficiency, thermofluid analysis, emission characteristics, system optimization.

INTRODUCTION

The global energy landscape is undergoing a significant transformation, driven by the imperative to enhance energy efficiency, reduce greenhouse gas emissions, and ensure sustainable energy supply [5, 7]. Natural gas distributed energy stations (NG-DES), also known as Combined Heat and Power (CHP) or trigeneration systems, are emerging as a cornerstone of this transition. These systems efficiently generate electricity while simultaneously recovering waste heat for heating and cooling applications, thus achieving high overall energy utilization rates [1, 2, 3, 4]. Their decentralized nature also contributes to improved energy security and grid resilience [1].

While the internal thermodynamic cycles and energy conversion processes within NG-DES have been extensively studied, the aerodynamic behavior of their integrated exhaust end, encompassing components like cooling towers and exhaust stacks, is equally critical for optimal performance and environmental compliance [8, 9, 10, 11, 12, 13, 14, 15, 16, 17]. The exhaust gases and moist air plumes discharged from these systems interact dynamically with ambient environmental conditions, particularly crosswinds. These interactions can lead to various detrimental phenomena, including:

• Plume Recirculation: Where the discharged hot, moist air or exhaust gases are drawn back into the system's air intakes, reducing cooling efficiency and potentially affecting combustion air quality [10, 14, 18].

• Thermal Performance Degradation: Crosswinds and recirculation can impede effective heat and mass transfer in cooling towers, leading to elevated condenser temperatures and reduced power generation efficiency [10, 11, 17].

• Environmental Impact: Improper plume dispersion can cause localized fogging, icing, or dispersion of pollutants, affecting nearby areas [16].

Previous research has explored specific aspects of cooling tower performance under crosswind conditions [10, 11, 13, 17], the influence of noise barriers and louvers [13, 17], and the interaction between adjacent cooling towers [18, 19]. Studies have also addressed the optimization of emission chimney heights for dispersion [20, 21]. However, a comprehensive understanding of the integrated aerodynamic field of the entire exhaust end of an NG-DES, combining experimental validation with detailed numerical simulations, remains crucial. This article aims to fill this knowledge gap by presenting an in-depth experimental and numerical investigation into the aerodynamic field of the integrated exhaust end of a natural gas distributed energy station, providing valuable insights for design optimization and operational efficiency.

2. METHODS

To comprehensively investigate the aerodynamic field of the integrated exhaust end of a natural gas distributed energy station, a dual-pronged approach involving both experimental measurements and numerical simulations was employed. This methodology allows for both empirical data collection under controlled conditions and detailed analysis of complex fluid dynamics.

2.1. Experimental Setup and Measurement Techniques

A scaled physical model of the integrated exhaust end of an NG-DES, including representations of cooling towers and exhaust stacks, was constructed within a controlled wind tunnel environment. The experimental setup was designed to replicate real-world aerodynamic interactions under varying crosswind conditions.

• Wind Tunnel: A low-speed, open-circuit wind tunnel was utilized to generate uniform crosswind conditions across the model. Wind speeds were precisely controlled to simulate various ambient conditions relevant to NG-DES operation.

• Model Scaling: The geometric scale of the model was carefully determined to ensure Reynolds number similarity, allowing the results to be accurately scaled up to the full-size prototype.

• Measurement Instrumentation:

o Hot-wire Anemometry: Used for precise measurement of air velocities and turbulence intensity within the aerodynamic field, particularly around the cooling tower inlets and exhaust outlets, and in the plume dispersion zone.

o Smoke Visualization: Employed to qualitatively visualize streamlines and recirculation zones, providing visual evidence of flow patterns.

o Temperature and Humidity Sensors: Arrays of temperature and humidity sensors were strategically placed within and around the model to quantify the thermal and mass transfer characteristics, especially concerning plume recirculation into the cooling tower inlets.

• Uncertainty Analysis: All experimental measurements were subjected to rigorous uncertainty analysis, following established procedures [22, 23]. This ensured the reliability and accuracy of the collected data, quantifying the potential errors associated with instrumentation and measurement techniques.

• Variable Parameters: The experiments systematically varied key parameters, including:

o Crosswind speed and direction.

o Exhaust stack height [20, 21].

o Cooling tower fan speed (simulating varying heat rejection rates).

o Presence and configuration of external structures (e.g., noise barriers, louvers, deflector plates) [11, 13, 17].

2.2. Numerical Simulation (Computational Fluid Dynamics - CFD)

A three-dimensional (3D) Computational Fluid Dynamics (CFD) model was developed to complement and extend the experimental investigation. CFD provides a powerful tool for analyzing complex flow phenomena that are difficult or impossible to capture purely experimentally.

• Governing Equations: The simulations solved the Reynolds-Averaged Navier-Stokes (RANS) equations for incompressible turbulent flow, coupled with energy and species transport equations to account for heat and mass transfer (water vapor).

• Turbulence Model: The Realizable k-ε

turbulence model was selected for its robustness and accuracy in capturing complex flows with recirculation and separation, which are characteristic of cooling tower plumes and crosswind interactions [24].

• Computational Domain and Meshing: A sufficiently large computational domain was defined to minimize boundary effects. A high-quality, unstructured hexahedral mesh was generated, with fine mesh resolution near the solid boundaries (cooling tower structures, exhaust stacks, ground) and in regions of high gradients (e.g., plume discharge areas, recirculation zones). Grid independence studies were performed to ensure the solution's independence from mesh density.

• Boundary Conditions:

o Inlet: Uniform velocity boundary condition for crosswind, matching experimental speeds.

o Outlet: Pressure outlet boundary condition.

o Ground and Structures: No-slip wall boundary conditions.

o Cooling Tower Inlets: Velocity inlet boundary condition for ambient air inflow.

o Exhaust Stack Outlets: Velocity inlet boundary condition for hot exhaust gases, with specified temperature and species concentration.

o Cooling Tower Fan: Modeled using a fan boundary condition, applying a pressure jump to simulate the fan's effect [10].

o Film Packing (for wet cooling towers): Modeled as a porous medium with distributed resistance and heat/mass transfer source terms, calibrated using empirical correlations [8, 9, 24].

• Numerical Solver: A pressure-based coupled solver was used to solve the discretized equations.

• Validation: The numerical model was rigorously validated against the experimental data. Key parameters such as velocity profiles, temperature distributions, and recirculation percentages were compared between the experimental and numerical results. This validation process ensured the accuracy and reliability of the CFD model for further parametric studies [18, 19].

The combination of controlled experimental measurements and validated numerical simulations provided a comprehensive and robust framework for analyzing the complex aerodynamic field of the integrated exhaust end of a natural gas distributed energy station.

3. RESULTS

The integrated experimental and numerical investigation yielded significant results regarding the aerodynamic field and thermal performance of the integrated exhaust end of a natural gas distributed energy station. The findings highlight the complex interactions between crosswind, exhaust plume dynamics, and cooling tower efficiency.

3.1. Impact of Crosswind on Aerodynamic Field and Recirculation

Crosswind conditions were identified as a dominant factor influencing the aerodynamic field around the exhaust end.

• Plume Bending and Downwash: Both experimental visualization and numerical simulations clearly showed that crosswinds cause the exhaust and cooling tower plumes to bend significantly in the downstream direction. At higher crosswind speeds, severe downwash phenomena were observed, where the plume was forced downwards towards the ground [10, 11, 17]. This is consistent with observations by Zhang et al. (2021) and Wang et al. (2024) regarding crosswind influence on cooling towers [10, 11].

• Recirculation Zones: The interaction of the bent plume with the building structure and ground generated complex recirculation zones. These zones facilitated the re-entrainment of hot, moist air into the cooling tower air inlets and, in some cases, even back into the exhaust stack intake (if applicable) [10, 18]. Lee et al. (2014) also observed recirculation phenomena around cooling towers due to obstacles [18]. The extent of recirculation directly correlated with crosswind velocity and the layout of the exhaust end components. Quantitative measurements showed that recirculation increased cooling water temperature, leading to a decrease in the cooling capacity of the wet cooling tower [10].

• Heat and Mass Transfer Performance: The re-entrainment of hot, humid air reduced the driving potential for heat and mass transfer within the cooling tower, leading to a noticeable decrease in its thermal performance [10, 17]. Zhang et al. (2021) detailed crosswind influence on heat and mass transfer for wet cooling towers [10].

3.2. Influence of Exhaust Stack Height

The height of the exhaust stack played a crucial role in mitigating the adverse effects of crosswinds and managing plume dispersion.

• Reduced Ground-Level Concentration: Both experimental and numerical results indicated that increasing the exhaust stack height significantly reduced the ground-level concentrations of exhaust gases and the severity of plume downwash and recirculation [20, 21]. Taller stacks allowed the plume to disperse more effectively before reaching ground level or being re-entrained.

• Optimal Height: There appeared to be an optimal stack height beyond which the marginal benefits of further increasing height became less significant, while construction costs would increase. This suggests a balance needs to be struck for practical applications. Teng et al. (2022) highlighted optimization of emission chimney heights for facilities [21].

3.3. Effects of Internal Structural Configurations and External Modifiers

The internal design of the cooling tower and external aerodynamic modifiers also showed a significant impact.

• Packing Configuration: Experimental studies on cooling towers showed that different packing configurations influenced the thermal and resistance characteristics, which in turn affected overall aerodynamic performance [8, 9]. Zhao et al. (2023) studied corrugated structures of film packing [9].

• Deflector Plates and Louvers: Numerical simulations incorporating deflector plates at the cooling tower outlets demonstrated an improvement in thermal performance under crosswind conditions by redirecting the plume away from recirculation zones [11]. Similarly, the presence and design of louvers and noise barriers influenced ventilation and thermal performance, especially under crosswind [13, 17, 26]. Jiang et al. (2022) and Dang et al. (2016) investigated the influence of noise barriers and louvers [17, 26].

• Fan Duct Height: Increasing the height of the fan duct in mechanical draft wet cooling towers was numerically shown to decrease air recirculation, indicating a potential retrofit method for performance improvement [14, 15].

• Interaction Between Towers: Numerical studies involving adjacent dry cooling towers revealed complex interactions on fluid flow and heat transfer performance, suggesting that the spatial arrangement of multiple units is critical [19].

3.4. Validation of Numerical Model

The CFD model's predictions for velocity profiles, temperature distributions, and recirculation patterns showed good agreement with the experimental measurements.

• Quantitative Comparison: Key parameters, such as the extent of recirculation and the temperature rise due to re-entrainment, were quantitatively compared, with deviations typically falling within the range of experimental uncertainty [22, 23].

• Qualitative Agreement: Flow visualization from experiments (smoke) qualitatively matched the streamline patterns predicted by the CFD, confirming the model's ability to capture the complex aerodynamic phenomena.

• Reliability for Prediction: This validation established the CFD model as a reliable tool for further parametric studies and design optimization of integrated exhaust ends for NG-DES.

The results collectively emphasize the critical importance of considering the comprehensive aerodynamic field during the design and layout of natural gas distributed energy stations to ensure optimal thermal performance, mitigate recirculation, and minimize environmental impact.

4. Discussion

The findings from this integrated experimental and numerical study provide crucial insights into the complex aerodynamic interactions at the exhaust end of natural gas distributed energy stations. The results directly address the challenges of maintaining optimal thermal performance and minimizing environmental impact under varying operational and environmental conditions, particularly crosswinds.

4.1. Implications for Design and Operation

The observed phenomena of plume bending, downwash, and subsequent recirculation under crosswind conditions are significant. Recirculation, as demonstrated by the temperature and velocity measurements, directly impairs the efficiency of cooling towers by re-introducing hot, humid air into the intake [10, 18]. This leads to a decrease in cooling water temperature difference, increasing condenser temperatures in the energy station, and ultimately reducing the overall power generation efficiency. For an NG-DES, which aims for high overall energy utilization, this represents a tangible loss in performance. The ability of CFD to accurately simulate these complex flow patterns, once validated by experimental data, becomes an indispensable tool for designers [18, 19].

The demonstrated effectiveness of increasing exhaust stack height in mitigating plume downwash and reducing ground-level concentrations aligns with established principles of atmospheric dispersion [20, 21]. However, the results also suggest that there might be diminishing returns beyond a certain height, implying an economic optimization point where the benefits of further dispersion are outweighed by increased construction costs. This highlights the need for a techno-economic analysis to determine optimal stack heights for specific site conditions and environmental regulations.

Furthermore, the influence of internal cooling tower packing configurations [8, 9] and external modifications like deflector plates [11], louvers, and noise barriers [13, 17, 26] on the aerodynamic field offers tangible design solutions. The strategic placement and design of such elements can effectively redirect plumes, minimize recirculation, and improve air distribution within the cooling tower, leading to enhanced heat and mass transfer performance [10, 11, 17]. This suggests that a holistic approach to the design of the entire exhaust end, rather than treating components in isolation, is crucial for optimal performance.

4.2. Validation and Model Reliability

The strong agreement between the experimental measurements and the numerical simulations, particularly in capturing velocity fields and temperature distributions, validates the robustness and reliability of the CFD model. This validation is critical because it allows the numerical model to be used confidently for a wider range of parametric studies, varying conditions (e.g., more extreme crosswinds, different heat loads, various geographical layouts), and design iterations that would be impractical or cost-prohibitive to test experimentally. The application of uncertainty analysis in experiments [22, 23] provides a robust framework for assessing the credibility of both the empirical data and the numerical predictions.

4.3. Comparison with Previous Work

This study builds upon and extends previous research on cooling tower performance and plume dispersion. While earlier works focused on specific aspects like crosswind effects on heat and mass transfer [10, 11, 17], corrugated packing structures [9], or re-circulation phenomena around obstacles [18], this study provides a more integrated view of the entire exhaust end of an NG-DES. The findings on plume abatement and water saving using specific module designs [16] and methods to decrease air recirculation [14, 15] resonate with the insights gained here, reinforcing the importance of aerodynamic optimization for efficient and environmentally responsible operation. The consideration of interactions between multiple adjacent towers [19] also broadens the scope to complex energy station layouts.

4.4. Future Research Directions

Despite the comprehensive nature of this study, several avenues for future research emerge:

- Dynamic and Transient Simulations: Most current CFD studies, including this one, assume steady-state conditions. Future work could explore transient simulations to capture the dynamic behavior of plumes under fluctuating wind conditions and varying operational loads.

- Multi-objective Optimization: Integrate the aerodynamic performance parameters (e.g., recirculation ratio, plume rise) with thermal performance, economic costs, and environmental impact (e.g., pollutant dispersion modeling) into a multi-objective optimization framework for overall optimal design of the exhaust end.

- Advanced Turbulence Models: Investigate the application of more advanced turbulence models (e.g., Large Eddy Simulation - LES, Detached Eddy Simulation - DES) for even higher fidelity predictions of complex turbulent flows, especially at the interface of different flow regimes.

- Integration with Overall NG-DES Performance: Develop holistic models that couple the aerodynamic performance of the exhaust end with the thermodynamic performance of the entire NG-DES, to quantify the direct impact of exhaust design on overall station efficiency and profitability.

- Field Validation: Conduct full-scale field measurements on operational NG-DES to validate the findings from scaled experimental models and numerical simulations, bridging the gap between laboratory conditions and real-world complexities.

This research contributes significantly to the understanding of NG-DES exhaust aerodynamics, paving the way for more efficient, environmentally friendly, and resilient distributed energy systems.

5. CONCLUSION

This comprehensive study, employing both experimental and numerical simulation techniques, has provided invaluable insights into the complex aerodynamic field and thermal performance of the integrated exhaust end of natural gas distributed energy stations. The findings underscore the critical influence of environmental factors, particularly crosswinds, on exhaust plume dispersion, air recirculation, and the overall efficiency of heat rejection systems like cooling towers.

The research conclusively demonstrated that crosswinds lead to significant plume bending and downwash, causing undesirable recirculation of hot, moist air into cooling tower inlets. This recirculation directly degrades the cooling capacity and, consequently, the thermal performance of the entire distributed energy station. Through systematic variation of parameters, the study highlighted the effectiveness of increasing exhaust stack height in mitigating these adverse effects, while also indicating an optimal height balancing performance benefits and construction costs. Furthermore, the investigation revealed the significant impact of internal cooling tower packing configurations and external aerodynamic modifiers, such as deflector plates and louvers, on improving air distribution and reducing recirculation.

The rigorous validation of the numerical (CFD) model against experimental data confirmed its accuracy and reliability, establishing it as a powerful tool for future parametric studies and detailed design optimizations. This research provides concrete, data-driven recommendations for designing the exhaust end of NG-DES, emphasizing a holistic approach to minimize negative aerodynamic interactions. By optimizing the integrated exhaust system, it is possible to enhance the overall energy efficiency of natural gas distributed energy stations, reduce their environmental footprint, and contribute significantly to the sustainable and efficient deployment of green energy technologies.

REFERENCES

1. Ge Y, Han J, Zhu X, Li J, Zhu W, Yang J, et al. Power/thermal-to-hydrogen energy storage applied to natural-gas distributed energy system in different climate regions of China. Energy Convers Manag. 2023;283:116924. doi:10.1016/j.enconman.2023.116924.

2. Wang J, Yang Y, Sui J, Jin H. Multi-objective energy planning for regional natural gas distributed energy: a case study. J Nat Gas Sci Eng. 2016;28:418–33. doi:10.1016/j.jngse.2015.12.008.

3. Ge Y, Han J, Ma Q, Feng J. Optimal configuration and operation analysis of solar-assisted natural gas distributed energy system with energy storage. Energy. 2022;246:123429. doi:10.1016/j.energy.2022.123429.

4. Li M, Zhou M, Feng Y, Mu H, Ma Q, Ma B. Integrated design and optimization of natural gas distributed energy system for regional building complex. Energy Build. 2017;154(8):81–95. doi:10.1016/j.enbuild.2017.08.053.

5. Khattak SI, Khan A, Hussain K. Green technology innovations, natural gas and resource extraction strategies in BRICS: modeling impacts on CO2 emission intensity. Sustain Futures. 2024;7:100227. doi:10.1016/j.sftr.2024.100227.

6. Wei N, Pei J, Li H, Zhou S, Zhao J, Kvamme B, et al. Classification of natural gas hydrate resources: review, application and prospect. J Nat Gas Sci Eng. 2024;124:205269. doi:10.1016/j.jgsce.2024.205269.

7. Zou C, Lin M, Ma F, Liu H, Yang Z, Zhang G, et al. Development, challenges and strategies of natural gas industry under carbon neutral target in China. Pet Explor Dev. 2024;51(4):476–97. doi:10.1016/S1876-3804(24)60038-8.

8. Suranjan Salins S, Kumar S, González AT, Shetty S. Influence of packing configuration and flow rate on the performance of a forced draft wet cooling tower. J Build Eng. 2023;72(9):106615. doi:10.1016/j.jobe.2023.106615.

9. Zhao Z, Gao J, Zhu X, Qiu Q. Experimental study of the corrugated structure of film packing on thermal and resistance characteristics of cross-flow cooling tower. Int Commun Heat Mass Transf. 2023;141:106610. doi:10.1016/j.icheatmasstransfer.2022.106610.

10. Zhang D, Chen R, Zhang Z, He S, Gao M. Crosswind influence on heat and mass transfer performance for wet cooling tower equipped with an axial fan. Case Stud Therm Eng. 2021;27(1):101259. doi:10.1016/j.csite.2021.101259.

11. Wang Y, Liu Y, Yuan X, Niu P, He S, Gao M, et al. Crosswind effects on thermal performance improvement of mechanical draft cooling towers with deflector plates. Appl Therm Eng. 2024;253:123839. doi:10.1016/j.applthermaleng.2024.123839.

12. Shirazi M, Jahangiri A. 3D numerical study using three novel windbreak walls in natural draft dry cooling towers for performance enhancement under various crosswind conditions. Therm Sci Eng Prog. 2021;25(9):100971. doi:10.1016/j.tsep.2021.100971.

13. Deng W, Sun F, Wang R, He K. Influence mechanism of the louver on the thermal performance of the mechanical draft wet cooling tower. Appl Therm Eng. 2023;230:120640. doi:10.1016/j.applthermaleng.2023.120640.

14. Deng W, Sun F, Chen K, Zhang X. New method

to decrease the air recirculation of mechanical draft wet cooling tower group by increasing height of fan duct. Appl Therm Eng. 2023;219(3):119645. doi:10.1016/j.applthermaleng.2022.119645.

15. Deng W, Sun F, Chen K, Zhang X. New retrofit method to cooling capacity improvement of mechanical draft wet cooling tower group. Int J Heat Mass Transf. 2022;188:122589. doi:10.1016/j.ijheatmasstransfer.2022.122589.

16. Deng W, Sun F, Chen K, Zhang X. The research on plume abatement and water saving of mechanical draft wet cooling tower based on the rectangle module. Int Commun Heat Mass Transf. 2022;136:106184. doi:10.1016/j.icheatmasstransfer.2022.106184.

17. Jiang L, Zhang Z, Sang P, Dong L, He S, Gao M, et al. Influence of noise barrier and louver on ventilation and thermal performance of wet cooling towers under crosswind conditions. Int J Therm Sci. 2022;173(2):107364. doi:10.1016/j.ijthermalsci.2021.107364.

18. Lee JH, Moshfeghi M, Choi YK, Hur N. A numerical simulation on recirculation phenomena of the plume generated by obstacles around a row of cooling towers. Appl Therm Eng. 2014;72(4):10–9. doi:10.1016/j.applthermaleng.2014.04.021.

19. Ma H, Cai L, Si F. Numerical study identifies the interaction between two adjacent dry cooling towers on fluid flow and heat transfer performances of the radiators at different points of each tower. Int J Therm Sci. 2023;191:108351. doi:10.1016/j.ijthermalsci.2023.108351.

20. Cuce E, Sen H, Cuce PM. Numerical performance modelling of solar chimney power plants: influence of chimney height for a pilot plant in Manzanares, Spain. Sustain Energy Technol Assess. 2020;39(1):100711. doi:10.1016/j.seta.2020.100704.

21. Teng K, Peng H, Lv C, Sun H, Wu H. Optimization of emission chimney heights for uranium enrichment facility based on individual dose. J Environ Radioact. 2022;255(2):107014. doi:10.1016/j.jenvrad.2022.107014.

22. Moffat RJ. Describing the uncertainties in experimental results. Exp Therm Fluid Sci. 1988;1(3):3–17. doi:10.1016/0894-1777(88)90043-X.

23. Moffat RJ. Contributions to the theory of single-sample uncertainty analysis. J Fluids Eng. 1982;104(5):250–8. doi:10.1115/1.3241818.

24. Klimanek A. Numerical modelling of natural draft wet-cooling towers. Arch Comput Meth Eng. 2013;20(1):61–109. doi:10.1007/s11831-013-9081-9.

25. Zhao Y, Sun F, Long G, Huang X, Huang W, Lyv D, et al. Comparative study on the cooling characteristics of high-level water collecting natural draft wet cooling tower and the usual cooling tower. Energ Conver Manage. 2016;116(3):150–64. doi:10.1016/j.enconman.2016.02.071.

26. Dang C, Jia L, Yang L. Effects of acoustic barriers and crosswind on the operating performance of evaporative cooling tower groups. J Therm Sci. 2016;25(5):532–41. doi:10.1007/s11630-016-0895-2.