Contract No:

This document was prepared in conjunction with work accomplished under Contract No. 89303321CEM000080 with the U.S. Department of Energy (DOE) Office of Environmental Management (EM).

Disclaimer:

This work was prepared under an agreement with and funded by the U.S. Government. Neither the U.S. Government or its employees, nor any of its contractors, subcontractors or their employees, makes any express or implied:

1) warranty or assumes any legal liability for the accuracy, completeness, or for the use or results of such use of any information, product, or process disclosed; or
2) representation that such use or results of such use would not infringe privately owned rights; or
3) endorsement or recommendation of any specifically identified commercial product, process, or service.

Any views and opinions of authors expressed in this work do not necessarily state or reflect those of the United States Government, or its contractors, or subcontractors.

PAPER NO:

CATEGORY: ENGINEERING STANDARDS AND MAINTANENECE

COOLING TECHNOLOGY INSTITUTE

CONDENSATION MODELING OF PLUME FROM A MECHANICAL DRAFT COOLING TOWER

KAI HAN, ADRIAN MISTREANU, CHRISTOPHER SOBECKI, BRIAN d'ENTREMONT

SAVANNAH RIVER NATIONAL LABORATORY

The studies and conclusions reported in this paper are the results of the author's own work. CTI has not investigated, and CTI expressly disclaims any duty to investigate, any product, service process, procedure, design, or the like that may be described herein. The appearance of any technical data, editorial material, or advertisement in this publication does not constitute endorsement, warranty, or guarantee by CTI of any product, service process, procedure, design, or the like. CTI does not warranty that the information in this publication is free of errors, and CTI does not necessarily agree with any statement or opinion in this publication. The user assumes the entire risk of the use of any information in this publication. Copyright 2022. All rights reserved.

Presented at the 2023 Cooling Technology Institute Annual Conference

Memphis, Tennessee – January 29 – February 3, 2023

Abstract

The mechanical draft wet cooling tower is the most commonly adopted heat dissipation system in the air conditioning and energy industries. Complex ambient conditions influence the aerodynamic behavior of plumes along with topography, such as nearby buildings and it is difficult to predict plume condensation behavior either experimentally or analytically. A computational fluid dynamic model is developed for condensation of water vapor from a mechanical draft wet cooling tower. This study applies realistic operating conditions, comprised of speed, wind direction, ambient air relative humidity, and air volume flow rate coming out of a tower. The effects of plume interactions with nearby buildings are parametrically investigated for the operating conditions using a newly proposed condensation model. It is found that the condensation behavior predicted by the computational fluid dynamic simulation with the developed model is reasonably interpreted with other thermohydraulic parameters.

Keywords: Condensation, plume, mechanical draft wet cooling tower

Introduction

Saturated water vapors exhaust from cooling towers and create a plume of condensation when it mixes with cooler surrounding ambient air. The condensation plume is visible which creates aesthetic concerns while its moisture deposition on the ground or on the buildings generate environmental concerns. As the plume approaches the ground, it can produce fogging and icing which consequentially increases the risk of accidents. The plume trajectory has to be predicted for various atmospheric conditions in order to avoid moisture, icing and fog at unintended locations. The plume dynamic is driven by various factors like fan, buoyancy, ambient conditions (atmospheric, turbulence, etc.), as well as various wind speeds and wind directions. These multivariable considerations make it difficult to predict plume behavior. Because of the limitation of the current analytical and experimental approaches in predicting plume condensation, a new reliable method is needed.

Unlike the relatively abundant and well-established droplet evaporation model[1], many analytical plume condensation studies are limited to estimate plume rise.[2] This type of model is unsuitable to estimate the recirculation behavior near the ground and is difficult to apply a complex environmental condition. A computational fluid dynamics (CFD) modeling approach was recently attempted[3] using the evaporation and condensation mass fluxes estimating equation that was derived from statistical mechanics as well as the thermodynamic states of the liquid and vapor without considering heat transfer.

In this study, a general purpose CFD software is used to monitor liquid droplets coming out of a cooling tower and predict the thermohydraulic behavior of the plume. A subroutine was developed, based on psychrometric equations to calculate liquid droplet condensation. A series of CFD analyses investigated the effect of the ambient conditions (wind speed, wind direction, relative humidity), volume flow rate ejected from a cooling tower, and nearby structural obstruction (height and relative distance from a cooling tower).

Numerical Method

The numerical analyses for two-dimensional domains have been carried out by simultaneously solving species transport equations along with turbulence models, discrete phase models, continuity equation, momentum equations, energy equations, and heat/mass exchange models to track liquid droplets and their condensation. The standard k-ε model is adopted as the turbulence model. The mixing and transport of species are modeled by solving conservation equations describing convection, diffusion, and reaction sources for each component species. The fluid phase is treated as a continuum by solving the Navier-Stokes equations, while the dispersed phase is solved by tracking a large number of liquid droplet particles throughout the calculated flow field. The dispersed phase exchanges momentum, mass, and energy with the fluid phase. Considering the lower volume fraction of the dispersed second phase, particle-particle interactions are neglected. The droplet trajectories are computed individually at specified intervals during the fluid phase calculation. The trajectory of a discrete phase particle is predicted by integrating the force balance on the particle. Heat and mass exchanges are modeled using heat and mass transfer relationships. The moisture amount in the bulk gas is acquired from the solution of the transport equation. When the humidity ratio exceeds the saturated value, droplet condensation is initiated. The exceeding moisture forms spherical liquid droplets, and the corresponding latent energy is used as a source term in the energy equation. All psychrometric parameters like saturation pressure and humidity ratio are calculated using the equations in ASHRAE fundamentals.[4] The second order upwind scheme is used for density, momentum, species, and energy discretization. The coupled scheme is introduced to pressure-velocity coupling. For turbulent kinetic energy and turbulent dissipation rate discretization, the first order upwind scheme is applied. The set of coupled governing equations are solved using a commercial CFD software. The detailed equations and modeling descriptions can be found in the liquid droplet evaporation modeling work.[1]

As seen in Figure 1, a numerical domain having 1200 m (3937 ft) length and 600 m (1969 ft) height is constructed in a two-dimensional rectangular shape. Figure 1 also shows the grids along with boundary conditions. Velocity inlet and pressure outlet boundaries are placed on the vertical boundary lines, and they are switched to investigate the wind direction as necessary. A symmetry boundary is assigned on the top line to reduce backflow and improve convergence. Additionally, a no-slip boundary condition, with a zero-roughness height, is applied to the ground and walls of both buildings. Velocity and temperature profiles are obtained from actual measurement conducted by Meyer et. al.[5] The selected ambient condition is -1ºC (30ºF) dry bulb temperature, 50% relative humidity, and 5 m/s (16.73 ft/s) wind speed. The measured temperature gradient per vertical height is very close to ASHRAE handbook[4] while its velocity does not follow a typical external flow velocity profile as Figure 1 illustrates. The cooling tower having 18.28 m (60 ft) height and 18.28 m (60 ft) width is located 400 m (1300 ft) from the left velocity inlet. A fan boundary is placed inside the cooling tower to depict the air motion near the cooling tower and predetermined air-water droplet mixtures (27.5 ºC [81.5ºF] dry bulb temperature, 98% relative humidity, and a 656.95 m^3/s [23,200 ft^3/s] flow rate) are discharged from the top cooling tower opening having a 9.5 m (31 ft) diameter opening. The base structural obstruction mimicking an air blockage has the identical dimension of the cooling tower and placed 91.44 m (300 ft) away from the cooling tower considering a typical orientation requirement of cooling towers[6] and its actual tower length.[5]

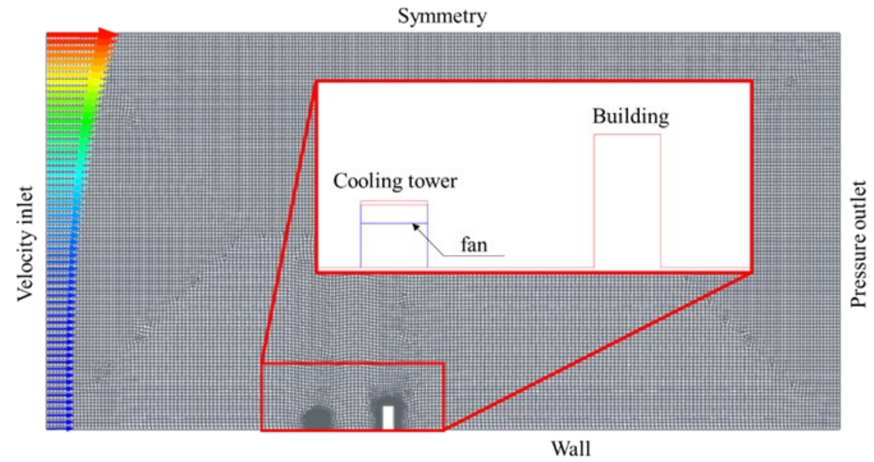

Figure 1. Numerical domain and boundary settings.

Grid dependency is checked using three meshes having 27802, 44278, and 79160 cells in terms of averaged temperature and relative humidity on the ground. Temperature differs from -0.15% to 3.69% and relative humidity changes in the range of -0.05% and -1.31% as grids. This fluctuation can be considered as minor when the result variation in Table 1 is accounted for. Especially for 27802 cells and 79160 cells, they agree well within -0.05% and -0.15%. Hence, the grid having 27802 cells is used for this study.

Results

A parametric study is conducted to investigate the effect of ambient moisture content, wind velocity, velocity profile, discharged air-water vapor mixture volumetric flow rate from the cooling tower, building height, the distance between the cooling tower and the building blockage, as well as wind direction. Case 1 is the base case corresponding to the actual measured data and excludes the building blockage. As can be seen in Figures 2 (a)-(c), recirculation is formed along the flow path on the right-hand side of the cooling tower, and the plume condensation can be discerned by the increase of liquid droplet size and mass as well as a decrease to increase in droplet temperature from the cooling tower exhaust to the opening near the ground as shown in Figures 2 (d)-(f). It can therefore be inferred that condensation progresses along the recirculation flow path. In this study, only condensation is allowed, and liquid droplet evaporation phenomenon is not included. The actual temperature profile measurement at 365 m (1198 ft) altitude is compared to CFD in Figure 3 where a temperature difference is noticeable near the cooling tower location. This temperature inversion in the measurement was caused by the mixing of two air flows having different wind directions at 300 m (984 ft) altitude. Another difference is a solid plume rising without obvious recirculation near the ground in the actual field test trial. Interestingly, Case 10 shows a plume rising pattern which is similar to the actual observation, therefore implying that some wind obstruction might exist near the cooling tower in the test site. These differences also might be originated from the limitation of two-dimensional simulation, the symmetry top boundary setting, and the complicated actual atmospheric wind direction.

Unlike the horizontally parallel inlet air velocity profile of this CFD setting, the actual measurement shows ascending and descending wind direction as altitude increases. Even though the temperature difference can be treated as minor when the measurement uncertainty and accuracy are considered, this disparity illustrates the necessity of three-dimensional analysis and challenging characteristic of atmospheric modeling. It should be noted that this study intends to develop a reasonable liquid droplet condensation model rather than plume behavior investigation. For now, the recirculation pattern is helpful to determine whether the liquid droplet condensation model works properly or not.

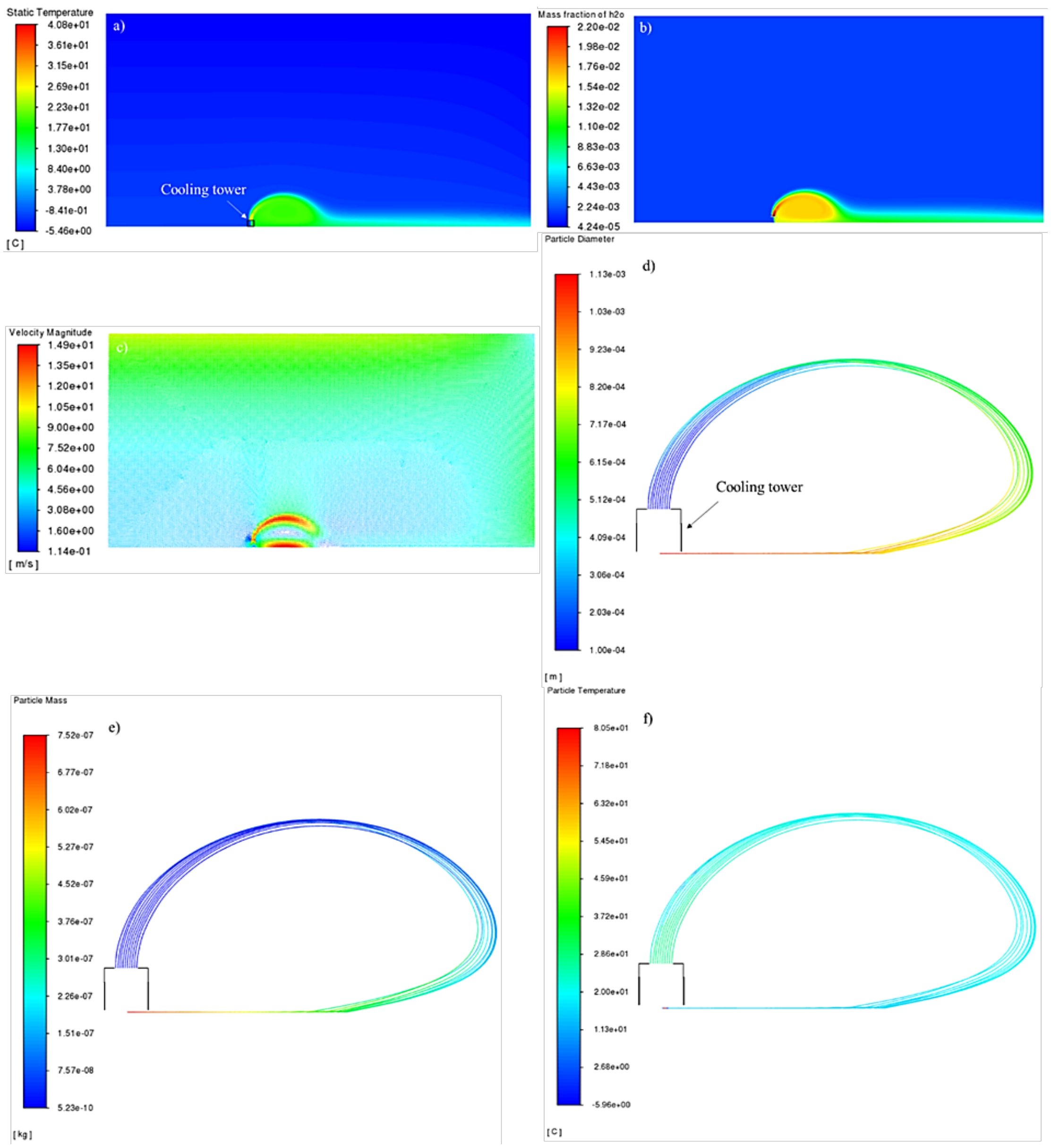

Figure 2. (a) Temperature (b) Water vapor mass fraction (c) velocity vector
(d) liquid droplet size (e) liquid droplet mass (f) liquid droplet temperature of Case 1.

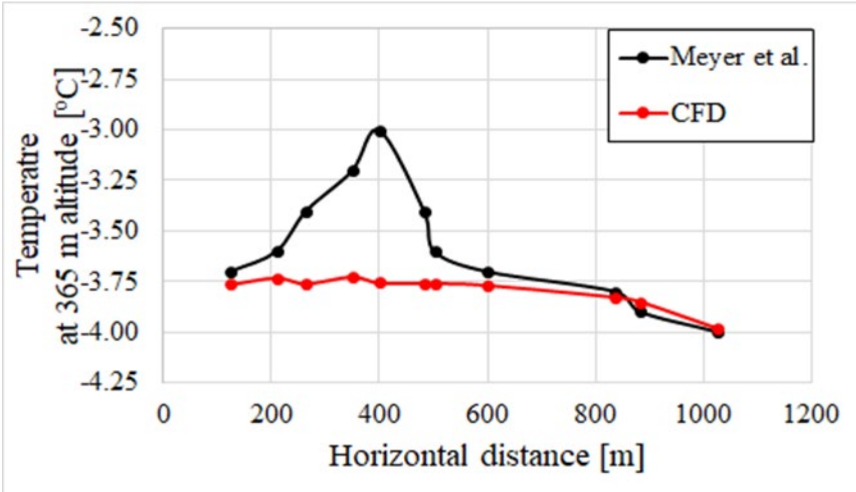

Figure 3. Comparison of temperature between the actual measurement and CFD result at 365 m (1198 ft) altitude.

Compared to Case 1, a total of 9 additional cases are simulated by changing a single parameter per case to investigate its impact. Case 2 is identical to Case 1 except it has a 100% relative humidity for the ambient air. Cases 3 and 4 have 50% reduced and 50% increased wind velocities, respectively, compared to Case 1. A uniform velocity profile is introduced in Case 5 and a 20% increase in plume volume flow rate is applied for Case 6. A building structure with an identical dimension of the cooling tower is placed for Case 7 and the building height is doubled in Case 8. In Case 9, the distance between the building and the cooling tower is halved compared to Case 7. Case 10 switches the pressure outlet and velocity inlet boundaries to observe the wind direction impact.

Table 1 shows the relative condensation rate (kg/s) change on the ground compared to Case 1. It is natural to see more condensation for Case 2 due to the abundant moist content in the ambient air. A reduced wind velocity attenuates the recirculation and decreases the chance of condensation as Case 3 presents. Conversely, increasing the wind velocity in Case 4 shows condensation enhancement by developing a much larger recirculation zone as Figure 4 presents. In Case 5, a uniform velocity profile annihilates the velocity gradient and suppress recirculation, thus decreasing condensation. For Case 6, the increased water vapor amount from the cooling tower also increases condensation. For Case 7, the existence of a structure deteriorates the condensation because the building does not allow the reentrance of plume and weakens the recirculation as seen Figure 5. No significant change is detected as the building height when Cases 7 and 8 are compared because the distance between the tower and the building is secured per the design criteria.[6] Furthermore, Case 9 illustrates that the distance between buildings is more influential on condensation than the building height. Finally, in Case 10, condensation does not occur near the ground since the plume escapes the numerical domain without recirculation as can be seen in Figure 5.

Table 1. Condensation rate change on the ground compared to Case 1

Case 2	Case 3	Case 4	Case 5	Case 6	Case 7	Case 8	Case 9	Case 10
30%	-76%	306%	-72%	54%	-81%	-83%	-92%	-100%

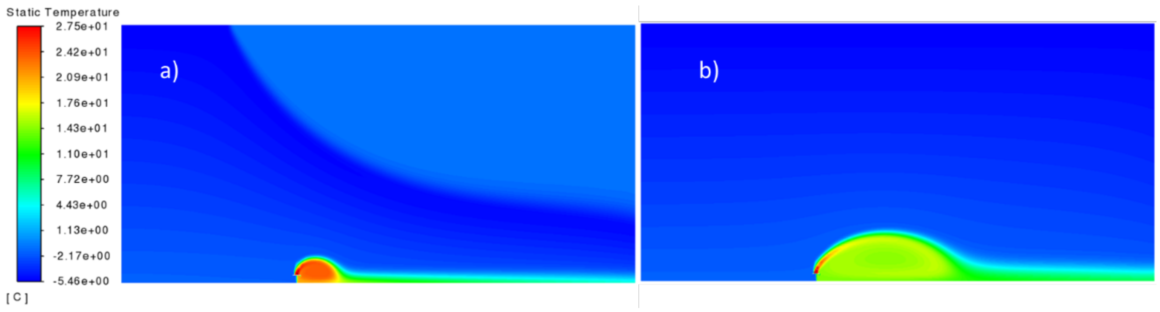

Figure 4. (a) Temperature contour of Case 3 (b) Temperature contour of Case 4.

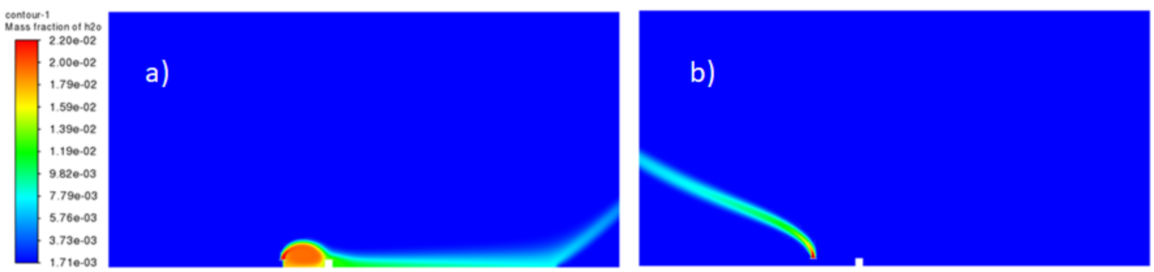

Figure 5. (a) Water vapor mass fraction of Case 7 (b) Water vapor mass fraction of Case 10.

Conclusion

A liquid droplet condensation behavior is investigated for a mechanical draft wet cooling tower with the assistance of a commercial computational fluid dynamics model. A systematic study is carried out to monitor the impact of ambient conditions, nearby building interactions, and cooling tower operating conditions. Reasonable condensation behavior is detected for the considered scenarios since the ambient moisture amount and the recirculation zone size can be correlated to understand the condensation behavior. Condensation increases with the ambient water vapor amount and discharged air-water vapor mixture from the cooling tower whereas wind speed and velocity profile affect the recirculation pattern. Slower wind speed and a flattened wind profile decrease condensation by reducing the velocity gradient and suppressing recirculation. A building structure traps the plume and decreases condensation by preventing the recirculation zone from expanding. Reducing the distance between buildings creates smaller recirculation zone and wind direction interacts with building structures. It is worth mentioning that a comparison to the actual measurement enlightens the modeling area for improvement. For the actual application and prediction, other ignored physical phenomena and a more delicate boundary condition set up should be followed.

Acknowledgement

"This work was produced by Battelle Savannah River Alliance, LLC under Contract No. 89303321CEM000080 with the U.S. Department of Energy. Publisher acknowledges the U.S. Government license to provide public access under the DOE Public Access Plan (http://energy.gov/downloads/doe-public-access-plan)."

Reference

1. D. Han, "Numerical Study on Air Temperature Reduction by Water Spray", International Journal Transport Phenomena, Vol. 14, 2016
2. S.R. Hanna, "Rise and condensation of large cooling tower plumes", Journal of Applied Meteorology and Climatology, Vol. 11, p.793–799, 1972
3. M.I. Ershov, V.G. Tuponogov, N. A. Abaimov, M.A. Gorsky, "CFD modelling of the condensation, evaporation, coagulation, and crystallization processes in the cooling tower emission", Journal of Physics: Conference Series, Vol. 2088, IOP Publishing, 2021
4. ASHRAE, ASHRAE Handbook-Fundamentals, Atlanta, 2017
5. J.H. Meyer, T.W. Eagles, L.C. Kohlenstein, J.A. Kagan, W.D. Stanbro, "Mechanical Draft Cooling Tower Visible Plume Behaviors: Measurements, Models, Prediction", Cooling Tower Environment. U.S. Atomic Energy Commission Symposium, p. 307-352, 1974
6. J.C. Hensley, ed. Cooling tower fundamentals. Stockton (California): Marley Cooling Tower Company, 1985