This document was prepared in conjunction with work accomplished under Contract No. DE-AC09-96SR18500 with the U.S. Department of Energy.

WSRC-STI-2008-00123

Proceedings of HT2008
ASME 2008 Summer Heat Transfer Conference
August 10-14, 2008, Jacksonville, FL, USA

HT2008-56080

CFD MODELING ANALYSIS OF MECHANICAL DRAFT COOLING TOWER

Si Y. Lee
Savannah River National Laboratory
si.lee@srnl.doe.gov
803-725-8462

James S. Bollinger
Savannah River National Laboratory
james02.bollinger@srnl.doe.gov
803-725-1417

Alfred J. Garrett
Savannah River National Laboratory
alfred.garrett.srnl.doe.gov
803-725-4870

Larry D. Koffman
Savannah River National Laboratory
larry.koffman@srnl.doe.gov
803-725-1038

ABSTRACT

Industrial processes use mechanical draft cooling towers (MDCT's) to dissipate waste heat by transferring heat from water to air via evaporative cooling, which causes air humidification. The Savannah River Site (SRS) has a MDCT consisting of four independent compartments called cells. Each cell has its own fan to help maximize heat transfer between ambient air and circulated water. The primary objective of the work is to conduct a parametric study for cooling tower performance under different fan speeds and ambient air conditions.

The Savannah River National Laboratory (SRNL) developed a computational fluid dynamics (CFD) model to achieve the objective. The model uses three-dimensional steady-state momentum, continuity equations, air-vapor species balance equation, and two-equation turbulence as the basic governing equations. It was assumed that vapor phase is always transported by the continuous air phase with no slip velocity. In this case, water droplet component was considered as discrete phase for the interfacial heat and mass transfer via Lagrangian approach. Thus, the air-vapor mixture model with discrete water droplet phase is used for the analysis.

A series of the modeling calculations was performed to investigate the impact of ambient and operating conditions on the thermal performance of the cooling tower when fans were operating and when they were turned off. The model was benchmarked against the literature data and the SRS test results for key parameters such as air temperature and humidity at the tower exit and water temperature for given ambient conditions. Detailed results will be presented here.

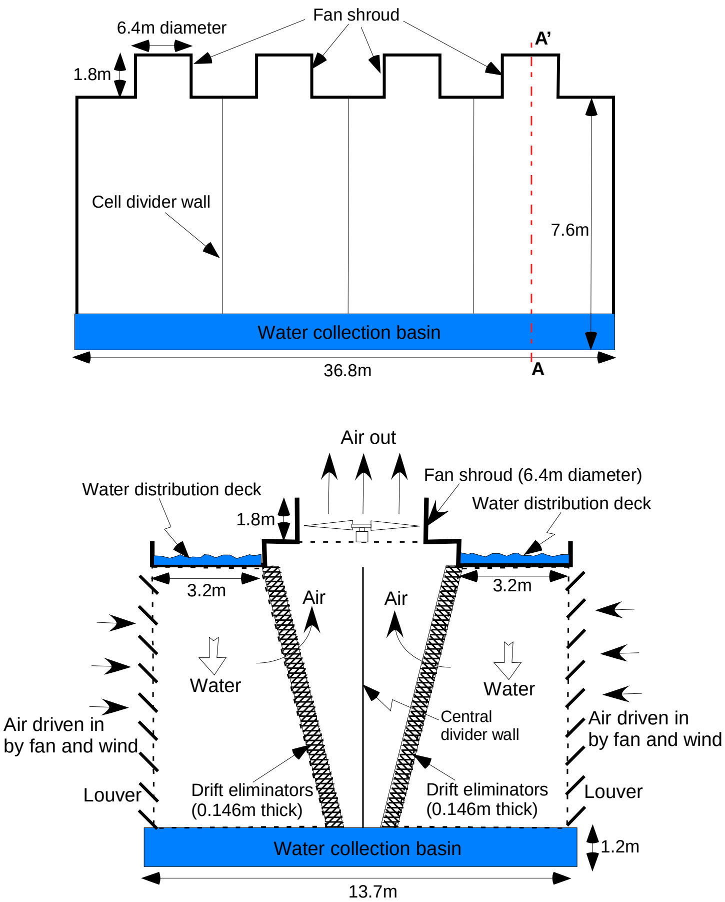
Keywords: Cooling Tower, Computational Fluid Dynamics, Heat Transfer, Mechanical Draft Cooling Tower

INTRODUCTION

Cooling tower is a system for water cooling coupled with air humidification process. It is a simultaneous phenomenon during which heat and mass transfer takes place. Heat is transferred as sensible heat due to the temperature difference between liquid and gas phases, and as the latent heat of the water as it evaporates. Mass of water vapor is transferred due to the difference between the vapor pressure at the air-liquid interface and the partial pressure of water vapor in the bulk of the air. Equations to govern these phenomena are discussed here. The governing equations are solved by taking a computational fluid dynamics (CFD) approach.

The purpose of the work is to develop a three-dimensional CFD model to evaluate the flow patterns inside the cooling cell driven by cooling fan and wind, considering the cooling fans to be on or off. A cooling tower considered here is mechanical draft cooling tower (MDCT) consisting of four compartment cells as shown in Fig. 1. It is 13.7m wide, 36.8m long, and 9.4m high. Each cell has its own cooling fan and shroud without any flow communications between two adjacent cells. There are water distribution decks on both sides of the fan shroud. The deck floor has an array of about 25mm size holes through which water droplet falls into the cell region cooled by the ambient air driven by fan and wind, and it is eventually collected in basin area. As shown in Fig. 1, about 0.15-m thick drift eliminator allows ambient air to be humidified through the evaporative cooling process without entrainment of water droplets into the shroud exit.

The model was benchmarked and verified against off-site and on-site test results. The verified model was applied to the investigation of cooling fan and wind effects on water cooling in cells when fans are off and on. This paper will discuss the modeling and test results.



(Vertical cross-sectional view along the line A-A')

Fig. 1. Geometry and dimensions for each of the four cells in Mechanical Draft Cooling Tower (MDCT)

NOMENCLATURE

°C	Degree Centigrade (or Celsius)
CFD	Computational Fluid Dynamics
DOE	U.S. Department of Energy
hr	Hour
kg	Kilogram
L	Length (m)
m	Meter
mm	millimeter
min	Minute
MDCT	Mechanical Draft Cooling Tower
Pa	Pascal
PN	Plant north
Re	Reynolds number
RH	Relative humidity
RSM	Reynolds Stress Model
s or sec	Second
SRNL	Savannah River National Laboratory
T_{amb}	Ambient air temperature (°C)
T_{wi}	Water temperature at water distribution deck (°C)
u_{ex}	Air velocity at exit of fan shroud (m/sec)
U_o	Wind speed (m/sec)
WSRC	Washington Savannah River Company
γ_{amb}	Vapor mass fraction at ambient condition
θ_o	Wind direction w.r.t. plant north

MODELING APPROACH AND SOLUTION METHOD

The present work took a three-dimensional steady-state CFD approach. The modeling domain is shown in Fig. 1. The air-vapor mixture model was considered, assuming that vapor phase is always transported by the continuous air phase with no slip. In this situation, water droplet component was considered as discrete phase for the interfacial heat and mass transfer to air via Lagrangian approach as shown in Fig. 3. The force balance for each droplet equates the particle inertia with forces acting on a spherical particle of uniform size, d_p. Thus, the air-vapor mixture model coupled with discrete water droplet phase is used for the analysis. The governing equations to be solved for the modeling domain are one air-vapor mixture balance, one vapor species transport, three momentum conservations, two standard turbulence equations, and one air-vapor mixture energy balance. κ-ε standard turbulent model is used for simulation of the turbulent airflow. The solution method is shown in Fig. 2.

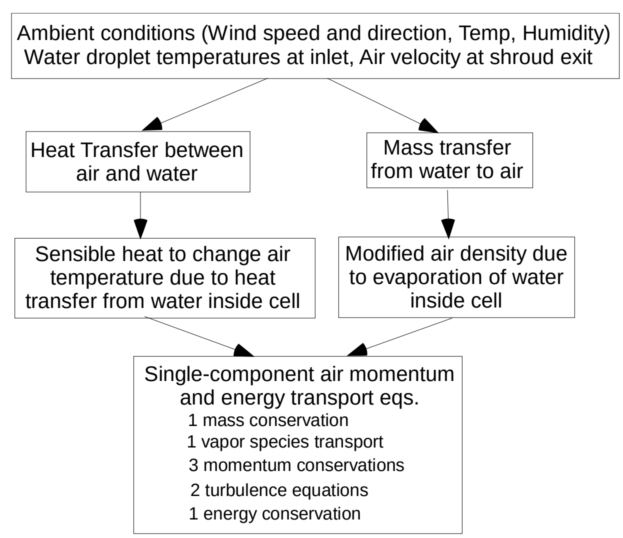


Fig. 2. Solution methods for single-phase mixture CFD modeling approach.

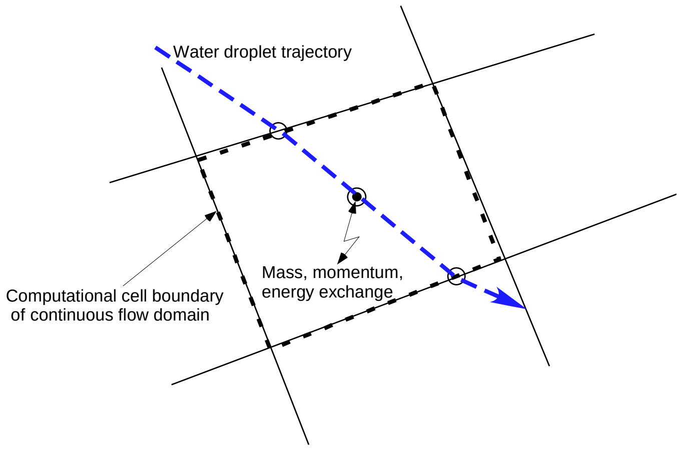


Fig. 3. Mass, momentum, and heat transfer between the continuous gas phase and discrete water droplet

TEST DESCRIPTIONS AND MODEL VALIDATION

Experimental Measurement

The second compartment cell of the cross-flow MDCT facility at Savannah River Site (SRS) was instrumented at the exit of shroud region and near the water collection basin. Sensor locations for the measurements of key operating parameters are shown in Fig. 4. Air temperature and humidity measurements were made by using HOBO data logger [1] at six locations near the top of cooling fan shroud. Water temperatures at the cell exit were also measured by waterproof Tidbit data logger at 0.7m above the free surface of collection basin. Water flowrate and temperature at the inlet of the distribution deck were measured by Doppler ultrasonic meter and Tidbit, respectively. Measurement data for each sensor location were recorded at a time interval of 15 minutes during two-month period in 2006. Test data for ambient air temperature and humidity were continuously obtained from SRNL meteorology station. Wind speed and direction were measured by the wind tower station at SRNL. The data recorded by the sensor logger were downloaded to the computer, and they were averaged over 1-hour period for the benchmarking database to validate the model. The measurement conditions for each test case are summarized in Table 1. Test results were used to benchmark and validate the model.

Model Validation

The analysis consists of two major parts. One part is to develop a model for the operation facility used to simulate cross-flow MDCT to benchmark the calculations with and without cooling fan operations. The second part is to calculate the flow patterns for the turbulent flow induced by fan and wind and to investigate fan and wind effects on water cooling inside the cell when cooling fans are on and off.

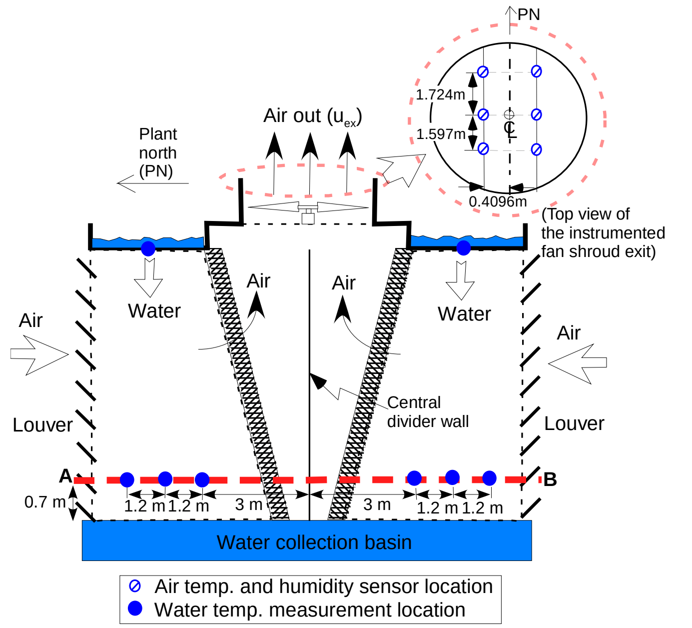


Fig. 4. Cross-section view of the compartment cell instrumented for the performance measurement.

Table 1. Test conditions and results

Test cases	u_{ex} (m/s)	Ambient conditions		T_{wi} (°C)
		T_{amb} (°C), γ_{amb}	U_o(m/s), θ_o	
Fast1	7.76	16.17, 0.0105	2.14, 85.5	27.78
Fast2	7.72	16.78, 0.0080	6.41, 298.2	27.12
Fast3	7.83	22.18, 0.0123	5.69, 263.5	31.78
Slow1	5.20	11.54, 0.0080	3.36, 306.8	26.90
Slow2	5.20	11.43, 0.0080	2.33, 291.80	26.96
Slow3	5.11	12.94, 0.0081	4.64, 301.3	27.06
Slow4	5.06	17.11, 0.0082	4.98, 299.8	27.92
Slow5	5.12	14.56, 0.0080	5.10, 294.3	27.07
Nofan1	-0.55	11.55, 0.0080	3.32, 305.4	26.79
Nofan2	0.24	11.36, 0.0080	2.88, 287.3	26.88
Nofan3	0.33	12.11, 0.0081	3.27, 291.0	26.95
Nofan4	0.37	16.24, 0.0079	5.23, 296.5	27.01
Nofan5	0.14	13.5, 0.0080	5.23, 296.5	27.01

The modeling work considers three basic cases with different operating conditions to examine how sensitive the flow patterns are to different fan and wind speeds. The basic cases are fast fan, slow fan, and no fan as shown in Table 1. Flow patterns coupled with heat and mass transfer were calculated to evaluate the effect of water cooling inside the cell of the cooling tower. A three-dimensional CFD approach was used to solve the governing equations for the flow domain as shown in Fig. 1. A prototypic geometry and domain of the cooling tower was created by a commercial finite volume code, FLUENT, and then it was meshed in non-orthogonal way to solve the governing equations. From the analysis of mesh sensitivity, about 3 million hexahedral meshes were established to perform the calculations.

Drift eliminators inside the cells were modeled as porous media by using Ergun's equation [3]. About 77% porosity was estimated for the 0.15m thick drift region from the literature data [4] as shown in Fig. 5.

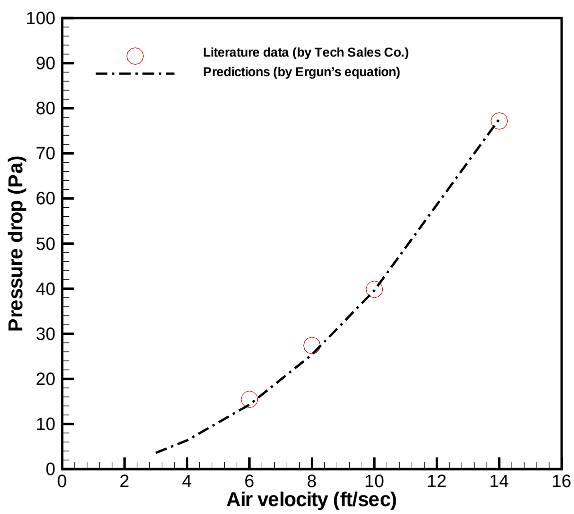
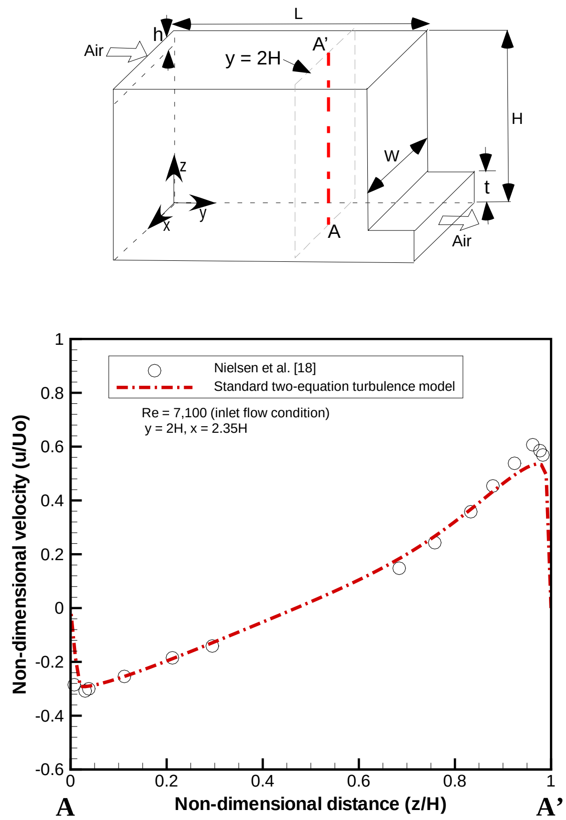


Figure 5 Comparison of the pressure drops across the drift eliminator with the literature data (77% porosity).

The flow conditions for the cooling tower operations are assumed to be fully turbulent since Reynolds numbers for typical operating conditions are in the range of 10^6. A standard two-equation turbulence model, the $k\!-\!\varepsilon$ model [5], was used since benchmarking results against the literature data [6] showed that the $k\!-\!\varepsilon$ model predicts turbulent flow evolution in a large fluid domain with reasonable accuracy. Figure 6 compares the model predictions for the standard two-equation model with the test results available in the literature. Although other turbulent models such as RSM has the potential to give more accurate results for flows in which streamline curvature, swirl, rotation, or rapid changes near the wall boundary might be important, the standard k-ε model is considered a good model for the current calculations over a large fluid domain of mechanical drift cooling tower. The results demonstrate that the k-ε model combined with standard wall functions generally predicts the test results better than other models [7]. Its predictions agree with the data within about 15%.

The literature correlation [8] was used to calculate the heat and mass transfer from water droplets to the continuous gas phase at steady state, assuming them to be spherical and uniform. Based on the literature information [9], the model used the droplet diameter to be 1 mm for the present analysis. As shown in Fig. 7, the present model was benchmarked against the test results available in the literature [10]. The calculation results show that when single droplet has 6mm diameter, the model underpredicts the data by about 18% on the average since the current model assumes spherical droplet. The experimental observations [10] clearly show that when droplet are larger than 4mm, it become non-spherical during free falling period.



(L/H = 3.1, W/H = 4.7, h/H = 0.056, t/H = 0.16, H = 0.0893 m)
Fig. 6. Benchmarking results of non-dimensional horizontal air velocity along the line A-A' on the plane of y=2H distance from the air inlet plane at Re = 7,100 inlet flow (inlet air velocity, U = 10.371 m/sec)

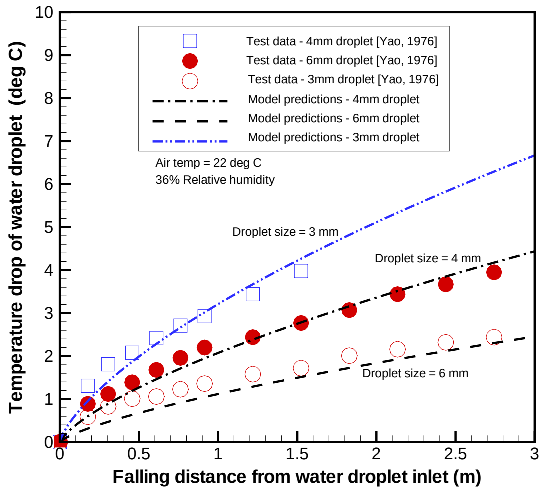


Fig. 7. Comparison of the predicted droplet cooling with the test data for free-falling water droplet in still air done by Yao (1976).

BENCHMARKING RESULTS AND DISCUSSIONS

Modeling predictions for turbulent airflow behavior and heat transfer characteristics were benchmarked against the literature data conducted under the simple geometrical systems. The verified model was extended to the prototypic MDCT system coupled with air humidification process to perform the integral benchmarking tests. The test cases for the SRS cooling tower consist of three basic cases. As shown in Table 1, they are typically three different air velocities at shroud exit, depending on the fan speeds of the cooling tower. Average computational time for each of the test cases was about 4 days using two-cpu parallel run under HP DL585 Linux IBM workstation.

Figure 8 compares the predicted air temperatures at shroud exit with the test results for Fast1 test conditions. The corresponding results for the air humidity at shroud exit are shown in Fig. 9. The results show that the model predictions are in agreement with the test data within about 15%. As shown in the figure, air temperature at the center of the shroud exit is lower than the peripheral region, which is consistent with the test data. This is mainly due to the higher air velocity at its center so that air phase has smaller contact time with the warmer water phase when air velocity becomes higher as shown in Fig. 10. The air temperature and vapor fraction distributions for the vertical plane crossing the second cell are shown in Figs. 11 and 12. The results show that air temperature increases by about 4^{o}C and humidity increases by about 8% RH through the cooling tower, while the predicted water temperatures at exit are lower than the data as shown in Table 2.

When cooling fans are off, water droplets inside each cell will be cooled by wind and natural convection. As shown in Figs. 13 and 14, the modeling predictions for air temperature and humidity distributions at the exit plane of the fan shroud are compared with the test results under the no fan conditions of Nofan5 test case. The model predicts the test data within about 10%. The results clearly show that when fans are off, air temperatures at shroud exit are nearly uniform.

The variations of exit air and water temperatures with air mass flowrate for the similar ambient conditions are shown in Figs. 15 and 16, respectively. The predictions are in reasonable agreement with the test results. As shown in the figures, it is noted that the exit air temperature tends to decrease with increasing air mass flowrate, and the exit water temperature decreases as air mass flow rate increases. From the literature correlation [8], it is clearly shown that when air flowrate increases, the exit water temperature decreases because of the increased heat transfer rate.

Figures 17 and 18 show the benchmarking results against all test results for air exit temperatures and vapor contents at shroud exit. It is noted that the predicted air temperatures are about 18% higher than the test results. As primary reason for this behavior, the model assumed water flow distribution at inlet to be uniform for the efficient computational time although the test results show that water distribution over the distribution decks is not uniform. It is concluded that the CFD model for the MDCT system captures flow patterns and heat transfer characteristics, and it predicts the test results in a reasonably accurate way.

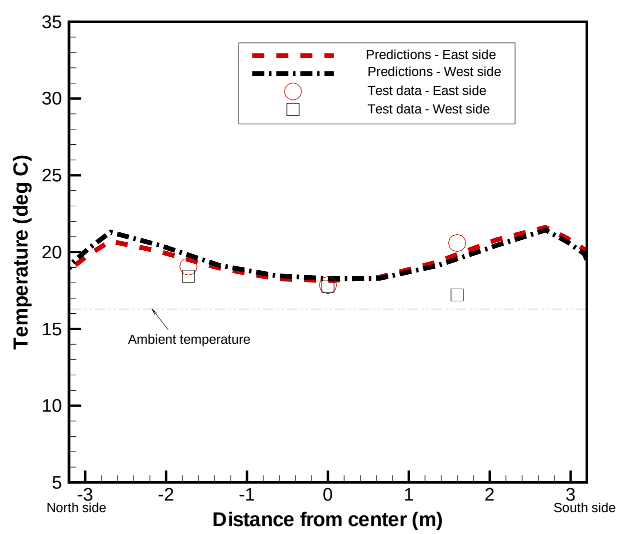


Fig. 8. Comparison of air temperature at shroud exit for Fast1 test conditions

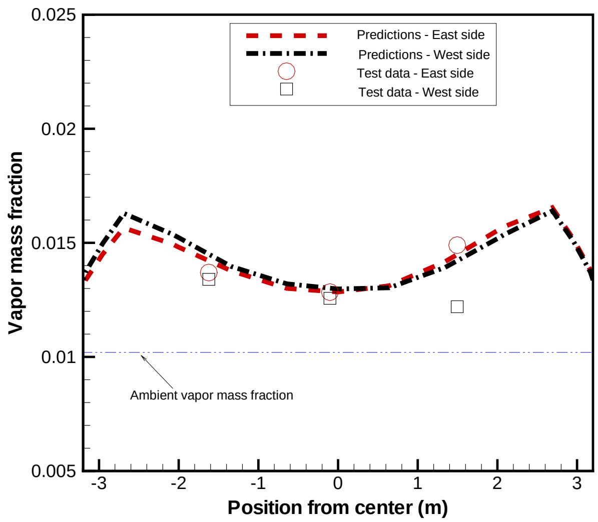


Fig. 9. Comparison of air humidity at shroud exit for Fast1 test conditions

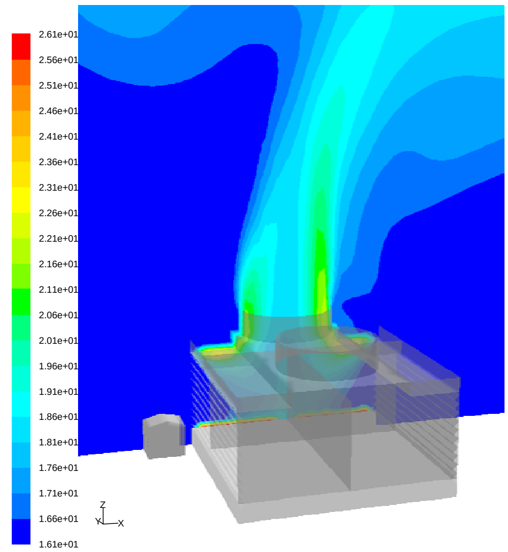


Fig. 11. Air temperature distributions for the vertical mid-plane crossing the instrumented cell of the cooling tower for Fast1 case, showing that red-color zone indicates about 26°C.

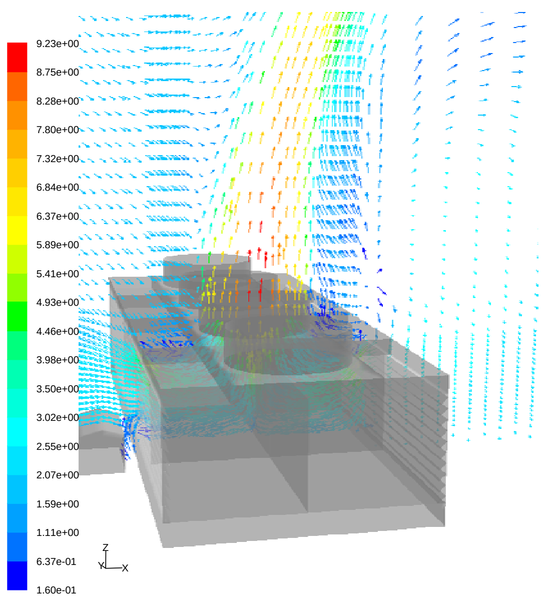


Fig. 10. Air flow patterns for the vertical mid-plane crossing the instrumented cell of the cooling tower for Fast1 case, showing that red-color vector indicates about 9 m/sec air.

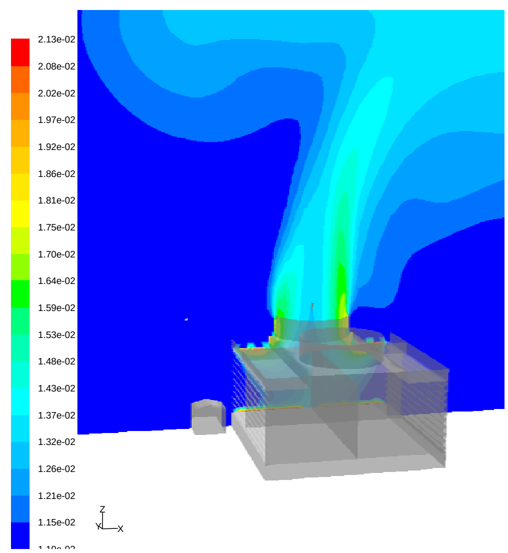


Fig. 12. Air mass fractions for the vertical mid-plane crossing the instrumented cell of the cooling tower for Fast1 case, showing that red-color zone indicates about 2.1% mass fraction.

Table 2. Comparison of water temperature predictions with test data at 28-in above the water basin surface under Fast1 operating conditions

Locations	Temperature (^oC)	
	Predictions	Test data
North outer	18.40	19.32
North middle	19.72	22.74
North inner	22.00	24.72
South outer	19.01	18.65
South middle	20.54	21.14
South inner	21.82	22.76

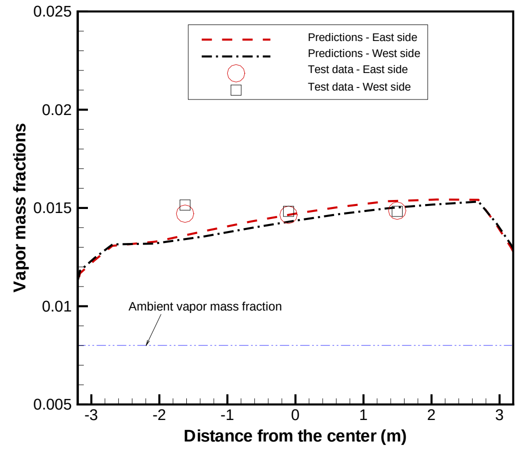


Fig. 14. Humidity distributions at the exit of the cooling fan shroud under the no fan conditions of Nofan5

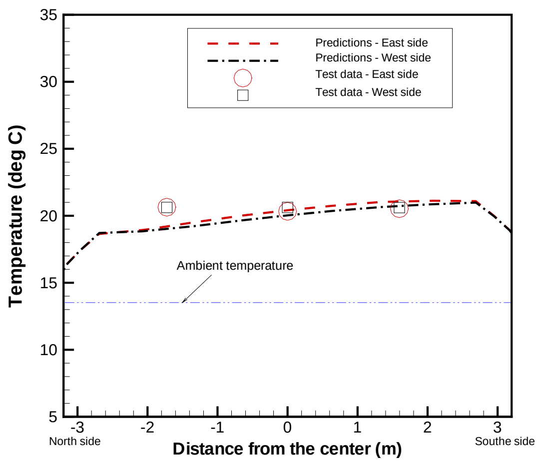


Fig. 13. Temperature distributions at the exit of the cooling fan shroud under the no fan conditions of Nofan5

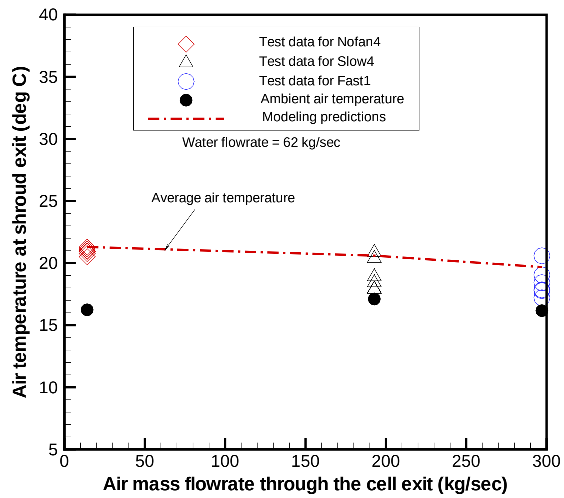


Fig. 15. Variation of air exit temperature for different air mass flow rates driven by fan and wind

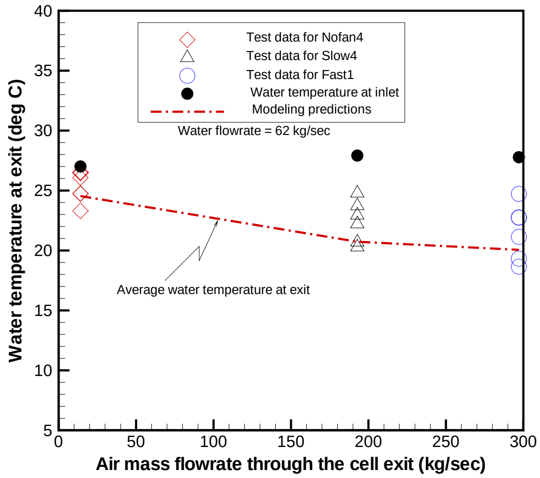


Fig. 16. Variation of water exit temperature for different air mass flow rates driven by fan and wind

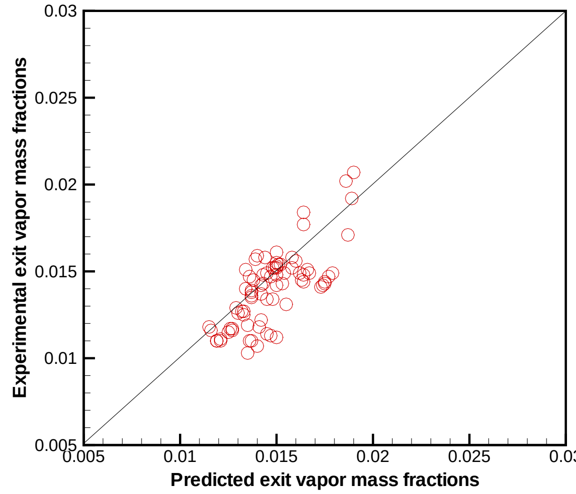


Fig. 18. Comparison of the model predictions with test results for vapor mass fractions at shroud exit.

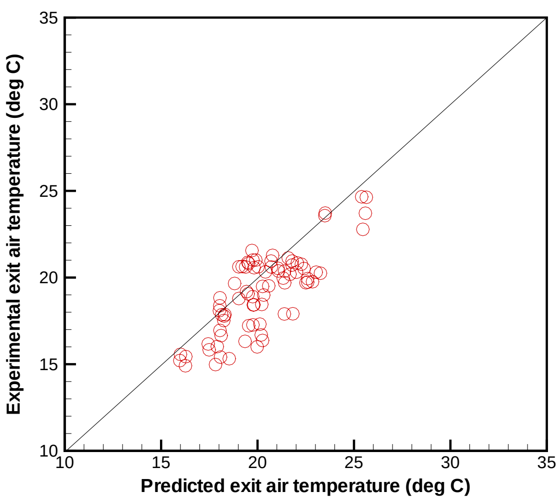


Fig. 17. Comparison of the model predictions with the test results for air exit temperature.

CONCLUSION

A three-dimensional steady-state CFD model was developed for the SRS MDCT system to evaluate the flow patterns and heat transfer characteristics inside the cooling cell driven by cooling fan and wind. It used standard two-equation turbulence model to capture turbulent flow behavior of air. The model considers the air-vapor mixture coupled with water droplet component, assuming that vapor phase is always transported by the continuous air phase with no slip velocity. In this work, water droplet component was considered as discrete phase via Lagrangian approach for the evaporative heat transfer. Experiments were conducted to obtain the benchmarking database for verifying the CFD model.

A series of the modeling calculations was performed to investigate the impact of the ambient and operating conditions on flow patterns and heat transfer characteristics inside the cell of the cooling tower. The modeling predictions are in reasonably good agreement with the test results. It is also demonstrated that CFD method is applicable to the detailed modeling analysis for the cooling tower system.

ACKNOWLEDGMENT

This work was funded by U.S. Department of Energy and performed at the Savannah River National Laboratory, which is operated by the Washington Savannah River Company.

REFERENCES

[1] HOBO Water Temp Pro Logger, www.microdaq.com.
[2] FLUENT, 2006, Fluent, Inc., Lebanon, New Hampshire.

[3] S. Ergun, 1952, “Fluid through Packed Columns”, *Chemical Engineering Progress*, Vol. 18, No. 2, pp. 87.

[4] Tech Sales Co. website, “Cooling Tower Drift Eliminators”, www.tech-sales/Kimre_cooling_tower_drift_eliminators.htm, June 14, 2006.

[5] W. P. Jones and P. E. Launder, 1972, “The Prediction of Laminarization with a Two-Equation Model of Turbulence”, *Int. Journal of Heat and Mass Transfer*, Vol. 15, pp. 301-314, 1972.

[6] P. V. Nielsen, A. Restivo, and J. H. Whitelaw, 1978, “The Velocity Characteristics of Ventilated Rooms”, ASME *J. of Fluids Engineering*, Vol. 100, pp. 291-298.

[7] S. Y. Lee, Richard A. Dimenna, Robert A. Leishear, David B. Stefanko, 2008, “Analysis of Turbulent Mixing Jets in a Large Scale Tank”, ASME *Journal of Fluids Engineering*, Volume 130, Number 1, pp. 011104.

[8] W. E. Ranz and W. R. Marshall, Jr., 1952, Part II, Evaporation from Drops”, Chemical Eng. Progress, vol. 48, pp. 173-180.

[9] S. P. Fisenko, A. A. Brin, and A. I. Petruchik, 2004, “Evaporative Cooling of Water in a Mechanical Draft Cooling Tower”, *Int. J. of Heat and Mass Transfer*, Vol. 47, pp. 165-177.

[10] S. C. Yao and V. E. Schrock, 1976, “Heat and Mass transfer from Freely Falling Drops”, Trans. ASME *J. of Heat Transfer*, pp. 120-126.