ENGINEERING RESEARCH JOURNAL (ERJ)
Vol. 51, No. 4 October. 2022, pp.149-160
Journal Homepage: erj.bu.edu.eg

Numerical Investigation of Cooling Tower Performance under Constant Heat Load

Ahmed ElDegwy[*, 1], Ashraf Sayed[1], Ahmed Aboulmagd [1]

[1]Mechanical Power Engineering Department, Faculty of Engineering, Cairo University, Giza, Egypt
[*]Crossponding author

Abstract: Cooling towers play a vital role in many large-scale process applications, and any decline in their performance has a considerable effect on the underlying process. It is known that the efficiency of a power plant is greatly affected by the temperature difference of the condenser. The objective of this paper was to produce cooling tower design recommendations and considerations that would prevent negative impacts and ensure stable and efficient operation. Computational fluid dynamics (CFD) was used to examine the components that contribute to cooling tower performance, using steady-state simulations and average weather data from the Egyptian Meteorological Authority. Air flow patterns in and around cooling towers were predicted using computational fluid dynamics. The current study includes a numerical analysis of the performance of the cooling tower at different wind speeds and heights of the cooling tower above the ground. This study found that some wind speeds have a negative effect and others have a positive effect, and the height of the cooling tower above the ground has a positive effect on the performance of the tower.

KEYWORDS: CFD; Cooling tower; Thermal performance; Heat transfer; Crosswind effect

Symbol	Description	Symbol	Description
English letters			
c_{pa}	Specific heat [J/kg·K]	$S_{p\phi}$	Additional source term of air-water interaction [W/m^3]
C_μ	Coefficient of eddy-viscosity [-]	S_ϕ	Energy or momentum source term [W/m^3]
D	Cooling tower base diameter [m]	t_m	Average temperature of circulating water [K]
H	Cooling tower height [m]	T_a	Local temperature [K]
h_o	Control volume height [m]	v_i	Local fluid velocity [m/s]
G	Mass flow rate [Kg/s]	u	Wind speed height h [m/s]
k	Turbulent kinetic energy [J/kg]	u_o	Wind speed height h_o [m/s]
Q_v	Volumetric heat transfer capacity [W/m^3·K]	x_i	Spatial flow direction [m]
Greek symbols			
ρ	Density [kg/m^3]	θ_m	Average temperature of moist air [K]
μ_t	Eddy viscosity [Pa·s]	ϕ	General variable
Γ_ϕ	Diffusion coefficient for variable ϕ [J]	υ	Kinematic viscosity [m^2/s]
θ_{in}	Inlet temperature of moist air [K]	$\dot{\forall}$	Volume [m^3]

θ_{out}	Outlet temperature of moist air [K]		
Abbreviations		**Subscripts and superscripts**	
CFCT	Cylindrical frustum cooling towers	m	mean variable
CFD	Computational fluid dynamics	in	inlet variable
NDDCT	Crosswind affects natural draught dry cooling towers	out	outlet variable
SLWCT	super large-scale natural draught wet cooling towers	pa	air constant pressure

1. INTRODUCTION

These studies focuses on the airflow nearby cooling towers that are positioned over ground. This application's parametric works are based on studies of the impact of different indicators on the cooling tower outflow recirculation rate [1-4]. Recirculation describes the movement of moist air expelled from the tower intake again. The cooling tower efficiency is drastically diminished when recirculation occurs, which has a direct influence on the served process stability, efficiency, and consumption of energy [5, 6]. In several instances for a range of tower designs, the detrimental impacts of crosswind on the thermal performance of natural draught cooling towers have been displayed. The current research status in this topic is synthesized in this paper's thorough overview. Each solution's primary benefits and drawbacks are outlined and addressed [7]. Some previous studies show that the effect of wind speed has a negative impact on the cooling towers performance, while others say it is positive. Under poor working conditions, gas from a dry-cooling tower could not be released smoothly. As a result, the cooling tower's inner shell may have been severely corroded as a result. They created a simulation analysis for a natural-draught dry-cooling tower (NDDCT) with gas injection to evaluate contaminant transport and scatter [8]. The researchers discovered that injecting flue gas from steam generators into a natural draught dry cooling tower improves performance and suction capabilities. Injecting hot gas (at 130 °C) into the towers also increased air intake and improved the towers' thermal performance, according to the research [9]. The numerical investigation of the merging and mixing of buoyant plumes coming from modest cooling towers into the atmosphere is underway. On plume mixing, the impacts of various cooling tower layouts and outlet geometries are explored. In each of the aforementioned combinations, the many ways in which the counter revolving vortex pair, as the primary mechanism, impacts the flow pattern are being examined [10]. It's based on the ANSYS Fluent computer program, which gives accurate equations for the physical processes that happen in natural draught wet cooling towers. Heat and mass fluxes among water and air, and also drag equations for the two-phase system, are calculated in specific zones like the cooling fill, spray, and rain zones [11, 12].

This study presents a numerical model of a natural draught wet-cooling tower with flue gas injection. After that, the model is used to examine the impact of gas injection on cooling tower performance, which is compared to experimental data. The rising plume and injected gas are affected by wind speed. According to the findings, the injected gas has no influence on the temperature of cooled water. Rising wind speeds because the formation of recirculation zones near the tower outlet, which raises the risk of corrosion [13]. Crosswind affects natural draught dry cooling towers (NDDCTs) of all sizes, although short towers indicated for geothermal or solar thermal power plants might prove fatal. Using 3D models with varying wind attack angles, the effect of windbreaks on NDDCT performance was examined [14]. A simple theoretical model was used to anticipate the effect of cross wind on the performance of natural draught dry cooling towers. Future research on air-cooled condensers, particularly for geothermal and solar thermal power plants, will benefit from the findings of this study [15]. Discharge recirculation in cooling towers can impair the energy efficiency of cooling facilities and increase the possibility of noticeable plumes around the cooling towers. The addition of reactor cooling towers might result in a 1.5%

increase in overall chilling plant energy consumption. Recirculation increases the frequency of plume occurrences, especially in the spring [16]. The air-side and water-side heat transfer coefficients, which are easily determined from just two rating points and data from product catalogues, have been analyzed using a suggested solution model. As a consequence, using the model enables one to estimate how much water and energy will be used under various operating circumstances, such as shifting air flow rates or wet bulb temperatures [17].

In order to better understand the effects of ambient air temperature and air input deviation angle on the effectiveness of cooling columns, the aerodynamic field around cooling deltas was examined in both windless and crosswind scenarios. The cooling capabilities of each sector under the influence of crosswinds were examined with constant heat load and constant entrance water temperature [18]. Based on the interplay of three zones, a cooling tower's thermal performance is examined. Lower exit water temperature and greater cooling tower efficiency result from smaller droplet diameter and better air to water mass flow ratio. Little effect of droplet velocity on these. The findings of this research offer the theoretical underpinnings for precise performance prediction and point the way toward cooling tower improvement [19]. Based on our prior research, the objective of this study is to investigate the effects of crosswind on super large-scale natural draught wet cooling towers (SLWCTs) using a three-dimensional numerical simulation of the SLWCT fitted with an axial fan. According to the results, the optimization technique successfully lessens the opposing effects of the crosswind, and the air speed close to the fan is considerably improved. Water heat transfer coefficient and the temperature decrease by 4.79 % and 0.226 ^{o}C, respectively, in contrast to situation of no fan, and the maximum evaporation loss is 2.86 % [20]. The relationship between cooling tower performance and crosswind speed is extremely sensitive to wind attack angles. The findings imply that the location of triangular windbreaks always contains one symmetry axis that is parallel to the main crosswind's direction. The results might be used to establish the windbreak fitting angles in relation to the predominant ambient wind direction in a district [21]. It was investigated physically and through numerical simulations how well cylindrical frustum cooling towers (CFCTs) cooled. According to the findings, they are mostly comparable to those of CFCTs and HCTs operating in no-wind situations. Using computational fluid dynamics (CFD), it was determined how the geometric parameter p affected the cooling performance. Recommendable parameters were then provided [22]. Recirculation causes the mean incoming wet bulb temperature at the cooling tower's intake to rise, which lowers cooling tower efficiency. By examining the thermal characteristics and moisture content of the air flow, ElDegwy and Khalil [23] work emphases on forecasting the airflow patterns near towers. Besides, the study uses computational fluid dynamics (CFD) to examine how wind speed, direction, and cooling tower configuration affect cooling tower recirculation. According to the previous literature and the knowledge of the author, this investigation into the result of wind speed as well as the height of the cooling tower above the ground in order to understand how these factors affect the thermal performance of the cooling tower has not been studied in this way before.

2. NUMERICAL INVESTIGATION

2.1. Simplifications and Assumptions

Distribution pipes and internal gaps of fills, which are geometrical construction aspects of the cooling tower, are ignored in the modelling process to make the model more straight forward and simpler to understand. The governing equations use source terms to account for the air motion loss caused by these buildings. Because of the little temperature and pressure variations inside the tower, air is believed to be a suitable gas [24]. Around the tower, there is a tumultuous air current. The steady-state Reynolds averaged Navier-Stokes equations may thus be used to simulate it, and the N-S equations can be solved using the standard k-ε turbulence models [24–26]. Specific calculation methods are presented in [27] for rainfall area, fill area, and water spray area. The primary equations are just briefly discussed in this paper.

2.2. Governing Equations

To investigate the humid air flow that is present around and through a tower, use the steady state equations. The equations were solved using the CFD modeling approach utilized in this study [25]:

$$\nabla(\rho\vec{v}\phi) = \nabla(\Gamma_{\phi}\nabla\phi) + S_{\phi} + S_{p\phi} \tag{1}$$

Also, the estimated variables ϕ stand for 1 for continuity equation, air temperature T_a for energy equation, velocity component v_i for momentum equation, and C_i species air in mass for species equation. These governing equations and the standard k-ε turbulence model might be solved together [24–26]. Equation (2), demonstrate that the Prandtl-Kolmogorov equation states that the turbulent viscosity in this model is connected with the dissipation rate, ε, and the kinetic energy, k, of the turbulence, according to [27].

$$\mu_t = \frac{C_{\mu}\rho k^2}{\varepsilon}\ , where\ C_{\mu} = 0.09 \tag{2}$$

The calculation method for volumetric heat capacity Q_v, a crucial quantity that reflects the efficiency of cooling towers' thermal systems, is illustrated in Eq. (3) [28].

$$Q_v = \frac{C_{pa}\ .G.(\theta_{in} - \theta_{out})}{\forall.(t_m - \theta_m)} \tag{3}$$

$$t_m = \frac{T_{in}-T_{out}}{2},\quad \theta_m = \frac{\theta_{in}-\theta_{out}}{2} \tag{4}$$

2.3. The Model Description

Egypt is the place of the facility that is currently being designed. Figure 1 show that the cooling tower boundary conditions and dimensions. Figure 1 depicts the general arrangement, which has a general domain of 1035 m x 500 m x 500 m. Grid independence tests were carried out on grids with 4, 6, 8, 10, 15, and 20 million mesh nodes. A 15 million cell tetrahedral grid covered the domain. Average output pressure and temperature variations of less than 3.5% and 4%, respectively, were generated with grid-independent findings. In the ANSYS program, convert the tetrahedral to polyhedral to save time on calculation with almost the same calculation accuracy as shown in Figure 2.

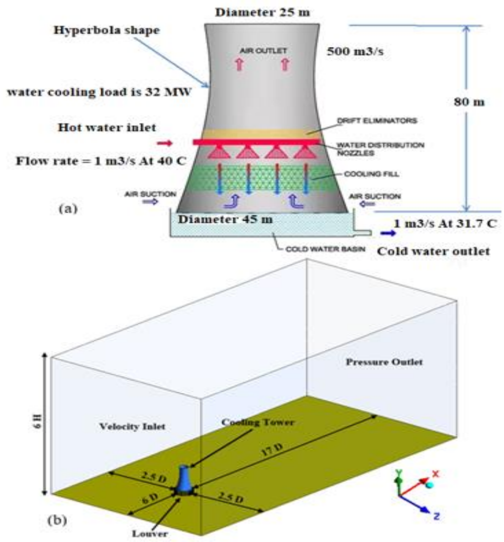
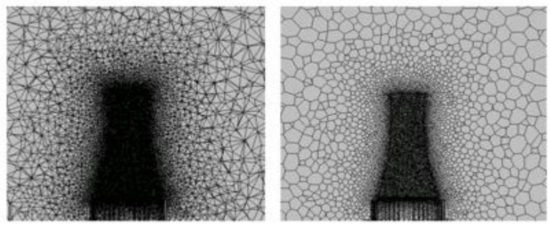


FIG 1. Cooling tower configuration **(a)** boundary conditions and dimensions of cooling tower construction, **(b)** dimensions of CFD domain



(a) (b)

FIG 2. Mesh size configuration: **(a)** tetrahedral, **(b)** polyhedral

2.4. Validation

The CFD model used to construct the current study was compared to a previously published research paper that provided an experimental and numerical comparison of wind speed within the wind tunnel [8]. This comparison was performed to examine the validity and applicability of the current CFD model. As shown in Figure 3, in the simulation, the standard k-ε turbulence model is utilized, and the findings were compared to the outcomes of the experiment [8]. This model was chosen from among several other turbulence models because it shows a high level of agreement with the experimental findings.

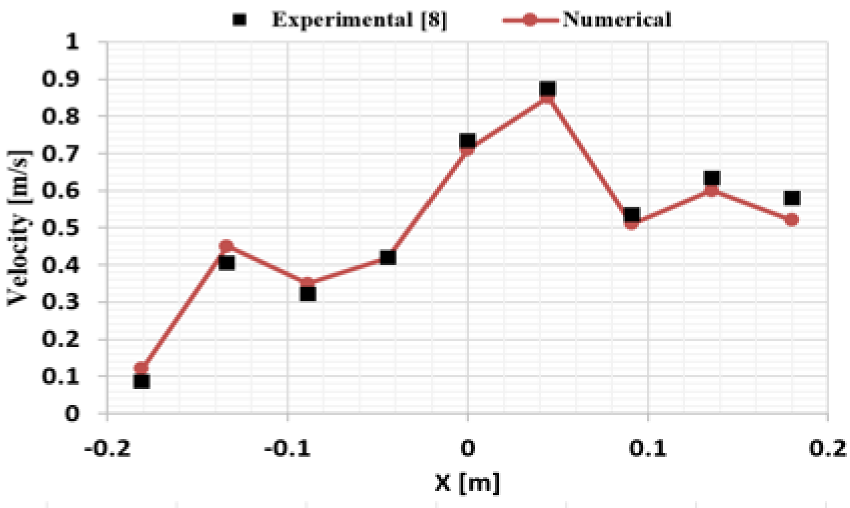


FIG 3. Wind tunnel experimental [8] vs. numerical results at the throat height of the cooling tower

2.5. Boundary Conditions

The water-cooling load is equal to 32 MW, water volume flow rate is 1 m^3/s at inlet hot water temperature 40 °C and outlet water temperature at 31.7 °C. The axial fan air flowrate is 500 m^3/s, and the surrounding air temperature is 40 °C and relative humidity 50%. The inlet hot water temperature is 40 °C. To get accurate simulation results, the velocity profile of wind speed is defined using average meteorological data from the Egyptian Meteorological Authority. Ref. [29, 30] can be used to determine the wind speed velocity profile.

$$u = u_o(\frac{h}{h_o})^m \quad (5)$$

where u_o is the observed velocity at height y_o and u is the velocity at height y, respectively. The index m is affected by the time of day and by the season. The value of m is 0.25 and has been incorporated into each computation [20, 31].

2.6. Convergence Criteria

The numerical solutions of the governing equations were carried out by means of the pressure-based steady-state solver with SIMPLE algorithm and second-order discretization [26]. In most of the cases, the case stopped after about 1500 iterations, and the convergence criterion of residuals being smaller than 10^{-5} for continuity, species, and momentum equations, and less than 10^{-7} for energy equation [30].

3. RESULTS AND DISCUSSIONS

The first scenario is looking at the impact of wind speed on the tower performance. Wind speeds of 0, 5, 10, 15, 20, 25 and 30 m/s were used in the simulation. In the x coordinate direction, the wind is blowing across the tower. The temperature of the tower outlet air is near 60 °C in most cases, and the relative humidity is 100%. **The second scenario** involves examining the impact of lowering the cooling tower's edge height above earth 5, 10, and 15 m. The goal of this article was to look at the flow pattern and how it affected the cooling tower's efficiency. The following figures are aimed to demonstrate the impact of tower edge elevations over ground of h= 5, 10, and 15 m on air flow inside the tower and the associated velocity and temperature distributions at the air intake of the tower.

Figure 4 displays the velocity contour within the tower at various heights and wind speeds on two different planes. The wind is blowing in a left to right direction. Additionally, the distribution of velocities through the tower at the various heights is compared, but the recirculation zone differs at the cooling tower's entry and goes on to the fill zone. At 10 m/s of wind speed, its impact is obvious. These recirculation zones have a negative impact on the cooling tower performance because they trap hot, moist air inside it close to the fill zone. As shown in Figure 4(a), when the wind's speed rises, so does the inlet velocity to the cooling tower, improving the velocity distribution through the tower. The best improvement occurs at a height equal to 15 m above the ground. In addition, the fan is unable to endure the negative impacts of the wind because high wind speeds cause the wind to enter the cooling tower deeply.

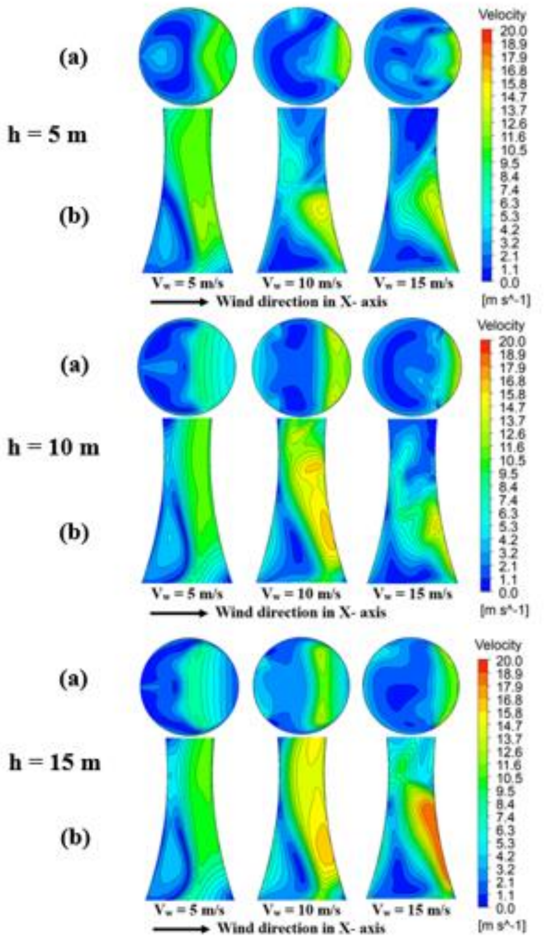


FIG 4. The aerodynamic field inside the tower at different heights 5, 10, and 15 m and different wind speeds of 5, 10, and 15 m/s: **(a)** at Horizontal plane Y = 23 m, **(b)** at middle of the tower (Z = 250 m)

Figure 5 shows the velocity contours within the tower at various heights and wind speeds on two different plans. The wind is blowing in a left to right direction. In addition, the distribution of velocities within the cooling tower at different heights is compared, but the effect of the recycling area is not as significant as that of previous wind speeds. According to Figure 5(a), as wind speed increases, so does the intake velocity to cooling tower, improving the velocity distribution inside the tower. The best improvement occurs at a height equal to 15 m above the ground. The wind acts to create an asymmetric distribution of velocities inside the cooling tower, and this has a detrimental influence on the performance of the tower. To get rid of that, it is possible to put a wind break at the intake of the tower, which works to reduce the detrimental impact of wind as well as improve the distribution of air at the entrance to the tower. In addition, the fan is unable to endure the negative impacts of the wind because high wind speeds cause the wind to enter the cooling tower deeply.

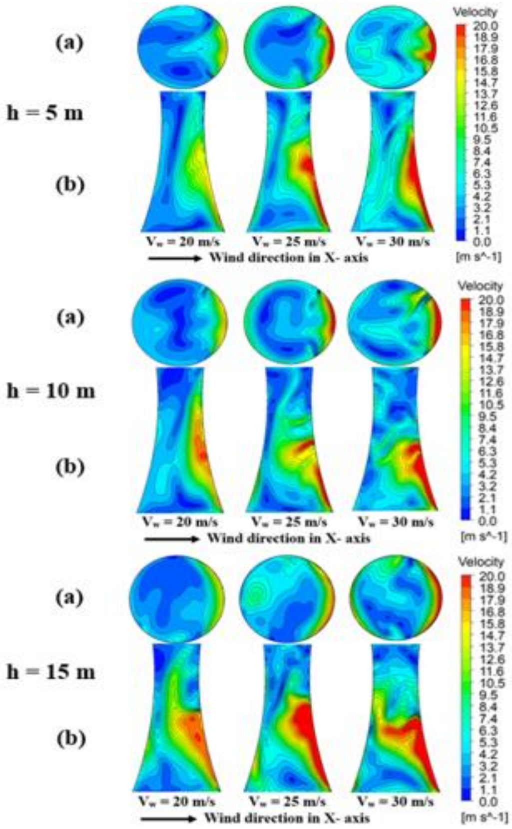


FIG 5. The aerodynamic field inside the tower at different heights 5, 10, and 15 m and different wind speeds of 20, 25, and 30 m/s: **(a)** at Horizontal plane Y = 23 m, **(b)** at middle of the tower

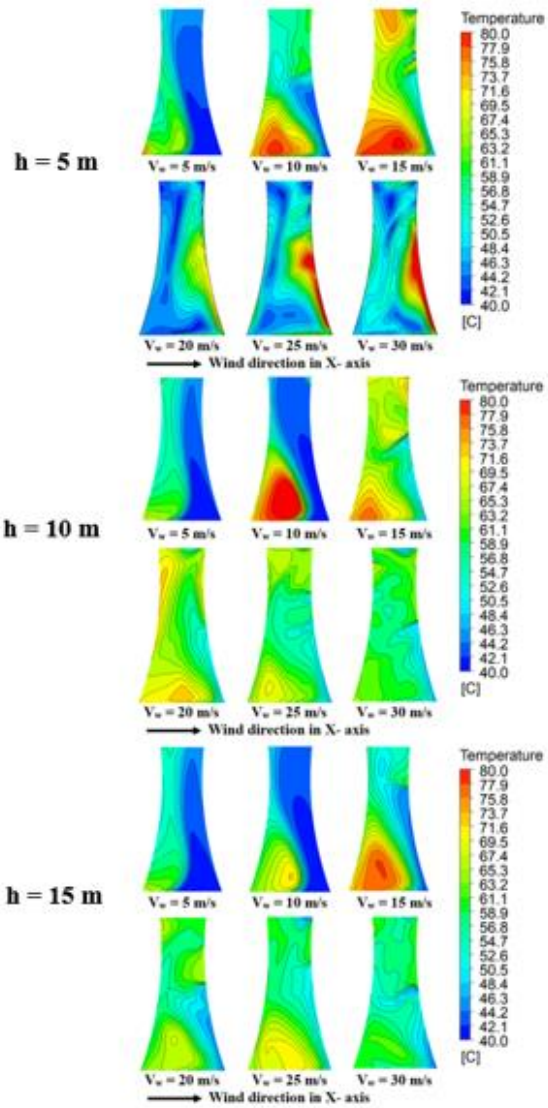


FIG 6. Temperature contour at different heights and different wind speeds at middle of the tower

Figure 6 shows temperature contours comparisons at different heights at Z = 250 m and different wind speeds. Due to internal cooling tower recirculation close to the cooling tower intake, the mean incoming temperature at the cooling tower's fill rises, which reduces cooling tower performance and eventually leads to the breakdown of the connected process equipment. Additionally, when the speed is 20 m/s or higher, we may see a favorable impact on the temperature dispersion throughout the cooling tower. Additionally, Figure 6 displays anticipated temperature contours; increasing wind speed from 20 to 30 m/s enhanced recirculation zones but that the temperature contours within the cooling tower depended on wind speed and the height of the cooling tower from the ground. The results showed a stronger dispersion throughout the tower and asymmetric wind-dependent thermal behavior at a wind speed of 10 m/s. In a vertical segment across the cooling tower, the matching temperature contours are depicted in Figure 6. Besides, the quasi-equivalent velocity copes with a wind speed of 20 and 25 m/s.

Figure 7 shows the colored streamlines with temperature lines at different wind speeds and a height of h = 10 m. It is observed how the wind speed can obstruct the exit of hot moist air from the top of the cooling tower, and this negatively affects the performance of the cooling tower. Typically, faster wind speeds because more blockage of the hot air outlet. But it is important to remember that the higher wind speed above the outlet of the cooling tower will lead to a decrease in the pressure level there, which leads to the drawing of hot, moist air out of the tower at higher speeds.

To determine how of the cooling tower performance is impacted by wind speed, pressure difference inside cooling tower is examined. Figure 8 shows the pressure difference between the inlet and outlet of the cooling tower at wind speeds. The pressure difference in the cooling tower is affected by wind speed, as seen in Figure 8. In actuality, a pressure difference seen between the faces of the cooling tower's input and output. As can be observed, wind speed has a detrimental effect up to 10 m/s. The performance of the cooling tower was found to be enhanced at high speeds of more than 10 m/s.

Figure 9 shows a decrease in the outlet mean pressure with wind speeds. As a result, a power plant's efficiency connected to the cooling tower can be effectively improved by this abrupt decline. The cooling tower's front and back are under dramatically different pressures when the wind speed increases. Because of this, a significant quantity of warm, humid air in the cooling tower can travel away from the fill downstream. As a result, there is a decrease in the warm air mass's flow rate at the cooling tower outlet.

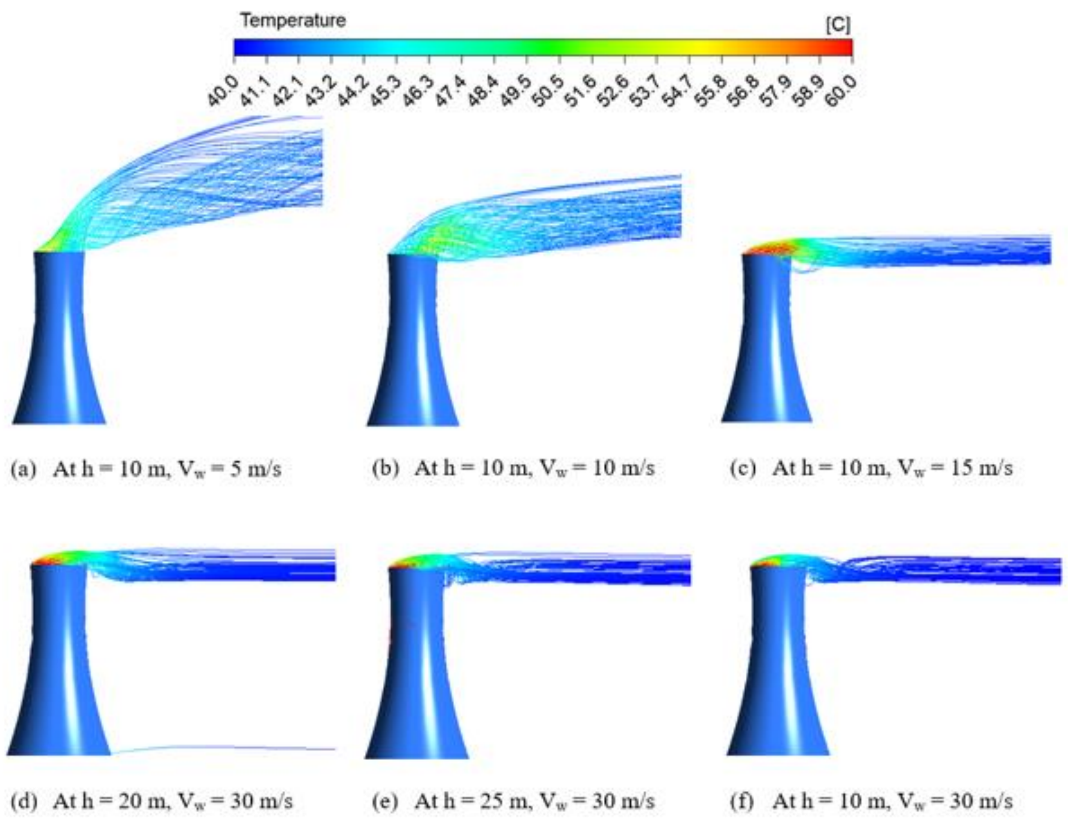


FIG 7. Streamlines coloured by temperature contours at different wind speed and at a height of h = 10 m

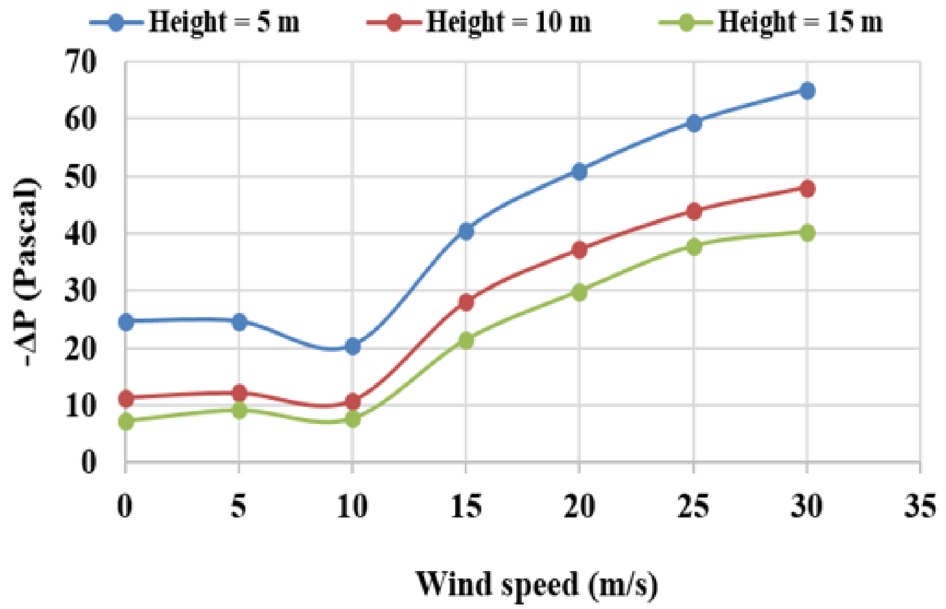


FIG 8. The pressure difference between inlet and outlet from the cooling tower at different heights and wind speeds

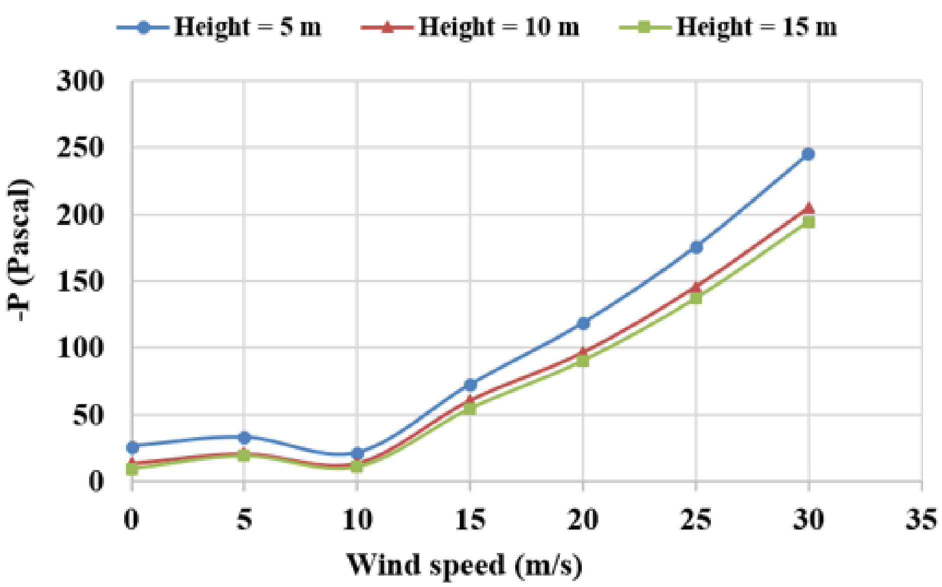


FIG9. The outlet mean pressure at different heights and wind speeds

Figure 10 shows the heat dissipation capacity of the cooling tower at different wind speeds and heights. The figure shows that at low wind speeds of less than 10 m/s, there is a slight decrease in heat loss and this indicates a poor performance of the cooling tower. However, at wind speeds of more than 15 m/s, heat loss increases significantly.

Figure 11 shows volumetric heat capacity variation at different cooling tower heights from ground and different crosswind speeds. As can be observed from Figure 11, high wind speeds cause the cooling tower to be thoroughly penetrated by the wind, which has a significant impact on the volumetric heat capacity. The volumetric heat capacity and heat dissipation capacity change rules do not agree, as shown by the comparison of Figures 10 and 11.

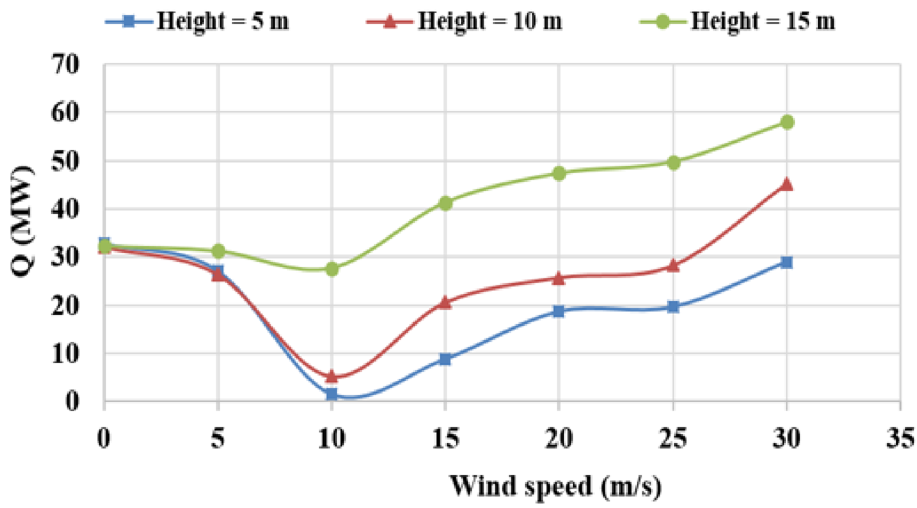


FIG 10. Heat dissipation capacity from cooling tower at different heights and wind speeds

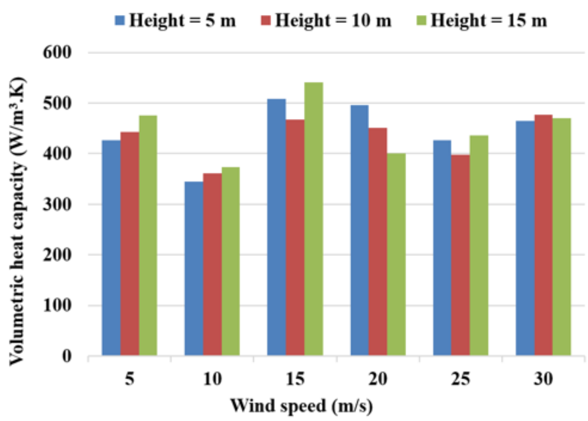


FIG 11. Volumetric heat capacity variation at different heights and wind speeds

CONCLUSIONS

A simulation was performed using the numerical analysis software of a real cooling tower subject to variable wind speeds. The results shown that when the wind speed increased, the pressure decreased and the heat dissipation efficiency of the cooling tower improved. However, the cooling tower starts to operate worse at about 10 m/s wind speed. To ensure the operation of a cooling tower with good performance, the different components of the tower and the effect of wind speeds on them must be studied.

The main results are as follows:

- Increasing the outlet speed of the cooling tower improves the performance, but the ratio of the outlet speed to the wind speed significantly affects the thermal performance of the tower.
- When the fan height is increased, the amount of rotation inside the tower decreases because the amount of rotation inside the tower works to trap heat and therefore the performance of the tower.
- There was an improvement in the thermal performance of the cooling tower when it was placed 15 meters above the ground.

REFERENCES

[1] Hensley, J.C., Cooling Tower Fundamentals, Kansas: SPX Cooling Technologies, Inc., (2009).

[2] Evapco Bulletin 311G, “Equipment Layout Manual”, (1999).

[3] Bhatia, A. (2001), "Cooling Towers," Continuing Education and, (2001).

[4] Patankar, S.V., Numerical Heat Transfer and Fluid Flow, McGraw-Hill Book Company, (1980).

[5] Rafat Al-Waked, Crosswinds effect on the performance of natural draft wet cooling towers, International Journal of Thermal Sciences, Volume 49, Issue 1, (2010), Pages 218-224.

[6] Du, Xiangdong, and Meiring Beyers. "Numerical Studies on Wind Effects on the Cooling Efficiency of Dry Cooling Towers." *The Fifth International Symposium on Computation Wind Engineering (CWE2010)*. 2010.

[7]Xiaoxiao Li, Hal Gurgenci, Zhiqiang Guan, Xurong Wang, Lin Xia” A review of the crosswind effect on the natural draft cooling towers” Applied Thermal Engineering 150 (2019), Pages 250–270.

[8] Guangjun Yang, Li Ding, Tongqing Guo, Xiaoxiao Li, Wenxin Tian, Zhen Xu, Zhigang Wang, Furong Sun, Junjie Min, Jingxin Xu, Sheng Wang, Zhaobing Guo, Study of flue gas emission and improvement measure in a natural draft dry-cooling tower

with flue gas injection under unfavourable working conditions, Atmospheric Pollution Research, Volume 11, Issue 5, (2020), Pages 963-972.

[9] A. Jahangiri, A. Borzooee, E. Armoudli "Thermal performance improvement of the three aligned natural draft dry cooling towers by wind breaking walls and flue gas injection under different crosswind conditions" International Journal of Thermal Sciences 137 (2019), 288–298.

[10] Ali Asghar Sedighi, Majid Bazargan "A CFD analysis of the pollutant dispersion from cooling towers with various configurations in the lower region of atmospheric boundary layer" Science of the Total Environment, Volume 696 (2019), 133939.

[11] Nicolas Blain, Antoine Belaud, Matthieu Miolane "Development and validation of a CFD model for numerical simulation of a large natural draft wet cooling tower" Applied Thermal Engineering, Volume 105, (2016), Pages 953-960.

[12] Lin Xia, Hal Gurgenci, Deyou Liu, Zhiqiang Guan, Ling Zhou, Pei Wang, CFD analysis of pre-cooling water spray system in natural draft dry cooling towers, Applied Thermal Engineering, Volume 105, (2016), Pages 1051-1060.

[13] Adam Klimanek, Michal Cedzich, Ryszard Bialecki "3D CFD modelling of natural draft wet-cooling tower with flue gas injection" Applied Thermal Engineering, Volume 91, (2015), Pages 824-833.

[14] Yuanshen Lu, Hal Gurgenci, Zhiqiang Guan, Suoying He, The influence of windbreak wall orientation on the cooling performance of small natural draft dry cooling towers, International Journal of Heat and Mass Transfer, Volume 79, (2014), Pages 1059-1069.

[15] Hooman, K., "Theoretical Prediction with Numerical and Experimental Verification to Predict Cross Wind Effects on the Performance of Cooling Towers," Heat Transfer Engineering, Volume 36, Issue 5, (2015).

[16] Gaoming ,G., Fu, X., Shengwei W., and Liang, P., "Effects of discharge recirculation in cooling towers on energy efficiency and visible plume potential of chilling plants," Applied Thermal Engineering, Volume 39, (2012), Pages 37-44.

[17] Pascal Stabat, Dominique Marchio, Simplified model for indirect-contact evaporative cooling-tower behavior, Applied Energy, Volume 78, (2004), Pages 433–451.

[18] Zhao Yuanbin, Sun Fengzhong, Li Yan, Long Guoqing, Yang Zhi, Numerical study on the cooling performance of natural draft dry cooling tower with vertical delta radiators under constant heat load, Applied Energy, Volume 149, (2015), Pages 225–237.

[19] J.H. Yu, Z.G. Qu, J.F. Zhang, S.J. Hu, J. Guan, Comprehensive coupling model of counter-flow wet cooling tower and its thermal performance analysis, Energy, Volume 238, (2022), 121726.

[20] Deying Zhang, Rui Chen, Zhengqing Zhang, Suoying He, Ming Gao, Crosswind influence on heat and mass transfer performance for wet cooling tower equipped with an axial fan, Case Studies in Thermal Engineering, Volume 27, (2021), 101259.

[21] Yuanshen Lu, Hal Gurgenci, Zhiqiang Guan, Suoying He, The influence of windbreak wall orientation on the cooling performance of small natural draft dry cooling towers, International Journal of Heat and Mass Transfer, Volume 79 (2014), Pages 1059–1069.

[22] Gaoming Ge, Fu Xiao, Shengwei Wang, Liang Pu, Effects of discharge recirculation in cooling towers on energy efficiency and visible plume potential of chilling plants, Applied Thermal Engineering, Volume 39 (2012), Pages 37-44.

[23] Essam E. Khalil, and Ahmed ElDegwy, "Flow Regimes around Industrial Cooling Towers", AIAA 2022-0109, San Diego, CA & Virtual, USA, January 3-7, (2022).

[24] Z.Q. Zhang, M. Gao, Z.G. Dang, et al., An exploratory research on performance improvement of super-large natural draft wet cooling tower based on the reconstructed dry-wet hybrid rain zone, Int. J. Heat Mass Transfer, Volume 142, (2019) 118465.

[25] ANSYS Inc., FLUENT User's Guide, ANSYS Inc., New Hampshire, USA, (2011).

[26] S.M. Yang and W.Q. Tao, “Heat Transfer,” 4th Edition, Higher Education Press, Beijing, 2006., Pages 266–271.

[27] Z.G. Dang, Z.Q. Zhang, M. Gao, et al., Numerical simulation of thermal performance for super large-scale wet cooling tower equipped with an axial fan, Int. J. Heat Mass Transfer, Volume 135, (2019), Pages 220–234.

[28] Deying Zhang, Rui Chen, Zhengqing Zhang, Suoying He, Ming Gao, Crosswind influence on heat and mass transfer performance for wet cooling tower equipped with an axial fan, Case Studies in Thermal Engineering, Volume 27, (2021), 101259.

[29] J. Juda, S. Chrosciel, Protection of the Atmospheric Air, WNT, Warsaw, Poland, (1974).

[30] R. Grybos, S. Tomaszek, Climate Change Processes above Industrialized Area, WPS, Gliwice, Poland, (1997).

[31] M. Lucas, P.J. Martínez, J. Ruiz, A.S. Kaiser, A. Viedma, On the influence of psychrometric ambient conditions on cooling tower drift deposition, International Journal of Heat and Mass Transfer, Volume 53, Issue 4, 2010, Pages 594-604.