

Numerical simulation of plume abatement and water-saving device on the top of wet cooling tower

Liu Wenming[2,3], Wen Chuanmei[2], Liu Xuedong[2,3,4], Ma Qian[2]

Abstract. Heated humid air from a cooling tower forms a visible plume and carry part of the water. The objective of this paper is to report the performance of plume abatement and water-saving device on the top of cooling tower. Computation fluid dynamics analysis and orthogonal test is conducted to investigate the influence of the height of gas chamber, the size of the nozzle and the length of mixing section on gas mixing homogeneity. The gas mixing homogeneity of the two streams is evaluated by the standard deviation of the volume fraction of the outlet liquid. The smaller the standard deviation, the more uniform the mixing. Then, the influence of the factors on gas mixing homogeneity of the two streams was analyzed by observing the contour of the liquid water where in the center of the device. Finally, proposed a kind of structural improvement method base on the conclusion.

Key words. Cooling tower, plume abatement and water-saving device, numerical simulation, orthogonal experimental design.

1. Introduction

Over the past ten years or so, a great deal of attention has been given to sustainable energy production and the environmental protection. For instance, reduce the consumption of industry. It is known to all that wet cooling towers are widely used in textile, petrochemical, papermaking, metallurgy, thermal power generation, nuclear power and other high water-consuming industries. Wet cooling towers which running in low temperature or high humidity will lead to evaporation loss of circulat-

[1]Acknowledgement - This paper is supported by the Jiangsu Province Industry University Research Joint Innovation Fund with project number: BY2016029-16.

[2]Workshop 1 - School of Mechanical Engineering, Changzhou University, Changzhou 213164, China

[3]Workshop 2 - Jiangsu Key Laboratory of Green Process Equipment, Changzhou, 213164, China

[4]Corresponding author: Liu Xuedong; e-mail: `lwm81@126.com`

ing water and producea lot of plume which spreads with wind, reducing the visibility around the area, increasing of humidity in the downwind area and affecting the operation status of the surrounding equipment and facilities [1]. Therefore, it is great significance to abatement plume on cooling tower.

At present, the research on cooling tower plume focuses on three aspects. The one is to study the formation mechanism of plume caused by the drift droplets and the evaporation of circulating water in cooling tower. The drifting droplets condense the water vapor in cooling-wet air that aggravates the plume phenomenon [2-3]. The other is to study the pattern of growth and spread of plume in the environment which are related to the drifting droplet diameter, the wet air state out of the tower, the ambient air state and the wind direction and speed [4-8]. The third is to study plume abatement according to wet cooling tower plume formation mechanism. The effect of plume abatement can be better through the application of wet and dry combination or increasing solar collectors to improve the original wet cooling tower [9-10]. However, this method not only requires a lot of money invested, but increases the cooling tower volume. In addition, there are also control of the formation of plume by adjusting the operating parameters and this method is not suitable for single cooling towers [11].

The above researches mainly focus on the growth and spread forms of the plume and apply plume formation mechanism to eliminate mist. However, the research on the plume abatement device is still rare. In this paper, a new type of plume abatement device is proposed, and the pressure and velocity inside the device are simulated by computational fluid dynamic (CFD) method. Based on the numerical simulation results, the mixing performance, plume abatement performance and water saving rate of the device are calculated and analyzed then the plume abatement device is optimized.

2. Structure and principle of abatement plume and water-saving device

2.1. Structure of plume abatement device

In this paper, plume abatement and water-saving device on the top of wet cooling tower which fixed in the mechanical draft cooling tower duct outlet. It is mainly composed of gas chamber, jet section, mixing section, condensate water collection area and set sink and is covered by fiberglass panels, structureof plume abatement device is shown in Figure 1. The gas chamber is composed of an upper inner seal plate and a lower inner seal plate and the inlet of the gas chamber link with duct. The left and right sides of the gas chamber are respectively provided with a tapered jet port which is as wide as the tower body.The outside of the jet port is a mixing zone and the outside of mixing zone is the section of condensate collection where is a water collector. The sections under the condensate collection zone are water collection tank and water outlet. According to the principle of the generation of plume, heated humid air and cold and dry ambient air in the mixing section can forms plume and the temperature and humidity of heated humid air can be reduced.

The plume is collected by water collector achieve water saving and reducing the formation of the exhaust fogs in the exhaust air outlet of the cooling tower.

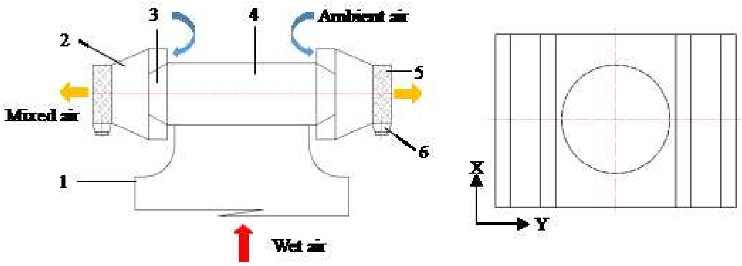

Fig. 1. Structure of plume abatement and water-saving condensing device

2.2. Principle ofplume abating and water-saving

The plume generation and elimination theory is shown in Figure 2.Heated humid air B is ejected into the mixing zonefrom both sides of the jet sectionafter entering the gas chamber.A negative pressure area is formed around the jet section due to the high flow rate of heated humid air, and this phenomenon is called Venturi effect.Ambient air A (cold source) is drawn into the mixing zone then mix with wet air from the cooling tower and condense water vapor to form supersaturated air C.Supersaturated air C is further condensed in the condensation and water collection zone and then the water droplets were collected by water collector to saving water. The exhaust air C′ from abatement plume and water-saving device incorporate with ambient air A and the condition line is C′A that in the unsaturation area. Therefore, the abatement plume and water-saving technology is capable of achieve the function of plume abating and water saving.

The mixing process of ambient air A and heated humid air B follows the law of conservation of mass and the law of conservation of energy. The ambient air mass flow is $m_A$and the wet air mass flow is $m_{B,}$ the mass flow of the mixed air C′ is m_A+$m_{B.}$ The mass conservation equation and the energy conservation equation are shown in equations (1) and (2)

$$m_A(1 + d_A) + m_b(1 + d_B) = (m_A + m_b)(1 + d_C) + w \tag{1}$$

$$m_A h_A + m_b h_b = (m_A + m_b)h_C + c_{w,p} w T_C \tag{2}$$

whered_A, d_B, d_C, h_A, h_B and$h_{C,}$respectively, ambient air moisture content, wet air moisture content, mixed air moisture content, ambient air enthalpy, wet air enthalpy and mixed air enthalpy. Moreover, w *is* the mass flow of liquid water,$c_{w.p}$is specific heat capacity of liquid water, $T_C$is mixed gas temperature.

When two fluids with different temperature and humidity meet in mixing section, the temperature and humidity of mixing air depends on the temperature, humidity and mass flow rate of two fluids. The mass flow rate of two fluids can be calculation according to the temperature and humidity of ambient air A and heated humid air B.

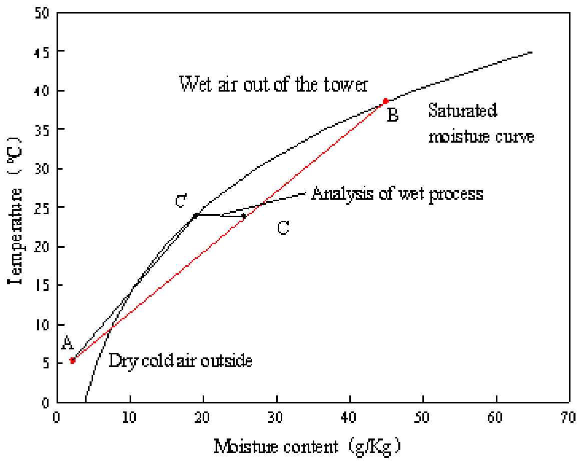

Fig. 2. Plume generation and elimination theory

2.3. Performance evaluation standard of plume abating and water-saving device

The principle of the device is mix ambient air and heated humid air in the plume to reduce the humidity and temperature of heated humid air and eliminating plume. Two fluids mixing uniformity is index of eliminating plume, most commonly measured as standard deviation of liquid water in outlet.

In addition, the device also has a water-saving function, which is mainly condensate water vapor in hot humid air and collect the condensate water. Therefore, evaluation of water saving performance by the recovery rate of water vapor.

2.3.1. Mixing performance of plume abating and water-saving device The mixing effect of gas and solid is judged by the value of mixing uniformity. But the colorless and tasteless of ambient air and heated humid air, the mixing uniformity can't be used to evaluate the mixing effect of the air streams. There is mixing of saturated hot humid air and unsaturation cold dry ambient air, water vapor of saturated hot humid air will surely be condensed into liquid water. And that the better the mixing effect, the better the effect of condensation, the more water be produced. Therefore, the standard deviation of the volume fraction of the liquid water can be used to evaluate the mixing performance of the device. The calculation formula is as following: (3).

$$S=\sqrt{\frac{\sum_{i=1}^{n}(x_i-\overline{x})^2}{n}} \tag{3}$$

where S is the standard deviation of the volume fraction of liquid water at the outlet; $x_i$is the volume fraction random variable of the liquid water at the exit; $\overline{x}$is the average of the volume fraction of liquid water at the exit; $n$is the number of random variables.

The smaller the standard deviation S of the volume fraction, the more uniform the liquid water distribution, that is, the less homogeneous the two air mixtures are.

2.3.2. Water vapor recovery rate Wet air which has more moisture content and more water content is in saturation or near saturation after the cooling tower heat., The temperature and humidity of wet air decreased through the application of the mixed condensate in this device. The effect of saving waterachieves because part of the precipitation of water vapor into liquid water. Therefore, the recovery of water vapor can be used as a basis for judging the water-saving performance of the device. The calculation formula of water vapor recovery $\eta_{H2O}$is as following (4).

$$\eta_{H_2O} = \frac{M_1 - M_2}{M_1} \tag{4}$$

where M_1 is wet air quality and $M_1 = Q_{V1}\rho_{V1}h$, $Q_{V1}$is wet air volume flow, M_2 is exit air quality and $M_2 = Q_{V2}\rho_{V2}h$,Q_{V2} is mixed air volume flow, ρ_V is water vapor density and $\rho_V = \frac{p_V}{TR_V}$, p_V is water vapor partial pressure, T is the air temperature, R_V is the gas constant of water vapor valued 461.53.

3. Numerical Simulation

3.1. Orthogonal experiment

The physical model for the device is based on the experimental cooling tower (5.75mm4.25mm). In order to explore the impact of structural parameters on the ambient air inlet flow and the mixing effect of dry and cold air with wet air, the previous theoretical calculation was carried out with the combination of plume abatement device and installation site area. After preliminary analysis, the air chamber height, jet taper and mixing chamber length of the three main factors affect the ambient air inlet flow and mixing effect. Each factor is considered three levels, the specific circumstances shown in Table1.

Table 1. influencing factors

Factor Level	Air chamber height E/m	Injection port Taper F	Mixing room length G/m
1	1.2	0.366	1.5
2	1.5	0.417	2
3	2	0.487	2.5

If every level of each factor is paired with a full test, it will take 27 times which is a huge amount of work. According to the method of orthogonal test, 9 representative groups were selected for orthogonal test. The orthogonal test table is shown in Table 2.

Table 2. orthogonal experiment table

Factor Number	E	F	G
1	1	1	1
2	1	2	2
3	1	3	3
4	2	1	2
5	2	2	3
6	2	3	1
7	3	1	3
8	3	2	1
9	3	3	2

3.2. CFD model

Application Pro/E to establish the physical model, the model wall all reduced to the surface. The physical model is meshed using ANSYS ICEM structured meshing method. The more detailed the mesh, the more accurate the result is. However, when the number of grids is increased to a certain extent, the load of the computer is increased, resulting in a slow calculation speed. Therefore, this paper does not verify the number of grids and finally determine the number of grid is about 250,000. Calculate the model mesh as shown in Figure 3.

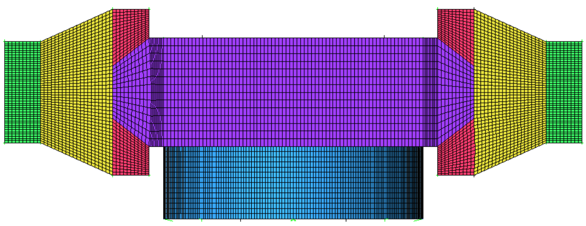

Fig. 3. Model grid

3.3. Boundary conditions

Simulation use SIMPLE algorithm and transient calculation. Turbulence model use standard k-ϵ model and multi-phase flow model use Mixture model. The device will condense water vapor which should use evaporative condensation model. The relationship between the dew point temperature and the saturation pressure is defined using Piecewise-linear. The generation of plumes usually occurs in the weather with low temperature or high humidity. In this paper, the boundary conditions are calculated based on the average temperature in the south of winter.

1. Wet air inlet use fan inlet, the pressure is 120Pa, the temperature is 40○C (318.15K) and the relative humidity is 100%. Inquire about the enthalpy diagram of air, the moisture content of wet air is 65.021 g/Kg, and the mass fraction of water vapor in wet air is calculated as 8%.
2. Air inlet use pressure inlet. Give the ambient air inlet a pressure value of 10Pa in order to calculate more stable, then changed to 0Pa. The temperature is 5○C (278.15K) and the relative humidity is 40%. Inquire about the enthalpy diagram of air, the moisture content of dry and cold air is 2.25 g/Kg, and the mass fraction of water vapor in dry and cold air is calculated as 2%.
3. Export use pressure outlet, static pressure is 1.013MPa.
4. Water section applied porous media simulation packing and condensation water-saving devices used in all wall FRP which are set to adiabatic wall.

4. Results and discussion

4.1. GAnalysis of pressure and speed of device

This device mainly uses Venturi effect to introduce ambient air into the device and decrease the number of ambient air pushing device. Based on the numerical simulation of each group structure, the pressure distribution and velocity distribution inside the device are obtained.

4.1.1. Minimum pressure and inlet speed of ambient air According to the Venturi effect, negative pressure is generated at the upper and lower parts of the injection port of the device and the ambient air is drawn into the mixing chamber. Figure 4 shows the minimum pressure of each device and the ambient air inlet and outlet velocities. As can be seen from Fig. 4, the minimum pressure decreases gradually from combination 1 to combination 9, while from combination 7, the minimum pressure tends to be steady. The average velocities of the upper and lower entrances are on an upward trend, and the combination 4 to 9 tends to be steady. From the point of view of the minimum pressure and the average speed of the ambient air inlet, the combination of the structural sizes of the combinations 7, 8, 9 is optimal.

4.1.2. Internal device pressure and velocity distribution The internal pressure and velocity analysis was done by selecting combination 9 (E3F3G2) from the optimum combination of structural parameters with speed and minimum pressure. Figure 5 (a) shows the pressure distribution cloud of the plane X = 2785m, there are negative pressure in the upper and lower jetting mouth and mixing section. At

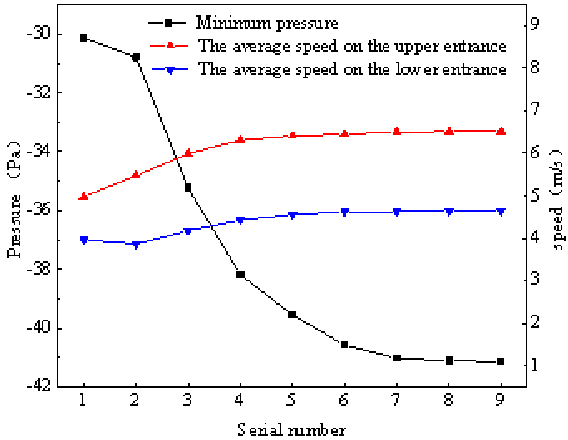

Fig. 4. device minimum pressure and inlet average speed

the plane X = 2785m, the negative pressure generating region is as shown in Fig. 5 (b). Negative pressure generated area in line with Venturi effect. However, negative pressure is also generated at the upper and lower edges of the water receiving section, mainly due to the jet of hot and humid air velocity, while the upper and lower edge of the air velocity is relatively small, there will have some reflux phenomenon at the bounded outlet. In the lower ambient air inlet section, there is no negative pressure in some area which is due to air moving from low to high part of the kinetic energy will be lost, resulting in the air velocity in the region is less than the speed of the entrance. In the air chamber and the pressure at the hair dryer is larger, mainly by the hot and humid air is drawn out by the fan part of the gas hit the ring on the ring chamber caused.

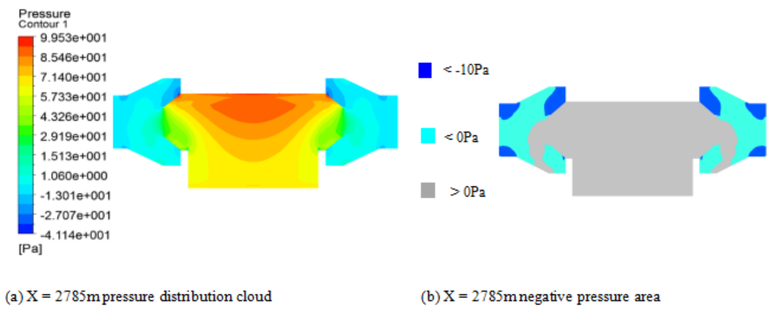

Fig. 5. Pressure distribution contours

Device internal X = 2785m plane velocity vector diagram is shown in Figure 6. When wet air is discharged from the blower tube, it is divided into two strands and enters the jetting section. The velocity of the gas becomes larger and the ambient air is sucked into the mixing section. In the mixing chamber, the velocity of the hot and humid air stream differs greatly from that of the ambient air, and the mixing is not uniform and can only be mixed at the edges of the hot and humid air jets. However, when the airflow enters the water receiving section, the two airflows are relatively evenly mixed due to the resistance of the water-absorbing filler and the effect of increasing the airflow mixing effect.

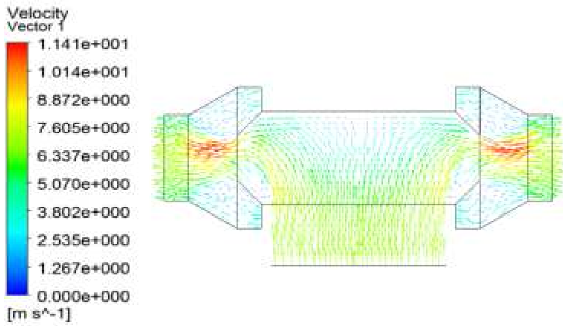

Fig. 6. Velocity vector of Plane X =2785m

4.2. Analysis of mixing uniformity

4.2.1. Analysis of mixing uniformity Read the outlet of each combination of liquid water volume fraction. 10904 sets of data are read and analyzed for each level. The results are shown in Table 3.

Table 3. Average volume fraction and the standard deviation of the liquid at the outlet

combination Serial number		result	
		$\overline{x}$	S
1		1.9862×10^{-5}	1.53×10^{-5}
2		2.3484×10^{-5}	1.35×10^{-5}
3		5.8374×10^{-5}	1.13×10^{-5}
4		5.3796×10^{-5}	1.25×10^{-5}
5		6.3798×10^{-5}	1.31×10^{-5}
6		6.5145×10^{-5}	1.05×10^{-5}
7		6.4439×10^{-5}	1.21×10^{-5}
8		6.5511×10^{-5}	1.02×10^{-5}
9		6.8794×10^{-5}	0.98×10^{-5}

The results show that when the height of the air chamber is constant, the larger the taper of the jet and the longer the mixing section, the larger the maximum volume fraction of liquid water generated and the more uniform liquid water distribution. The mixing is most uniform when the average volume fraction of E3F3G2 is the largest and the standard deviation is the smallest. As shown in Figure 7, the maximum liquid volume fraction of each combination is not much difference, but the minimum liquid volume fraction difference is very big. Since there are two areas where gas is well mixed at all levels, the maximum liquid volume fraction in these areas is similar. But the non-uniform mixing area will have two different proportions of airflow mixing. Therefore, the difference of the liquid volume fraction is big.

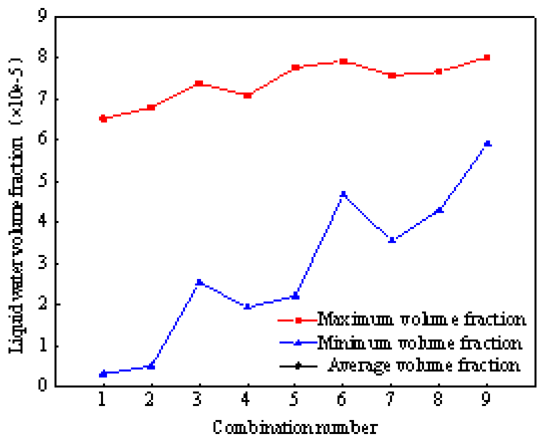

Fig. 7. Volume of liquid water at the outlet

4.2.2. Analysis of the internal mixing uniformity of the device Figure 8 shows the distribution cloud of volume fraction of liquid water in the center plane of combination 9 (E3F3G2). When hot and humid air is jetted out of the jet port at a high speed, a negative pressure is generated above and below the jet port so that ambient air is sucked into the mixing chamber. However, the relatively large difference between the jet velocity and the ambient air inlet velocity results in a poor mixing effect in the mixing section. Only the edge of the jet flow can exchange heat with the ambient air to generate liquid water. Two non-uniform flow of air enter the receiving section through the mixing chamber. There is some resistance due to packing arrangement, helping to mix two kinds of air so that there is more liquid water in the receiving section. The reason for the liquid water being generated at the edge of the gas chamber through which the hot fluid flows is that the temperature of all the wall surfaces of the apparatus is set to 300K when the boundary conditions are set.

4.3. Water vapor recovery rate in the device

According to the above analysis, the best combination of structure parameters of velocity and pressure distribution is E3F1G3, E3F2G1 and E3F3G2, and the best combination of mixing effect is E3F3G2. Therefore, the combination of E3F3G2 is

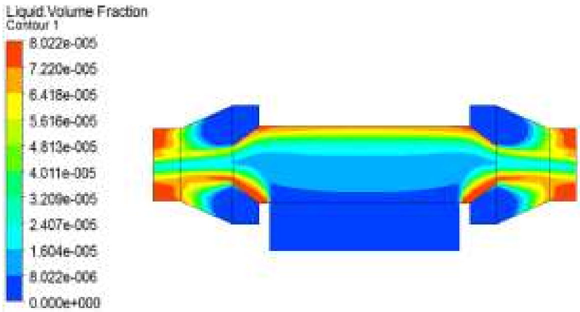

Fig. 8. Liquid water volume fraction distribution cloud of X = 2785m

the optimal combination of structural parameters. The average volume fraction of liquid water at the outlet of the combination is 6.8794×10^{-5}. If there is six months a year to produce the plume,the average exit speed is 7.012m/s. According to the formula (4), the annual water saving can be calculated as 350408.502m3 and the water vapor recovery rate is 0.32%.

4.4. Analysis of structural refinement apparatus

The average temperature of E3F3G2 combination outlet is 36○C (309.15K). As can be seen from FIG. 1, the moisture content of the mixed air C at this time is 40 g/Kg. The moisture content is large and the plume elimination is not obvious. The device studied in this paper has no wind-driven device at the entrance of ambient air. The ambient air inhaled by the Venturi effect is less. Fan flow can be calculated according to formula (1) (2).

5. Conclusion

According to the numerical simulation study, the combination of E3F3G2 is the optimal combination of structural parameters, having better pressure and velocity distribution inside the device. At the mixing section, the mixing is mainly at the edge of the jet due to the high jet speed of the wet air, resulting in unsatisfactory mixing effect. However, the mixing effect is enhanced due to the filler after entering the water receiving section.

Under the ambient air temperature of 5?C and the relative humidity of 40%, saturated wet air with a temperature of 45 C enters the device at a flow rate of 6.9m/s and passes through 6 months, the water which is saved by recovering the plume is about $350000m^3$.

When the temperature of the ambient air is too low, the amount of ambient air is sucked in by using the venture effect of the device can only eliminates a small part of the plume. An external wind force pushing device needs to be installed at the ambient air inlet to increase the ambient air flow to enhance the plume abatement effect of the device.

References

[1] S. K. Tyagi, A. K. Pandey, P. C. Pant: *Formation, potential and abatement of plume from wet cooling towers: A review.* Renewable and Sustainable Energy Reviews *16* (2012), No. 5, 211–220.

[2] Deziani. M, Rahmani, Roudaki. S. J. M: *Feasibility study for reduce water evaporative loss in a power plant cooling tower by using air to Air heat exchanger with auxiliary Fan.* Desalination *406* (2015) 257–262.

[3] R. H. Gutierrez, P. A. A. Laura: *Experimental measurement of cooling tower emissions using image processing of sensitive papers.* Atmospheric Environment *69* (2013), No. 3, 171–180.

[4] A. J. Consuegro, A. S. Kaiser, Zamora. B: *Numerical modeling of the drift and deposition of droplets emitted by mechanical cooling towers on buildings and its experimental validation.* Building and Environment *78* (2014), No. 3, 41–52.

[5] K. Takata, T. Michioka, R. Kurose: *Prediction of a Visible Plume from a Dry and Wet Combined Cooling Tower and Its Mechanism of Abatement.* Atmosphere *7* (2016), No. 4, 59.

[6] F. Sánchez, A. S. Kaiser, B. Zamora: *Prediction of the lifetime of droplets emitted from mechanical cooling towers by numerical investigation.* International Journal of Heat and Mass Transfer *89* (2015) 521–528.

[7] A. Chahine, P. Matharan, D. Wendum: *Modelling atmospheric effects on performance and plume dispersal from natural draft wet cooling towers.* Journal of Wind Engineering and Industrial Aerodynamics *136* (2015) 555–563.

[8] J. H. Lee. M. Moshfeghi, Y. K. Choi: *A numerical simulation on recirculation phenomena of the plume generated by obstacles around a row of cooling towers.* Applied Thermal Engineering *72* (2014), No. 1, 198–210.

[9] M. Lucas, P. J. Martínez, J. Ruiz: *On the influence of psychrometric ambient conditions on cooling tower drift deposition.* International Journal of Heat and Mass Transfer *53* (2010), No. 4, 587–606.

[10] S. Wang, X. Xu: *Effects of alternative control strategies of water-evaporative cooling systems on energy efficiency and plume control: A case study.* Building and Environment *43*, (2008), No. 11, 264–275.

[11] Y. Kumar: *Effects of discharge recirculation in cooling towers on energy efficiency and visible plume potential of chilling plants.* Applied Thermal Engineering *39* (2012) 589–597.

Received November 16, 2017