A study on multi-nozzle arrangement for spray cooling system in natural draft dry cooling tower

Yubiao Sun, Zhiqiang Guan, Hal Gurgenci, Xiaoxiao Li, Kamel Hooman

PII: S1359-4311(17)31333-9
DOI: http://dx.doi.org/10.1016/j.applthermaleng.2017.05.157
Reference: ATE 10472

To appear in: Applied Thermal Engineering

Received Date: 27 February 2017
Revised Date: 11 May 2017
Accepted Date: 27 May 2017

Please cite this article as: Y. Sun, Z. Guan, H. Gurgenci, X. Li, K. Hooman, A study on multi-nozzle arrangement for spray cooling system in natural draft dry cooling tower, Applied Thermal Engineering (2017), doi: http://dx.doi.org/10.1016/j.applthermaleng.2017.05.157

This is a PDF file of an unedited manuscript that has been accepted for publication. As a service to our customers we are providing this early version of the manuscript. The manuscript will undergo copyediting, typesetting, and review of the resulting proof before it is published in its final form. Please note that during the production process errors may be discovered which could affect the content, and all legal disclaimers that apply to the journal pertain.
A study on multi-nozzle arrangement for spray cooling system in natural draft dry cooling tower

Yubiao Sun1, Zhiqiang Guan, Hal Gurgenci, Xiaoxiao Li, Kamel Hooman

Queensland Geothermal Energy Centre of Excellence,
School of Mechanical and Mining Engineering,
The University of Queensland, Brisbane 4072, Australia

Abstract:
Natural draft dry cooling tower (NDDCT) technology is especially attractive to power plants built in arid regions with limited water resource. However, high ambient temperature in summer deteriorates the performance of built NDDCT. To address this problem, evaporative pre-cooling technology has been developed by using nozzles to disintegrate water into fine droplets to achieve quick evaporation. The pre-cooled air flowing through radiator, has an enhanced heat exchange with the hot working fluid in the tube side. This paper reports a spray cooling system for the experimental tower built in UQ by combining several nozzle LNN1.5 to cool the inlet air and consequently improve the cooling efficiency of the NDDCT. To minimize water usage, a careful arrangement of spray nozzles should be investigated to achieve the maximum cooling outcome. With five nozzles installment, the inlet air is cooled by 6.3 °C, corresponding to 51.2% cooling efficiency. A dimensionless analysis is presented to correlate cooling efficiency with influencing factors. The advantage of this pre-cooling system lies in the efficient water usage: more than 96% of the injected water extracts substantial heat from hot air and evaporates into vapor, leading to a pre-cooled airflow.

Keywords:
Natural draft dry cooling tower; full evaporation; spray cooling; multi-nozzle arrangement

1 Introduction
For both thermal power plants and air conditioning industry, cooling towers are widely used to cool circulating water, which serve as a medium to transfer substantial waste heat to the surrounding environment. The cooling tower performance has a significant impact on the operation and efficiency of the whole power generation system. A defective cooling tower design, failing to provide adequate cooling for the power generation process, would lead to decreased electricity production and induce tremendous economic loss. In order to avoid such economic punishment, an effective cooling system is necessary for power plant normal operations.

In most power plants, mechanical and natural draft cooling towers are commonly used. However, the high running costs caused by the energy-consumptive motor-driven fans makes mechanical draught less attractive for many power plants, even though the capital costs are

1 Corresponding author: y.sun3@uq.edu.au, cesyb2013@gmail.com
generally higher for natural draft towers. The wet and dry cooling towers are the two commonly seen natural draft cooling towers. In wet ones, hot water, in direct contact with air, cools by releasing some heat into the surrounding air. Theoretically, wet cooling can cool hot water down to atmospheric wet bulb temperature and is considered as more effective than dry cooling. However, the large quantity of water consumption due to evaporation, drift and draining losses, requires a continuous water supplement. This huge water consumption as well as the environmental concerns such as the visible plume and entrainment and impingement issues make wet cooling tower unsuitable for the regions suffering from water shortage [1].

In arid areas, dry cooling towers, with the advantages of low water consumption, low maintenance cost and little parasitic loss, become a good choice for some thermal power plants to release the waste heat to the atmosphere by cooling down hot fluid to a lower temperature. Unfortunately, the convective heat transfer mechanism of dry cooling towers makes them inferior to the evaporative wet cooling towers [2]. More importantly, the performance loss becomes remarkable during high ambient temperature periods and under strong crosswind conditions [3].

Some researchers had conducted pioneering work to explore the tower performance loss caused by the crosswind. Wei et al. [4] used both experimental and theoretical methods to study the crosswind effects on the performance of dry cooling towers. They found that the unfavorable pressure distribution around tower entrance, the affected tower hot plume and the leading edge separation induced cool air contributed to reduce the tower cooling performance. Su et al. [5] simulated the thermal performance of dry tower affected crosswinds, and confirmed the declining thermo-dynamical effect of crosswinds. Zhao et al. furthered this study by considering the delta layout form of column radiators. They developed a three-dimensional (3D) numerical model to explore the cooling performance of a natural draft dry cooling tower with vertical two-pass column radiators (NDDCTV) [6]. Their conclusion was that the poor cooling performance of NDDCTV caused by crosswind would lead to a raised water exit temperature. Specifically, the worst scenario occurs at the 12 m/s crosswind condition, rising the water temperature by 6 °C when compared with the no-crosswind situation. More recently, Zhao et al. updated their research by coupling the ambient air temperature impacts with the crosswind influence on the performance of NDDCTV [7]. Simplifying the model with an assumption of constant heat load and uniform entry water temperature, they focused on analyzing the cooling performance of each sector under crosswinds. The deteriorating performance under crosswinds shows two patterns: for low crosswind velocity, the cooling performance of NDDCTV deteriorates sharply, while for high crosswind conditions, it experiences a slight variance.

The decreased heat rejection rate in summer days, as well as the susceptibility to the crosswind, contributes to the low acceptance of NDDCT [3]. Generally, power plants utilizing dry cooling technologies can experience a significant 20% net power reduction during high ambient temperature periods [8]. This is catastrophic for plants based on low temperature resources (e.g. geothermal plants) where the power output reduction can be as high as 50% in hot summer days [9,10]. What is worse, this issue is compounded since the reduction goes along with the peak power demand which means a greater loss for power plant owners with flexible electricity pricing.

Spray cooling provides a solution to overcome the poor tower performance caused by hot ambient conditions. This technology makes use of a controlled, small quantity of water to cool the inlet air on hot days. The method, known for its simplicity, low capital cost, and easy
operation and maintenance, has been previously reported and used in other industries [11]. Nozzle, as the core part of the spray system, is used to break bulk water into fine water droplets and distribute these droplets into the inlet air (Fig. 1). The large water-air contact surface area of fine droplets accelerates the evaporation process. Since the water flowrate is quite small, the air stream motion is barely affected and the pressure drop caused by the spray can be neglected [6]. The sensible heat of the hot ambient air feeds the evaporation of water droplets, and then a temperature drop follows. The pre-cooled inlet air improves the cooling tower performance and consequently increases the thermal efficiency of a power plant. Consequently, dry cooling towers assisted by the spray cooling contribute to higher power generation for power plants than that of pure dry-cooling towers.

Inlet air spray cooling technology has been practiced in the fields of food refrigeration [12] and gas turbine fogging [13,14]. This technology is reportedly in use in more than 1000 gas turbine stations [15]. Chaker et al. [16–18] made a series of studies on the physics and engineering applications of the fogging process in gas turbines, including droplet measurement methods, droplet kinetics, and the duct behavior of droplets. Montazeri et al. [19] made use of the Lagrangian–Eulerian approach to simulate spray cooling produced by a hollow-cone nozzle and concluded that CFD simulation can accurately predict evaporation process.

However, most publications on spray cooling deal with gas turbine fogging application, few efforts are made on pre-cooling for NDDCT. Since the cooling towers have such a huge difference from gas turbine in both physical geometry and working principles, the conclusions from previous researches cannot be applied directly to the cooling system design for cooling tower. To design a proper cooling system for NDDCT, the investigation of tower-directed spray cooling design ought to be conducted. Alkhedhair et al. [2] carried out a CFD study to simulate the NDDCT and developed a 3D numerical model to study the evaporation from a single spray nozzle. The results showed that up to 81% evaporation can be achieved for water droplets of 20 µm at the air velocity of 1 m/s and droplet transport and evaporation strongly depend on droplet size and air velocity. Wind tunnel test data confirmed the enhanced cooling effect at low air velocity and narrow water droplet distributions [20]. Xia et al. [21] furthered Abdullah’s work by studying the pre-cooling performance of a vertically arranged nozzle (VAN) and a horizontally arranged nozzle (HAN) installed in a wind tunnel. He found that the VAN configuration has better performance than HAN configuration in the inlet air velocity range of 0.8-1m/s. Another useful conclusion is that the increased turbulent intensity has a positive effect on the fully evaporated water flowrate. Sadafi et al. [22,23] used saline water rather than fresh water for spray cooling. They first performed a theoretical modelling to study the four-stage saline-water evaporation process, and verified their simulated results against experimental data.

Previously reported studies focused on the arrangement of a single nozzle. But in real situation, multiple nozzles are generally needed to cool tower inlet air. As far as we know, there are no reports on configurations of several nozzles for a cooling tower inlet air spray cooling systems. Filling this gap by studying nozzle arrangement to achieve the maximum cooling effect is necessary and important. In this study, the numerical study was conducted to get the optimum nozzle locations and injection directions for multi-nozzle arrangements to provide cooling for the University of Queensland Gatton test tower. A 3D CFD model was first developed to simulate this NDDCT to get the velocity field. Then this velocity field was used for spray cooling calculations. The relationship between the number of hollow-cone
nozzle LNN1.5 and the pre-cooling effect were unveiled. The temperature distributions at the 
heat exchanger surface corresponding to various nozzle configurations were also displayed.

2 Numerical Method

A water spray involves two-phase flow interaction and experiences heat, mass and 
momentum transfer when injected into air. This complex two-phase phenomenon makes 
experimental analysis costly and challenging. Fortunately, CFD provides a simple way to 
analyze spray cooling. For instance, it allows researchers to control the boundary conditions 
and physical parameters of the two-phase flow independently, which is almost impossible for 
experimental investigation. In our study, ANSYS FLUENT (version 16.2) was selected as the 
CFD tool to explore spray cooling options for the inlet air flowing through an NDDCT. 
Eulerian-Lagrangian methods are generally used to explore the interaction between the 
droplets (discrete phase) and the continuous phase (air). According to Elgobashi [24], there 
are two approaches to model the transport of water droplets in a turbulent air flow. The first 
one is the “one way coupling” where the influence of air on the droplets is considered while 
the air properties are not impacted by the existence of droplets. The second one is the “two-
way coupling” where the influence of the droplets on the airflow characteristics is large 
enough to affect the airflow. Therefore, modification to the airflow field governing equations 
is necessary to take into account the two-phase coupling. A more complicated case emerges 
when the droplet-droplet interaction has to be considered, i.e. “four way coupling” to include 
the momentum exchange of droplets [25]. The different coupling mechanisms are closely 
related to the volume fraction of discrete phase. The volume fraction is an indication of 
whether the spray is dilute or dense. For extremely dilute mixtures, one-way coupling can be 
considered and for dilute ones, the two-way coupling should be used. The four-way coupling, 
generally speaking, is only used together with the two-way coupling for dense ones [25]. In 
this study, the volume fraction of spray is low (less than 10%) and the influence of droplets 
on the airflow was taken into account by using the two-way coupling approach [26]. The 
coupling influence is quantified by means of an iterative process as illustrated in the flow 
chart (Fig. 2), based on the concept of Crowe [27].

2.1 Governing Equations

2.1.1 Continuous Phase

The airflow was modelled as a steady, incompressible, turbulent and continuous flow. The air 
flow field was described by the Reynolds-averaged Navier-Stokes conservation equations 
(RANS) combined with the standard k-ε model to account for the turbulence effects [28]. The 
governing equations of the airflow are given in the Eulerian modelling as [29]:

\[
\frac{\partial (\rho_u v_{ui})}{\partial x_j} = S_m
\]  

\[
\rho_u \frac{\partial (v_{ui})}{\partial x_j} = \rho_u g_i - \frac{\partial P}{\partial x_j} + \frac{\partial \tau_{ij}}{\partial x_j} - \frac{\partial}{\partial x_j} \left( \rho_u v_{ui} v_{uj} \right) + S_{mo}
\]  

\[
\rho_u v_{ai} \frac{\partial E}{\partial x_j} = -p \frac{\partial v_{ai}}{\partial x_j} + \frac{\partial}{\partial x_j} \left( K_a \frac{\partial T_a}{\partial x_j} \right) - \frac{\partial}{\partial x_j} \left( \rho_u c_v v_{ai} T_a \right) + \Phi + S_e
\]
\[
\rho_a v_{aw} \frac{\partial Y_i}{\partial x_j} = -\frac{\partial}{\partial x_j} \left( \rho_a D_f \frac{\partial Y_i}{\partial x_j} \right) - \frac{\partial}{\partial x_j} \left( \rho_a v_{aw} Y_i \right) + S_m \tag{4}
\]

The additional parameters \( S_m, S_{mo}, S_e \) are the source terms of droplet mass, momentum and energy, respectively. \( \tau_s \) is the stress tensor.

### 2.1.2 Discrete phase (water droplets)

In spray systems, water injected into the air quickly disintegrates on exit from the nozzle into droplets that follow their own trajectories. Simulating all these droplets individually needs tremendous computational resource. To reduce computational time, droplets are represented by a specified number of parcels equivalent to the entire spray. Each parcel contains identical particles sharing the same properties (diameter, velocity, trajectory, temperature, etc.). Only one droplet is computed to represent the whole parcel, assuming that all other droplets in the parcel behave in the same manner.

By modeling droplet trajectories via the Lagrangian framework, each discrete droplet is tracked individually within the air flow by integrating the motion equations governed by Newton’s second law and including the influence of the relevant forces from the air. As described earlier, by using the assumption that all droplets are isolated and have spherical shapes, adjustment in speed or direction of a droplet in air is brought mainly by air drag and gravity. The effect of turbulence on droplets is addressed by calculating the instantaneous air velocities in the time-averaged Navier-Stokes equations employing a stochastic velocity model as part of the particle tracking model.

In addition, the influence of droplets on the airflow was taken into account by using the two-way coupling regime. These source terms \( S_m, S_{mo}, S_e \) that appear in equations (1,2,3 and 4) are introduced to represent the mass, energy and momentum exchange of the droplets with air. These source terms are computed from the Lagrangian framework by an alternative process through volume averaging method and then incorporated into the Eulerian airflow RANS equations. For every computational cell, the volume averaged source terms are computed by collecting the influence of the number of droplets within the computational cell. Thus, the influence of droplets on the surrounding airflow is recognized. These source terms are given as [30]:

\[
S_m = -\frac{1}{V_{cell}} \sum_n \frac{d(m_d)}{dt}
\]
\[
S_{mo} = -\frac{1}{V_{cell}} \sum_n \frac{d(m_d V_d)}{dt}
\]
\[
S_e = -\frac{1}{V_{cell}} \sum_n \frac{d(m_d E_d)}{dt}
\tag{5}
\]

where \( V_{cell} \) is the volume of one computational cell and \( E_d \) is the total energy of a single droplet.
2.1.3 Momentum and Heat Exchange

The inlet air pre-cooling makes use of the latent heat corresponding to the evaporation of water droplets to take away the thermal energy from ambient air, resulting in cooler air flow. Once the sprayed water droplets contact with the dry, hot and unsaturated air, simultaneous heat and mass transfer occurs at the water-air interface. Compared with the latent heat transfer caused by mass transfer, the concurrent convective and radiative heat transfer are negligible [31]. The exposed water droplets would be covered by a film of saturated air-vapor. This film is responsible for heat transfer caused by the temperature difference between the water droplet and the unsaturated air. Meanwhile, mass transfer is observed when a vapor concentration gradient exists between the vapor layer and the ambient air. The rate of energy absorbed by each droplet can be expressed as:

\[ \dot{m}_wC_{pw}\Delta T_d = h_c \cdot S_d \cdot (T_a-T_d) + \frac{dm_d}{dt} \cdot \rho_{fg} \]  

The convection heat transfer coefficient, \( h_c \), is computed by using an empirical correlation from [32]:

\[ Nu = \frac{h_c \cdot d}{k_a} = 2 + 0.6Re_{ed}^{0.5} \cdot Pr_f^{0.33} \]  

\( \frac{dm_d}{dt} \) is the mass flux transferred to the air by evaporation and governed by the differences between the vapor densities at droplet surface and air:

\[ \frac{dm_d}{dt} = S_d h_D (\rho_{s,\text{int}} - \rho_{va}) \]  

where, \( h_D \) is the mass transfer coefficient and \( (\rho_{s,\text{int}} - \rho_{va}) \) is the water vapor mass density difference between the air and the saturated air-vapor layer. The mass transfer coefficient was obtained from the empirical correlation of Ranz and Marshall [32]:

\[ Sh = h_D \cdot \frac{\rho_{s,\text{int}}}{d_f} = 2 + 0.6Re_{ed}^{0.5} \cdot S_{f,33}^{0.33} \]  

\( Re_{ed} \) is the relative Reynolds number between the droplet and the airflow and is given as:

\[ Re_{ed} = \frac{\rho_a D_d |\vec{V}_r|}{\mu_a} \]  

where \( \mu_a \) and \( \rho_a \) are the dynamic viscosity (kg/ms) and density of air (kg/m\(^3\)). \( V_r \) is the droplet velocity relative to air \( |\vec{V}_r - \vec{V}_d| \) (m/s).

\( Sc_c \) is the the Schmidt number and written as:

\[ Sc_c = \frac{\mu_a}{\rho_a D_f} \]  

\( Pr_f \) is the Prandtl number and is defined as:

\[ Pr_f = \frac{\mu_a C_{p_a}}{K_a} \]
2.1.4 Droplet trajectory

The droplet trajectory can be determined by obtaining droplet velocity and consequently the droplet position.

$$\frac{d(X_d)}{dt} = V_d$$  \hspace{1cm} (13)

where $V_d$ is the droplet velocity (m/s); and $X_d$ is the droplet position (m).

Newton’s second law of motion was used to predict the velocity of an evaporating spherical droplet moving in a continuous airflow. The two-way coupling of air and droplet contribute to the heat and mass exchange with air. The motion equation of a single droplet can be written as:

$$\frac{d(m_d V_d)}{dt} = F_D + F_g$$  \hspace{1cm} (14)

Fig. 3 shows the forces exerted on a single spherical droplet. The forces acting on the single droplet include gravity force and drag force, which affect droplet trajectory when moving into air. The gravity force is expressed as:

$$F_g = m_d \ddot{g} = \frac{\pi}{6} D_d^3 \rho_a \ddot{g}$$  \hspace{1cm} (15)

Where $F_g$ is the gravity force (N), and $\ddot{g}$ is the gravitational acceleration (9.81 m/s$^2$).

The drag force acts in the direction opposite to the relative velocity between the droplet and airflow. This resistant drag force depends on the droplet shape and size, the relative velocity of the droplet with respect to the air and the viscosity and density of the air [33]. All these influencing factors are accounted for in the drag coefficient. For a spherical drop, the drag force is

$$F_D = -\frac{\pi}{8} C_D \rho_a D_d^2 \dot{V} \left| \dot{V} \right|$$  \hspace{1cm} (16)

where $C_D$ is the drag coefficient and $\dot{V}$ is the droplet relative velocity (m/s). $C_D$ is a function of the droplet Reynolds number and the shape of the droplet. Here an assumption of a spherical droplet shape is made, so the drag coefficient becomes a function of droplet Reynolds number only [34]. Dozens of empirical correlations have been proposed in the literature to calculate drag coefficients of a spherical droplet moving in the air. In this study, the Morsi and Alexander correlation for spherical drag coefficient was selected for it is quite popular and valid for a wide range of Reynolds number, from 0.1 up to 50,000 [35]. This correlation has the same formulation with varied constants dependent on the Reynolds number. The Morsi and Alexander drag coefficient correlation is expressed as:

$$C_D = a_1 + \frac{a_2}{R_{ed}} + \frac{a_3}{R_{ed}^2}$$  \hspace{1cm} (17)

where $a_1$, $a_2$, and $a_3$ are constants for different range of Reynolds numbers (Table 1).
2.2 Computational Model

2.2.1 Model Geometry

The subject of this study is an experimental tower built at the University of Queensland Gatton campus (Fig. 1). The 20m-tall tower has a hyperbolic shape and the diameter is 12.525m at both the heat exchanger level and at the top exit. The minimum diameter is 10.213m. The heat exchanger is horizontally placed at the height of 5m from ground. In view of the small variation in the tower diameter, a cylinder is used to model this hyperbolic cooling tower to facilitate the simulation process. Since our experimental tower has a smaller narrowing effect (throat diameter/base diameter:10.3/12.525=0.82) than that of an industrial counterpart (throat diameter/base diameter:113.6/177.6=0.64) [36], it is reasonable to neglect this small diameter variation. Additionally, the small tower size (20m) and the limited capacity of installed radiator (1.2MW) make it quite difficult to produce large natural draft. Therefore, the induced airflow has a low velocity, leading to a small airflow acceleration based on the narrowness at the tower throat. Another reason for this simplification is that despite the hyperbolic tower can produce a slightly different velocity field inside the cooling tower, our focus is the spray simulation, which is more related to the velocity distribution at the bottom of the tower rather than the field inside the tower. Hence this simplification would be acceptable. More importantly, the simulated results based on cylinder geometry have a good agreement with the experimental data, which gives us confidence that the simplification is reasonable.

The model configuration, dimensions and boundary conditions are illustrated in Fig. 4. Considering the symmetry of the cylinder and computational cost, a 30 degree wedge is used to represent the cooling tower. The smaller 30° partial cylinder representing cooling tower is placed within a much larger cylinder section, which represents the large surrounding air domain. The height of the air domain is 120m and the radius 80m. Such a large computational domain guarantees that the airflow inside the cooling tower was fully developed so all the necessary features of the velocity field can be captured and used for further calculations.

Natural draft resulted from the buoyancy effect was numerically simulated based on the model shown in Fig. 4(a). The mesh independent test results were summarized in Table 2. The test result shows that 2,239,000 cells is capable to give accurate results. Increased cell number would not make a big difference in the obtained air velocity and heat exchanger temperature. Structured mesh with 2,239,000 cells was used to discretize the computational domain (Fig. 4(b)). The geometry (Fig. 4(c)) used for water spray calculation is much smaller than that for air velocity calculation. It should be noted that in the lower part of tower, a wall cover with a radial length of 3m was installed, aligning with the heat exchanger surface. The reason to introduce the wall cover is to reduce the blockage caused by the vortex near the peripheral of the radiator so that the pre-cooled air could flow upward through the radiator peripheral part. To investigate the effects of the wall cover the velocity distribution at the mid-plane is presented for two cases with and without wall cover as shown in Fig. 5.

For the case without wall cover, the air velocity is either horizontal or tends to move down. Near the rectangular corner surrounded by the tower wall and the heat exchanger, there is a
large vortex (Fig. 5(B)). The circulating air flow in this region would prevent the air move upward into the tower, so the air flow was forced to travel a bit further towards the central part of tower and then flowed through heat exchanger. The near-wall vortex blocks outer edge of the radiator, so the cold air cannot be sucked into tower in this part, leaving this area isolated from the ambient air. However, once the wall cover was installed outside the tower, the situation would be somewhat different. The introduction of this wall guides the surrounding air horizontally flow into the bottom part of the tower and then raises upward, flowing into the tower. However, the most obvious effect caused by this wall cover is the vortex damping. The enlarged image of the velocity distribution shows the weakened vortex near the tower wall (Fig. 5(D)). Therefore, the blockage caused by this vortex would decrease accordingly, making it possible for the heat exchanger bundles to access the pre-cooled air.

The hollow-cone nozzle is widely used for humidifying purposes [37]. The mechanism of hollow-cone nozzle to produce droplets can be simply described as follows: the injected liquid exiting from the nozzle in the form of a sheet, quickly disintegrates into droplets due to the aerodynamic instability in the ‘break-up region’ and interacts strongly with the atmosphere. Just downstream in the ‘spray region’, the liquid exclusively exits in the form of droplets [38]. The hollow-cone nozzle produces the spray pattern with droplets concentrated in the outer cone edge forming an annular cross section. The resultant spray pattern of a typical hollow cone nozzle is illustrated in Fig. 6. The apparent popularity of hollow-cone nozzles is due to the fact that they produce finer droplets compared with full cone nozzles and consequently provides a larger contact surface between air and droplets since droplets are discharged at the edge of the cone [39]. In view of its excellent performance for producing fine drops to accelerate the evaporation process, a commercial hollow cone nozzle LNN1.5 was employed in this numerical study.

Since the model geometry for water spray is much smaller than that for velocity distribution calculation, a finer mesh size was adopted to obtain a good result without increasing too much computation cost. Based on the mesh independence test for the single LNN1.5 injection (Table 3), the model simulated with 2,836,500 cells achieved the satisfactory results and was used for further calculation. If the nozzle number increased, more droplets were tracked, so a preliminary calculation with 2,836,500 cells was firstly made. Then the mesh was adapted according to the preliminary calculation results. The adapted mesh was confined in the area with relative humidity in the range of 60%--100%, where droplet concentration was high and more cells were needed to get a good result. This adaptive mesh strategy allowed us to increase the cell number to a limit extent while capturing the necessary features of spray cooling.

The heat exchanger in the tower is simulated as a radiator in FLUENT. A radiator is considered to be infinitely thin, and the pressure drop through the radiator is assumed to be proportional to the dynamic head of the fluid, with an empirically determined loss coefficient [26]. The radiator model in FLUENT was used to calculate the performance of the air-cooled heat exchanger of the cooling tower. The heat transfer process and the pressure drop in the heat exchanger could be represented by the following equations:

\[ Q = h_f(T_{rd} - T_a) \]  \hspace{1cm} (18)

\[ \Delta P = L_f \frac{1}{2} \rho_a v_a^2 \]  \hspace{1cm} (19)
Here the heat transfer coefficient and pressure loss coefficient were determined by the following polynomial correlations [40]:

\[ h_r = 2480.9V_a^4 - 8623V_a^3 + 11080V_a^2 - 5957.4V_a + 2389.3 \]  

(20)

\[ L_f = 28.759V_a^2 - 80.819V_a + 78.076 \]  

(21)

### 2.2.2 Boundary and Operating Conditions

The ambient air flow through the tower was considered as an ideal air mixture containing water vapor, oxygen and nitrogen. The air consists of the dry air part with 77% of nitrogen and 23% of oxygen by mass and different concentration of water vapor depending on the humidity. Air velocity profile obtained from a separate tower simulation was used as the velocity inlet boundary condition. The inlet turbulence intensity was assumed as 1% for all cases [2]. The turbulence intensity was selected based on the research outcome of Alkhedhair et al. [2,20]. They assumed the turbulence intensity was 1% in their simulations, and conducted wind tunnel test to simulate the NDDCT, the good agreement between the simulated results and experiental ones proved the effectiveness of this assumption. Also his experimental tests showed the produced intensity for the spray at air velocity of 1m/s was around 1%, which is quite similar to our simulation conditions, hence we used the 1% turbulence intensity for our simulations. The operating pressure was assumed to be the atmospheric pressure, 101.325 kPa. At the top of the large domain, the pressure outlet boundary condition was used. The wall of tower was set as adiabatic walls with no-slip condition. The enhanced wall function was used to model the near wall regions.

Fresh water droplets were injected as the discrete phase at a constant temperature of 28 °C. The droplets are assumed to be perfect spheres and the temperature gradient within the droplets is neglected due to their small size [41]. Droplet collision and coalescence were not considered in the simulation as the spray is dilute [37]. The trajectories of droplets were tracked by grouping them into parcels. Three parcel sizes of 200, 600, 1500 droplets were trialled. The calculated mean temperature at the radiator varied as small as 0.03 °C as the number of parcels increased from 200 to 1500. Thus, 200 parcels were used to reduce computation load. In the spray cooling model, a hollow cone nozzle LNN1.5 is used. The key parameters for the nozzle and ambient air are listed in Table 4. The boundary condition for droplets impacting the no-slip walls was set as “escape”, i.e., droplets impacting the walls are terminated and excluded from further calculation. This regime is also assigned for the inlet and outlet. In the tower velocity simulation, the two cutting plane was set as symmetry boundary due to the geometric consideration and the aim to avoid introducing additional resistance. However, as to the situation of spray cooling, this symmetry condition is not appropriate, so the slip wall is assigned to the cutting planes. According to the manual of FLUENT, the symmetry condition assumes that there is no flux of any quantity across a symmetry boundary. The zero-flux across a symmetry plane means that once some droplets hit the symmetric plane, the unbalanced discrete phase flux fails to meet such requirement [26]. Therefore, a slip-wall is used to replace the symmetry condition. The shear stress caused by the wall is fixed to zero and droplets hitting the wall would be reflected back for further calculation. This particular setting of the slip wall can be reckoned as a symmetry boundary with some modification of the wall-droplet interaction.
2.2.3 Model Validation

The UQ Gatton cooling tower was tested under windless condition to validate our cooling tower model. The experiment tests were conducted on an isolated cooling tower with its own heating unit to generate hot water to provide the heat source. Fig. 7 illustrates the details of this heating system. It is composed of three parts: heater, water tank and water circulating pipelines. Diesel was used as fuel for heater to produce hot water. The total heat input was fixed at 840 kW. Two pumps were installed to drive water from water tank to heater and then to cooling tower.

The heat exchanger is consisted of 18 bundles water, each of which is equipped with two temperature sensors to measure the temperature of inlet and outlet water. The water mass flow rate for each bundle was measured by the mass flowmeter installed at the inlet of each heat exchanger bundle. The air temperature and air humidity is measured at 36 different locations across various heights of the tower. To be specific, the temperature and humidity sensors are located at four different levels: the heat exchanger inlet plane, heat exchanger outlet plane, the middle of the tower and the top of the tower. Each level has 9 temperature sensors and 9 humidity sensors. 14 pressure transducers were placed inside the tower to collect pressure change at various locations. Fig. 8 shows the arrangement of these sensors. The accuracy and measurement range of these sensors were summarized in Table 5. All the experimental data were recorded via a National Instrument CRIO real time data logging and analysis system.

Table 6 shows the seven experimental test conditions, which served as input data for numerical simulation. The comparisons between the measured and predicted values for NDDCT are shown in Fig. 9. The comparison results demonstrate the good agreement between the CFD predictions and the experimental data. The model can accurately predict the temperature of hot air after the radiator, with all an average deviation less than 5%. The predicted temperatures of cooled recirculating water flowing through the radiator have a slightly larger deviation than the predictions for hot air temperature, with only one data point having a deviation larger than 5%. However, the simulated results for air velocity inherent to the induced natural draft have two data points lie between the deviation of 5% and 10%. All other 5 points approach the test results closely. These good agreements verify the accuracy of the built model for tower simulations. It is worth noting that the simulated air velocity is slightly higher than the experimental result. The possible reason is that the small crosswinds under the field tests would pose negative effect on the heat transfer process. The presence of winds disturb air flow inside the tower, leading to the uneven distribution of the induced air flow. In the windward part of the heat exchanger, air flow decreases and becomes smaller than that in the leeward part. With the increased unequal distribution of air flow, vortices are formed in the tower, which redistribute the hot air and further impair the heat transfer. The depressed heat transfer would cause the decreasing velocity during the test. Since the crosswind is not strong, we neglect this effect in our simulation model. The negligence of this detrimental factor results in the slight overestimated air velocity from CFD calculations.

Since there is a lack of experimental data related to spray cooling in NDDCT, the model used for spray cooling cannot be directly validated. An indirect way would be used for spray cooling validation. In spray cooling study, a common practice is to validate the model with experimental data obtained from droplet evaporation test, which provides accurate and ample data for model validation. For instance, in the open literatures published by Alkhedhair [2], Tissot [42] and Sadafi [43], they all used experimental data from single droplet evaporation tests to validate their model. Therefore, in this research, the same approach was adopted to
validate our model for spray cooling simulation. According to the experimental study conducted by Sartor and Abbott [44], a single droplet falling with a zero initial velocity in the air was simulated. Numerical conditions have been set in order to match the experimental conditions: the temperature of ambient air and droplet were fixed at 295K with the pressure of 82.8 kPa and a relative humidity 98%. As is shown in Fig. 10, the droplet velocity was plotted as a function time. The excellent agreement between the simulated results and the experimental results demonstrates the ability of our model to predict water evaporation phenomenon.

2.3 Nozzle representation and cooling performance

In the design of spray cooling system, two commercially available hollow-cone nozzle LNN1.5 were employed to disintegrate bulk water into droplets. The nozzles were bought from the Spraying System Co. Ltd. and were characterized by Alkhedhair based on wind tunnel tests [20]. The injected flow rates for LNN1.5 is 5 g/s. The produced droplet size distribution for nozzle LNN1.5 is shown in Fig. 11. As an important parameter of spray, droplet size distribution considerably affects the water-air transportation and spray cooling efficiency. In practice, uniform droplet size distribution is quite difficult to obtain and the sizes of droplets usually change from a few microns to several hundred microns. It is quite difficult to describe a spray consisting of various size fractions using a single value parameter. To characterize the spray produced by the LNN1.5, a wind tunnel equipped with Phase Doppler Particle Analyzer (PDPA) was employed to get the droplet size distribution. The shape of the droplet size distribution is described by a continuous Rosin-Rammler function. This function assumes that there is an exponential relationship between the droplet size $D$, and the volume fraction of droplets with diameter greater than $D$. The equation of the Rosin-Rammler distribution is:

$$f(D) = 1 - \exp\left(\frac{D}{D_m}\right)^\alpha$$

where $f(D)$ is the fraction of the cumulative percentage of the spray with droplet diameters greater than $D$. $D_m$ and $\alpha$ are the mean diameter and spread parameter related to the distribution center and width, respectively.

The experimental results and the fitting curve are shown in Fig. 11. This figure shows a good agreement between the measured droplet data and the fitting curve predicted by Rosin–Rammler function. This consistence makes it possible to employ this function to predict droplet distribution in FLUENT package. For the nozzle LNN1.5, $D_m=63.5$ µm and $\alpha=3.14$ were used to produce widely-distributed droplets. These parameters derived from Fig. 11 indicate that LNN1.5 is capable to produce small droplets to facilitate the evaporation process.

As is illustrated in Fig. 12, the positions of employed nozzles were identified by three parameters: the nozzle height (H), radial length (R) and separation distance (Ds). The third parameter is relevant only when more nozzles than one are placed at a given value of H and R. If there is only one nozzle, it is placed at the wedge center line. If there are more, they are distributed symmetrically about the centerline with a separation distance, Ds. Based on the XYZ coordinate system denoted by the red color, the value of height ranges from 0-5 m, R changes from 0-9.2625 m and Ds has a value from 0 m to 4.8m.

The cooling effect of the spray system is characterized by the cooling efficiency, which is defined as the ratio of the actual air temperature drop to the maximum possible temperature drop. It can be formulated as:
\[ \eta_c = \frac{T_a - T_{rd}}{T_a - T_{wb}} \]  

(23)

where \( T_a \), \( T_{wb} \) are the dry-bulb temperature and wet-bulb temperature of the ambient air at the outside the cooling tower, respectively. \( T_{rd} \) is the mass-weighted average temperature at the radiator surface. Here the radiator is modelled as a very thin surface. The mass-weighted average temperature is expressed as:

\[ T = \frac{\int \rho T|\vec{v} \cdot dA|}{\rho|\vec{v} \cdot dA|} = \frac{\sum_{i=1}^{n} \rho T|\vec{v}_i \cdot A_i|}{\sum_{i=1}^{n} \rho |\vec{v}_i \cdot A_i|} \]  

(24)

where \( T \), \( \rho \) and \( \vec{v}_i \) are the mass-weighted average temperature, air density and the corresponding local velocity at the small areas denoted by \( \vec{A}_i \).

The mass-weighted average temperature can be used to characterize the cooling performance achieved by different nozzle configurations. Furthermore, the temperature drop is defined as the temperature difference between the mean (mass-averaged) temperature at the heat exchanger inlet and the ambient air temperature (\( T_a = 40^\circ C \)).

\[ \Delta T = T_a - T_{rd} \]  

(25)

Where \( T_a \) is the dry-bulb temperature of the ambient air outside the cooling tower; \( T_{rd} \) is the temperature of air at the radiator surface.

If an area at the radiator surface experiences a temperature drop larger than 0.62 \( ^\circ C \), corresponding to the cooling efficiency higher than 5%, it is denoted as part of the impact area. The impact area is used to denote the size of the radiator surface influenced by the pre-cooled air. On the basis of impact area, the spray cover ratio \( \psi \) is expressed as:

\[ \psi = \frac{\text{The size of impact area}}{\text{The size of radiator surface (10.27 m}^2\text{)}} \]  

(26)

In addition to the average temperature of the radiator and the corresponding temperature drop, the evaporation rate is another important parameter to evaluate spray cooling. The more and faster water evaporation, better cooling performance will be achieved. Hence the careful design of the NDDCT cooling system should be done to reach full evaporation of water droplets at the bottom of tower, i.e., the lower 5m inlet area. The latent heat for water evaporation is provided by the sensible heat from hot ambient air, thus the larger fraction of evaporated water, the lower the inlet air temperature will be and the better pre-cooling performance is achieved. To quantitatively compare the cooling performance in terms of the evaporated water amount, an evaporated water fraction \( \beta \) is defined as below:

\[ \beta = \frac{\text{Evaporated water flowrate}}{\text{Injected water flowrate}} \]  

(27)

A larger value for \( \beta \) corresponds to the larger flowrate of evaporated water. To avoid the corrosion problem caused by droplets evaporating on the heat exchanger surface and to minimize water waste, the system should satisfy the condition of \( \beta \geq 0.95 \) so that the majority of water would evaporate into vapor.
3 Results and Discussions

3.1 Inlet Air Velocity

Fig. 13(a) shows the temperature distribution at the vertical cross section of cooling tower. The raised air temperature is caused by the heat transfer from the hot water inside the tube to the outside air. As is shown by the streamline (the black solid line) in Fig. 13(b), the ambient atmosphere, driven by the buoyancy force originating from the density difference between the outside and inside of the tower, flows into the tower and through the radiator. The reverse pressure gradient is conspicuously observed inside the tower to balance the buoyancy force and viscous force. The velocity vector distribution is shown in Fig. 13(c).

Water spray modelling involves complex heat and mass transfer computations and requires large computational resources. To address this problem, we did not use the model in Fig. 4(a) for spray simulation. Instead, we used a smaller model (Fig. 4(c)), consisting of an isolated tower and spray system, for spray nozzle investigations. In this smaller geometry, the heat exchanger was turned off, excluding the complex coupling between heat exchanger and evaporating droplets. Therefore, the limited computational resources can be used for the water sprays simulations with varied nozzle arrangements. Once the radiator model was deactivated, the large air domain required for the buoyance-driven air flow calculation was unnecessary. Hence, a smaller tower model (Fig. 4(c)) allowed us to concentrate on the detailed information of spray cooling. However, being deprived of the heat exchange with the radiator, the small cooling tower could not produce any air flux. To address this problem, a velocity-inlet boundary condition was used to introduce some air flows for the isolated tower. The velocity distribution (Fig. 13(c)) obtained from the whole cooling tower simulation was employed as the input velocity profile for the isolated spray cooling assisted tower. In water spray calculation, air flows could freely pass through the heat exchanger surface because the heat exchanger was modelled as an interior rather than a radiator.

To test the effectiveness of above two-step strategy, we firstly checked whether the air flows modelled in the large (Fig. 4(a)) and small (Fig. 4(c)) domains are identical. To reach this end, the comparisons of air velocity distribution based on the whole tower simulation results and the interpolated data used for spray cooling were made. As is indicated by 9(c), two locations were selected for velocity comparisons. The first one was the lateral tower inlet surface (nozzle-containing surface at radius of 6m) and the second one was the horizontally placed radiator surface. The velocity magnitudes ($\sqrt{V_x^2 + V_y^2 + V_z^2}$) at both locations were compared first. From Fig. 13 (d) and (e), we can draw the conclusion that there exists a consistent velocity distribution at these two critical locations. From Fig. 13(c), we can see the upward movement dominates the air flowing through the radiator, hence the velocity magnitude mainly depends on $V_y$, and so we did not make a detailed comparison in terms of decomposed velocity. However, for the tower inlet part, in addition to the comparison of velocity magnitude, the decomposed velocities in X, Y and Z directions were also compared for they have a great influence on droplet movements. Fig. 14 shows the result comparisons for $V_x$, $V_y$ and $V_z$, respectively. The interpolated velocity components coincide with their corresponding counterparts based on whole tower simulation. The consistency between two sets of data illustrates the effectiveness of the adopted two-step modelling.

3.2 Nozzle distance investigation

When a system of several spray nozzles is designed, an inevitable question is how to determine the distance between two nozzles. To answer this question, a preliminary study
was made. In this study, two nozzles were placed at the same horizontal and vertical plane, i.e., they shared the same vertical height $H$ and same radius $R$. In addition to shared vertical height and radius, both nozzles injected in the positive $Z$ direction. The locations of the two LNN1.5 were listed in Table 7.

The temperature distribution at the heat exchanger surface and the vertical cross-section plane were displayed in Fig. 15. The temperature profiles for the heat exchanger surface show a perfectly symmetric distribution for all the separation lengths. This symmetry comes from the symmetric arrangement of two LNN1.5 leading to the expected symmetrical temperature distribution at the radiator surface. However, the most important conclusion we can get from Fig. 15 is that as the separation distance between two nozzles increases, the impacted regions by the cooling air, as is denoted by the green and yellow color, tend to separate gradually. For the cases with $Ds=0.4m$ and $1m$, the impacted regions display a roughly circular pattern, indicating strong overlapping of the sprays produced by two nozzles. But as the value of $Ds$ rises to $1.6m$ and $2.4m$, the two sprays have less interaction, the overlap is somewhat reduced and the separation is clearly seen. At a separation distance of $3m$ and higher, ($Ds=3m$ and $3.6m$), the two LNN1.5 barely influence each other with fully separated impact areas.

This qualitative analysis still fails to give us information about the optimal separation distance between two LNN1.5. Thus, a quantitative comparison ought to be made. Fig. 16(a) shows the mass-weighted average temperatures at the radiator surface and the corresponding temperature drops relative to the surrounding air. The comparison shows an interesting trend. When the separation distance between two LNN1.5 increases from $0.4m$ to $1m$, the temperature drop at the radiator surface grows from $2.6 ^\circ C$ to $2.9 ^\circ C$, indicating an enhanced pre-cooling effect. While as these two nozzles were separated further from each other, the deteriorated cooling effect was observed, as was illustrated by the decreasing temperature drop. Since the temperature drop was caused by the evaporative water, a larger temperature drop usually corresponded to more evaporated water. This consistency was proved by the Fig. 16(b). That figures shows that a peak exists at the separation distance of $1m$, a smaller or larger value would pose some negative effects on the evaporation of water. For the optimal case with $Ds=1m$, 98.7% (9.87g/s) of injected water (10g/s) became evaporated, while for the injection of larger $Ds$ ($1.6m$), 98% (9.8g/s) of injected water evaporated. In spite of the different separation distances, these two cases achieved almost the same cooling effect. The minor differences in terms of cooling effect produced by these two cases give us the flexibility to arrange nozzles. It should also be noted that, at separation distance above $1.6m$, a significant fraction of the unevaporated droplets escaped from the boundaries and were excluded from cooling calculation. Due to the escaping of these drops, the potential cooling correlated with these unevaporated droplets, were lost and thus lead to the deteriorated cooling results. Therefore, the separation distance between two LNN1.5 should be carefully chosen to avoid the deteriorated cooling effect.

3.3 Multi-nozzle arrangements

The investigations on the arrangements of two LNN1.5 show that the proper distances between two nozzles along $X$-axis should be in the range of $1m$-$1.6m$. This is an important and useful conclusion that enables the design of a spray cooling system consisting of several nozzles. In the multi-nozzle spray cooling system, the configurations of nozzles were based on the previous case. We started from the one-nozzle situation, and then increased the nozzle number to two, three, four and five to analyze the produced cooling effect. The positions of each nozzle in different cases were summarized in Table 8 and the cooling effect was illustrated in Fig. 17. All the explored nozzle had positive $Z$-axis directed injection.
By combining the nozzle position (Table 8) with its caused cooling effect (Fig. 17), we can make a useful analysis. For the situation of single nozzle (Fig. 17(N1)), the nozzle LNN1.5 was placed at the middle section plane of the geometry with a counter flow injection. The pre-cooled region was constrained in a small circular part of plane, leaving the majority of the heat exchanger unaffected by the pre-cooled air. For the two-nozzle case (Fig. 17(N2)), two LNN1.5 were arranged symmetrically about the middle plane with a separation distance of 1.6m. It is obvious that the cooling air influence the outside part of the radiator, an impacted area much larger than that of one-nozzle case. The three-nozzle configuration (Fig. 17(N3)) had one nozzle at the middle plane while the other two were symmetrically arranged with Ds=3m. An enhanced cooling effect was achieved, as is evidenced by the dominance of the low-temperature profile (green color). When the nozzle number became four (Fig. 17(N4)), the nozzles were arranged at two different heights. Two nozzles were grouped together and symmetrically put at a lower horizontal plane (H=3m) with a smaller separation distance (Ds=2.4m), giving droplets longer residence time to evaporate. Another group was placed at a higher horizontal plane of H= 4.6m, but the two nozzles had larger separation (Ds=3m) to reduce the overlapping of these two sprays. The temperature contour shows that the majority of the radiator surface was influenced by the cooling air. The stratified temperature distribution is closely related to the cooling effect at different degrees. The central part of the tower is not cooled as much as the outside part of the radiator, which would impair the overall performance of the radiator. Therefore, in order to achieve relatively uniform temperature distribution at the radiator surface, a system of five LNN1.5 was investigated. The five nozzles were divided into three groups. One group had a single nozzle placed at the middle part of the geometry with a height of 4.6 m. For the second group, two LNN1.5 were placed at the horizontal plane at 3m with a 2.4m separation and the radial length R=7.5m. The third group had two nozzles located higher (H=4.6m) with larger separation (Ds=3m) and further away from the tower center (R=8.5m). This nozzle arrangement, to some extent, was designed to reduce the spray overlapping caused by the increased nozzle number. The corresponding cooling effects are seen in Fig. 17(N5), where both the outside and central part of cooling tower are better cooled, having lower temperatures. With all the other four nozzles having the same configuration as that of case N4, an additional LNN1.5 was placed near the tower center (R=8.5m), at a lower height (H=4m), and have a counter-flow injection. This configuration helps to cool the air in the central part of tower, thus the five nozzle employment reduces the uneven distribution of temperature at the radiator surface, improving the cooling performance. The arrangement of these nozzles are illustrated in Fig. 18. As is expected, the central part of tower is better cooled, having more regions dominated by low temperatures. The relatively uniform temperature distribution is achieved, as is illustrated by Fig. 17. Almost the whole surface of the radiator is accessed by the pre-cooled air, thus all the heat exchanger bundles at this surface would experience an enhanced heat and momentum transfer.

The increasing cooling effect connected with the increment of nozzle number was better proved by Fig. 19. Fig. 19(a) shows the change of temperature drop at the radiator surface and cooling efficiency in terms of the nozzle number. The positive relationship between the cooling efficiency and the nozzle number can be seen. The increasing number of nozzles leads to higher cooling efficiency, which corresponds to larger temperature drop. The improved cooling effect caused by increased numbers is most obviously observed from the case with one nozzles to the case with five nozzles. Continue to increase nozzle number, at one side, can increase the cooling performance, but on the other hand, would simultaneously be associated with larger water consumption. The improved cooling effect for multi-nozzle cases is attributed to the large water flowrate and thus the more evaporated water amount.
The detailed information about the evaporated water flowrate were summarized in Fig. 19(b). Naturally, a spray system composed of more nozzles has larger water flowrate but a corresponding evaporated water flowrate is not guaranteed. Fortunately, the chart indicates that the evaporated water flowrate increases as more and more nozzles are employed. This increment is connected with the fact that the evaporated water fractions ($\beta$) for different cases change in a small range. The largest value (98.6%) of $\beta$ is achieved for the case of one nozzle (N1) while the smallest one (96.7%) occurs in the situation composed of three nozzles (N3). The value of $\beta$ for all the five cases (N1-N5) changes from 96% to 99%. This relative large value of $\beta$ mean that almost all the injected water evaporates into water vapor, absorbing substantial amount of heat from the surrounding hot air. The latent heat of water evaporation is provided by the sensible heat of the ambient air, therefore, the more nozzle employed, the more water would evaporate, so the lower ambient temperature would be. This low air temperature characterizes the better cooling effect.

Since more water was introduced by the spray cooling system as the nozzle number increased, the ratio between evaporated water flowrate and air flowrate grew ($\frac{m_e}{m_a}$) as well. As is shown in Fig. 20, the cooling efficiency has a positive correlation with $\frac{m_e}{m_a}$, which illustrates the enhanced cooling effect as more spray nozzles are used. The spray cover ratio is also determined by the ratio of evaporated water flowrate and air flowrate. As the value of $\frac{m_e}{m_a}$ rises, the spray cover ratio shows a remarkable increase. Finally, as five nozzle were used, the $\frac{m_e}{m_a}$=2.65, all the radiator surface were covered by the pre-cooled air ($\psi$=1). Since our goal is to achieve at least 50% cooling efficiency, so the spray cooling system composed of five nozzles was selected for the further explorations.

To make a more general conclusion that is useful for other tower geometry, we made a dimensionless analysis based on aforementioned results. Three nondimensional parameters are taken into consideration: evaporated water mass flowrate to air mass flowrate ($\frac{m_e}{m_a}$), the ratio between wet bulb temperature and ambient temperature ($\frac{T_{wb}}{T_a}$) and nozzle separation distance divided by tower radius ($\frac{d}{R}$). These three dimensionless numbers account for both the water-air heat and mass transfer, ambient air influence as well as nozzle arrangement configuration effect. The derived formula is shown as below:

$$\eta_c = 0.052 + 0.6215 \left( \frac{m_e}{m_a} \right)^{0.619} \left( \frac{T_{wb}}{T_a} \right)^{1.352} \left( \frac{d}{R} \right)^{0.623}$$

This correlation has the similar structure as the one put forward by Kaiser et al., which has a small discrepancy lower than 5% when compared with experimental results [45]. The differences between the result predicted by equation 28 and the CFD results are quite small, and the achieved consistency is illustrated in Fig. 21. The figure shows the results predicted by correlation have small deviation from the CFD simulated ones. Thus the correlation can serve as a practical tool for designers to improve the cooling efficiency.

### 4. Conclusions

We designed a spray cooling system to improve the poor cooling performance of natural draft dry cooling tower under hot ambient conditions. The introduction of a small amount of water to precool the inlet hot air helps to improve the performance of NDDCT and thus increase the overall efficiency for the whole power plant. Commercial available nozzles LNN1.5 were characterized experimentally and employed in this spray cooling system. Two important factors were considered when designing the spray cooling system. The first one is to ensure
that spray nozzles were carefully arranged to make sure the injected water evaporate as much as possible before it reached the radiator. This would prevent the corrosion problem related to the unevaporated drops. Secondly, the pre-cooled inlet air should be evenly distributed at the radiator surface. Considering that radiator is composed of a number of heat exchanger bundles, the spray cooling system should be designed to ensure that each bundle is accessible to the pre-cooled air. With this careful design, an enhanced heat exchange between radiator and ambient air would be achieved. With these two goals, nozzle arrangements needs extensive exploration. The main conclusions are as follows:

(1) An optimal distance between two LNN1.5 placed at the same horizontal plane is identified. If two nozzles are too close (Ds=0.4m), little space is available for injected water to reach full evaporation and the correspondent impact area is restricted at the central part of the radiator. As the separation distance increases to 1m, the impact area expands gradually and more water become evaporated. However, further increasing this distance would be detrimental to water evaporation. Therefore, the proper distance is found to be in the range of 1-1.6m.

(2) Increasing the number of nozzles will increase $\frac{m_w}{m_a}$. Meanwhile the cooling efficiency also increases, enhancing cooling performance of NDDCT. The rising $\frac{m_w}{m_a}$ leads to larger spray cover ratio, indicating more and more radiator sections are accessible to the pre-cooled air. When five nozzles were employed, the spray cover ratio reached the maximum value ($\psi=1$). As more nozzle LNN1.5 are used, the impact area of pre-cooled air grows accordingly.

(3) For the five-nozzle case, the largest temperature drop (6.3 °C) was obtained with a cooling efficiency of 51.2%. Dimensionless analysis was conducted to correlate cooling efficiency with influencing factors. It is found that cooling efficiency can be determined by the ratio of evaporated water mass flowrate to air mass flowrate, wet bulb temperature to ambient temperature and nozzle separation distance to tower radius. The derived formula shows that the efficiency is influenced by the water-air heat and mass transfer, ambient air conditions as well as nozzle arrangement configurations.

Acknowledgement

This research was performed as part of the Australian Solar Thermal Research Initiative (ASTRI), a project supported by the Australian Government, through the Australian Renewable Energy Agency (ARENA).

Nomenclature

- $\overline{A_t}$ Small areas at the radiator surface
- $C_D$ Drag coefficient
- $C_{pa}$ Specific heat of air (J/kg·K)
- $C_{pw}$ Specific heat of water (J/kg·K)
- $D_d$ Droplet diameter (µm)
- $D_f$ Diffusion coefficient (m²/s)
- $D_{32}$ Sauter mean diameter (µm)
- $D_m$ Rosin-Rammler mean droplet diameter (µm)
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>D_{v90}</td>
<td>90% of water volume made up of droplets of this size and smaller (µm)</td>
</tr>
<tr>
<td>D_s</td>
<td>Separation horizontal distance between nozzles at the same plane</td>
</tr>
<tr>
<td>g</td>
<td>Gravitational acceleration (m/s²)</td>
</tr>
<tr>
<td>E</td>
<td>Total energy (J)</td>
</tr>
<tr>
<td>F</td>
<td>Forces acting on droplet (N)</td>
</tr>
<tr>
<td>F_d</td>
<td>Drag force (N)</td>
</tr>
<tr>
<td>F_G</td>
<td>Gravity force (N)</td>
</tr>
<tr>
<td>f(D)</td>
<td>Rosin-Rammler droplet size distribution function</td>
</tr>
<tr>
<td>G_k</td>
<td>Production of turbulent kinetic energy</td>
</tr>
<tr>
<td>h_c</td>
<td>Heat transfer coefficient (W/m²/K)</td>
</tr>
<tr>
<td>h_d</td>
<td>Mass transfer coefficient (m/s)</td>
</tr>
<tr>
<td>h_f_g</td>
<td>Latent heat of water vaporization (J/kg)</td>
</tr>
<tr>
<td>h_r</td>
<td>Heat transfer coefficient for radiator</td>
</tr>
<tr>
<td>k</td>
<td>Thermal conductivity (W/(m·K))</td>
</tr>
<tr>
<td>L_f</td>
<td>Loss coefficient</td>
</tr>
<tr>
<td>L_c</td>
<td>Characteristic length (m)</td>
</tr>
<tr>
<td>\dot{m}_a</td>
<td>Air flow rate (kg/s)</td>
</tr>
<tr>
<td>\dot{m}_e</td>
<td>Evaporative mass flux (kg/s)</td>
</tr>
<tr>
<td>\dot{m}_w</td>
<td>Water flow rate (kg/s)</td>
</tr>
<tr>
<td>m_d</td>
<td>Droplet mass (kg)</td>
</tr>
<tr>
<td>Nu</td>
<td>Nusselt number</td>
</tr>
<tr>
<td>P_r</td>
<td>Prandtl number</td>
</tr>
<tr>
<td>P</td>
<td>Pressure (Pa)</td>
</tr>
<tr>
<td>Q</td>
<td>Heat transfer rate for radiator (W)</td>
</tr>
<tr>
<td>R</td>
<td>Tower radius</td>
</tr>
<tr>
<td>R_{ed}</td>
<td>Droplet Reynolds number</td>
</tr>
<tr>
<td>S_c</td>
<td>Schmidt number</td>
</tr>
<tr>
<td>S_{ct}</td>
<td>Turbulent Schmidt number</td>
</tr>
<tr>
<td>S_e</td>
<td>Source term of energy (W/m³)</td>
</tr>
<tr>
<td>S_m</td>
<td>Source term of mass (Kg/m³)</td>
</tr>
<tr>
<td>S_{mo}</td>
<td>Source term of momentum (Kg/m²·s²)</td>
</tr>
<tr>
<td>Sh</td>
<td>Sherwood number</td>
</tr>
<tr>
<td>T</td>
<td>Temperature (°C)</td>
</tr>
<tr>
<td>V_a</td>
<td>Air velocity (m/s)</td>
</tr>
<tr>
<td>V_d</td>
<td>Droplet velocity (m/s)</td>
</tr>
<tr>
<td>V_{cell}</td>
<td>Computational cell volume (m³)</td>
</tr>
<tr>
<td>V_r</td>
<td>Droplet relative velocity (m/s)</td>
</tr>
<tr>
<td>V_w</td>
<td>Droplet volume (m³)</td>
</tr>
<tr>
<td>w</td>
<td>Humidity ratio (kg/kg of dry air)</td>
</tr>
<tr>
<td>X_d</td>
<td>Droplet position (m)</td>
</tr>
<tr>
<td>Y_j</td>
<td>Mass fraction of specie j</td>
</tr>
<tr>
<td>\Delta P</td>
<td>Pressure drop</td>
</tr>
</tbody>
</table>

**Greek symbols**
α  Spread parameter
β  Evaporated water fraction
ρ  Density (kg/m³)
ε  Turbulent dissipation rate (m²/s³)
δ_ijkl  Mean strain tensor (1/s)
τ_ijkl  Mean stress tensor (Kg/m² s)
μ  Dynamic viscosity of air (kg/m s)
μ  Turbulent dynamic viscosity (kg/m s)
Φ  Viscous dissipation (W/m³)
τ_c  Droplet relaxation time (s)
η_c  Cooling efficiency
ψ  Spray cover percentage

Subscripts
a  Air
d  Droplet
l  Local value
w  Water
v  Vapor
sat  Saturation
e  Evaporation
t  Time
int  Droplet-air interface
i,j,k  Cartesian coordinate Directions
wb  Wet-bulb
rd  Radiator

Abbreviations
NDDCT  Natural draft dry cooling tower
CFD  Computational fluid dynamics
NDDCT  Natural draft cooling tower
UQ  University of Queensland
PDPA  Phase Doppler Particle Analyzer

References


6.


Fig. 1 The experimental tower built at UQ and the specifications used for simulation (a and b). A schematic diagram of inlet air pre-cooling for NDDCT (c).
Fig. 2 Coupled calculation between continuous and discrete phase calculations flowchart
Fig. 3 Forces acting on the droplet

Fig. 4 The dimensions of geometric model and boundary conditions utilized for air velocity distribution calculation (a) and for water spray calculation (c). The mesh generated at the vertical middle cross plane of the cooling tower for air velocity distribution (b) and for spray calculation (d).
Fig. 5 Velocity distribution of the vertically middle plane for the cooling tower without wall cover (A), and with wall cover (C). The enlarged velocity field (inside the blue rectangle) for the tower without cover wall (B) and with tower wall (D).
Fig. 6 Hollow-cone spray pattern

Fig. 7 Hot water control system
Fig. 8 Test sensors distribution

Fig. 9 Comparisons of CFD predictions and experimental test data for (a) the temperature of hot air heated by the radiator, (b) the temperature of cool water exiting from the radiator, and (c) the velocity of induced draft across the radiator.
Fig. 10 Predictions of evaporation of three free-falling droplets. The diameters of these three droplets are 67.92 µm, 101.14 µm and 157.26 µm, respectively. The comparisons are based on our numerical simulations and the experimental measurements conducted by Sartor and Abbott [47].

Fig. 11 The diameter distribution and Rosin–Rammler distribution fitting for LNN1.5.
Fig. 12 The nozzle arrangement at the inlet area of NDDCT. H represents the height of nozzle location (H= 0-5m), R is the radial distance between nozzle location and the tower center. Ds is the distance between two nozzles in the X direction.
Fig. 13 The temperature contour of vertical middle cross section of 30-degree NDDCT (a); the air streamline and gauge pressure distribution of vertical middle cross section of tower (b); velocity vector distribution of the vertical middle cross section of NDDCT (c); the consistency of the velocity across the radiator between the calculated results from tower simulation and the interpolated results for spray cooling modelling (d); the green square denotes the results calculated by whole tower simulation, and the red asterisk denotes the results obtained from the interpolated velocity profile used for spray simulation. The consistency of the velocity at the tower inlet part between the calculated results from tower simulation and the interpolated results for spray cooling modelling (e).
Fig. 14 The consistent distributions of velocity components at tower inlet part. (a), (b) and (c) show the velocity components $V_x$, $V_y$ and $V_z$, respectively. The green square denotes the results calculated by whole tower simulation, and the red asterisk denotes the results obtained from the interpolated velocity profile used for spray simulation. The magnitude of the total velocity is shown in Fig. 9(e).
Fig. 15 Temperature distributions for injections generated by two LNN1.5 with different separation distances (Ds=0.4m, 1m, 1.6m, 2.4m, 3m and 3.6m). The top figures show the temperature profiles at heat exchanger surface and the bottom figures show the temperature profile of vertically cut plane aligned with the nozzle of positive X position. Both nozzles were placed at the height of 4.6m and the radius of 8.5m, sharing the positive Z-axis injection direction. The plane with teal color represents the middle section plane for the whole geometry.

Fig. 16 (a) The mass-weighted average temperatures at the surface of heat exchanger and the corresponding temperature drops relative to the ambient air for two LNN1.5 injections with various separation distances. (b) The evaporated water flowrates produced by two LNN1.5 with various separation distances and the corresponding evaporated water fractions.
Fig. 17 Temperature distributions generated by different spray cooling systems consisted of multi-nozzles (N1: one LNN1.5; N2: two LNN1.5; N3: three LNN1.5; N4: four LNN1.5; N5: five LNN1.5). The top figures show the temperature profiles at the surface of heat exchanger. The bottom figures show the temperature profiles at the vertically cut plane aligned with nozzles arranged at varied X positions. The transparent plane is the middle cross-section plane of the geometry, helping to identify the relative locations of the other planes with temperature distribution.
Fig. 18 The arrangement of spray nozzles for the case N5; (a) is the overview of the nozzle arrangement; (b) is the front view (in X direction); (c) is the top view (in Y direction).

Fig. 19 (a) The temperature drops relative to the ambient air at the surface of heat and the cooling efficiency for spray cooling system consisted of multi-nozzles. (b) The evaporated water flowrates and the corresponding evaporated water fractions for spray cooling system consisted of multi-nozzles.
Fig. 20 The positive influences of flowrate ratio ($\frac{m_e}{m_a}$) on the cooling efficiency and spray cover ratio. The flowrate ratio is calculated using the evaporated water flowrate divided by the air flow.

![Cooling efficiency comparison by the CFD simulation and correlation prediction.](image)

Fig. 21 Cooling efficiency comparison by the CFD simulation and correlation prediction.
## Tables

**Table 1** Morsi and Alexander drag coefficient correlation constants

<table>
<thead>
<tr>
<th>( R_{ed} )</th>
<th>( a_1 )</th>
<th>( a_2 )</th>
<th>( a_3 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>( R_{ed} &lt; 0.1 )</td>
<td>0</td>
<td>24</td>
<td>0</td>
</tr>
<tr>
<td>( 0.1 &lt; R_{ed} &lt; 1 )</td>
<td>3.69</td>
<td>22.73</td>
<td>0.0903</td>
</tr>
<tr>
<td>( 1 &lt; R_{ed} &lt; 10 )</td>
<td>1.222</td>
<td>29.167</td>
<td>-3.8889</td>
</tr>
<tr>
<td>( 10 &lt; R_{ed} &lt; 100 )</td>
<td>0.6167</td>
<td>46.5</td>
<td>-116.67</td>
</tr>
<tr>
<td>( 100 &lt; R_{ed} &lt; 1000 )</td>
<td>0.3644</td>
<td>98.33</td>
<td>-2778</td>
</tr>
<tr>
<td>( 1000 &lt; R_{ed} &lt; 5000 )</td>
<td>0.357</td>
<td>148.62</td>
<td>-4.75e4</td>
</tr>
<tr>
<td>( 5000 &lt; R_{ed} &lt; 10000 )</td>
<td>0.46</td>
<td>-490.546</td>
<td>57.87e4</td>
</tr>
<tr>
<td>( 10000 &lt; R_{ed} &lt; 50000 )</td>
<td>0.5191</td>
<td>-1662.5</td>
<td>5.4167e4</td>
</tr>
</tbody>
</table>

**Table 2** Grid independence test for velocity of NDDCT

<table>
<thead>
<tr>
<th>Cell number</th>
<th>Vertical air velocity (m/s)</th>
<th>Air temperature (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>512,000</td>
<td>0.808</td>
<td>327.13</td>
</tr>
<tr>
<td>2,239,000</td>
<td>0.792</td>
<td>326.18</td>
</tr>
<tr>
<td>3,518,000</td>
<td>0.785</td>
<td>326.12</td>
</tr>
</tbody>
</table>
**Table 3** Grid independence test for spray cooling

<table>
<thead>
<tr>
<th>Cell number</th>
<th>Air velocity</th>
<th>Temperature(°C)</th>
<th>Evaporated water</th>
</tr>
</thead>
<tbody>
<tr>
<td>1,475,200</td>
<td>0.821</td>
<td>38.95</td>
<td>5</td>
</tr>
<tr>
<td>2,836,500</td>
<td>0.789</td>
<td>38.82</td>
<td>4.9</td>
</tr>
<tr>
<td>3,675,200</td>
<td>0.786</td>
<td>38.78</td>
<td>4.86</td>
</tr>
</tbody>
</table>

1: The velocity is the area-weighted vertical velocity at the heat exchanger surface. The unit is m/s.

**Table 4** Operating conditions of the air and the water droplets

<table>
<thead>
<tr>
<th>Continuous phase (Air)</th>
<th>Discrete phase from LNN1.5</th>
<th>(Droplet)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Vertical velocity: 0.8 m/s</td>
<td>Flow rate: 5 g/s</td>
<td></td>
</tr>
<tr>
<td>Dry-bulb temperature: 40°C</td>
<td>Temperature: 28°C</td>
<td></td>
</tr>
<tr>
<td>Wet-bulb temperature: 27.7°C</td>
<td>Velocity: 22 m/s</td>
<td></td>
</tr>
<tr>
<td>Relative humidity: 40%</td>
<td>Cone angle: 39°</td>
<td></td>
</tr>
<tr>
<td></td>
<td>D&lt;sub&gt;32&lt;/sub&gt;: 55 µm</td>
<td></td>
</tr>
<tr>
<td></td>
<td>D&lt;sub&gt;90&lt;/sub&gt;: 85 µm</td>
<td></td>
</tr>
</tbody>
</table>
Table 5 The measurement instruments used for experimental tests

<table>
<thead>
<tr>
<th>Sensors/instruments</th>
<th>Supplier</th>
<th>Measuring range</th>
<th>Uncertainty/accuracy</th>
<th>Quantities of the sensor</th>
</tr>
</thead>
<tbody>
<tr>
<td>Air temperature</td>
<td>Thermistor</td>
<td>0 ~150°C</td>
<td>±0.2°C</td>
<td>36</td>
</tr>
<tr>
<td>Air humidity</td>
<td>Vaisala</td>
<td>0 ~ 100% RH</td>
<td>±3% ~ ±5%</td>
<td>36</td>
</tr>
<tr>
<td>Water temperature</td>
<td>TC Direct</td>
<td>0~90°C</td>
<td>0.5°C</td>
<td>38</td>
</tr>
<tr>
<td>Water pressure</td>
<td>Thermo Fisher</td>
<td>0~100 kPa</td>
<td>0.2% FS</td>
<td>14</td>
</tr>
<tr>
<td>Water mass flow</td>
<td>Krohne</td>
<td>0~20 kg/s</td>
<td>0.50%</td>
<td>1</td>
</tr>
<tr>
<td>Crosswind velocity</td>
<td>Vaisala</td>
<td>0~60 m/s</td>
<td>±3%</td>
<td>2</td>
</tr>
<tr>
<td>Wind direction</td>
<td>Vaisala</td>
<td>-</td>
<td>±3%</td>
<td>2</td>
</tr>
</tbody>
</table>

Table 6 Test conditions used for data input for model validation

<table>
<thead>
<tr>
<th>Ambient hot air temperature (°C)</th>
<th>Inlet hot water (°C)</th>
<th>Heat load: Q (kW)</th>
</tr>
</thead>
<tbody>
<tr>
<td>11.58</td>
<td>40.95</td>
<td>840</td>
</tr>
<tr>
<td>13.67</td>
<td>43.41</td>
<td>840</td>
</tr>
<tr>
<td>18.2</td>
<td>48.34</td>
<td>840</td>
</tr>
<tr>
<td>21.37</td>
<td>51.33</td>
<td>840</td>
</tr>
<tr>
<td>24.97</td>
<td>54.02</td>
<td>840</td>
</tr>
<tr>
<td>26.48</td>
<td>55.28</td>
<td>840</td>
</tr>
<tr>
<td>27.94</td>
<td>57.16</td>
<td>840</td>
</tr>
</tbody>
</table>
Table 7 The locations of two LNN1.5 with the Z-axis injection.

<table>
<thead>
<tr>
<th>Case</th>
<th>Horizontal position (X coordinate)/m</th>
<th>Height (Y coordinate)/m</th>
<th>Radius (Z coordinate)/m</th>
<th>Distance/m</th>
</tr>
</thead>
<tbody>
<tr>
<td>N2-c1</td>
<td>±0.2</td>
<td>4.6</td>
<td>8.5</td>
<td>0.4</td>
</tr>
<tr>
<td>N2-c2</td>
<td>±0.5</td>
<td>4.6</td>
<td>8.5</td>
<td>1.0</td>
</tr>
<tr>
<td>N2-c3</td>
<td>±0.8</td>
<td>4.6</td>
<td>8.5</td>
<td>1.6</td>
</tr>
<tr>
<td>N2-c4</td>
<td>±1.2</td>
<td>4.6</td>
<td>8.5</td>
<td>2.4</td>
</tr>
<tr>
<td>N2-c5</td>
<td>±1.5</td>
<td>4.6</td>
<td>8.5</td>
<td>3.0</td>
</tr>
<tr>
<td>N2-c6</td>
<td>±1.8</td>
<td>4.6</td>
<td>8.5</td>
<td>3.6</td>
</tr>
</tbody>
</table>
Table 8 The nozzle arrangements for multi-nozzle spray cooling system. The orange color highlights the positions of nozzles placed at the middle of the geometry and had no symmetric counterpart.

<table>
<thead>
<tr>
<th>Case</th>
<th>Nozzle ID</th>
<th>Height (Y coordinate)/m</th>
<th>Horizontal position (X coordinate)/m</th>
<th>Radius (Z coordinate)/m</th>
<th>Injection direction</th>
<th>Nozzle type</th>
</tr>
</thead>
<tbody>
<tr>
<td>N1</td>
<td>N1</td>
<td>4.6</td>
<td>0</td>
<td>8.5</td>
<td>Z</td>
<td>LNN1.5</td>
</tr>
<tr>
<td>N2</td>
<td>N2-1</td>
<td>4.6</td>
<td>0.8</td>
<td>8.5</td>
<td>Z</td>
<td>LNN1.5</td>
</tr>
<tr>
<td></td>
<td>N2-2</td>
<td>4.6</td>
<td>-0.8</td>
<td>8.5</td>
<td>Z</td>
<td>LNN1.5</td>
</tr>
<tr>
<td>N3</td>
<td>N3-1</td>
<td>4</td>
<td>0</td>
<td>8.5</td>
<td>Z</td>
<td>LNN1.5</td>
</tr>
<tr>
<td></td>
<td>N3-2</td>
<td>4.6</td>
<td>1.5</td>
<td>8.5</td>
<td>Z</td>
<td>LNN1.5</td>
</tr>
<tr>
<td></td>
<td>N3-3</td>
<td>4.6</td>
<td>-1.5</td>
<td>8.5</td>
<td>Z</td>
<td>LNN1.5</td>
</tr>
<tr>
<td>N4</td>
<td>N4-1</td>
<td>3</td>
<td>1.2</td>
<td>7.5</td>
<td>Z</td>
<td>LNN1.5</td>
</tr>
<tr>
<td></td>
<td>N4-2</td>
<td>3</td>
<td>-1.2</td>
<td>7.5</td>
<td>Z</td>
<td>LNN1.5</td>
</tr>
<tr>
<td></td>
<td>N4-3</td>
<td>4.6</td>
<td>1.5</td>
<td>8.5</td>
<td>Z</td>
<td>LNN1.5</td>
</tr>
<tr>
<td></td>
<td>N4-4</td>
<td>4.6</td>
<td>-1.5</td>
<td>8.5</td>
<td>Z</td>
<td>LNN1.5</td>
</tr>
<tr>
<td>N5</td>
<td>N5-1</td>
<td>3</td>
<td>1.2</td>
<td>7.5</td>
<td>Z</td>
<td>LNN1.5</td>
</tr>
<tr>
<td></td>
<td>N5-2</td>
<td>3</td>
<td>-1.2</td>
<td>7.5</td>
<td>Z</td>
<td>LNN1.5</td>
</tr>
<tr>
<td></td>
<td>N5-3</td>
<td>4</td>
<td>0</td>
<td>8.5</td>
<td>Z</td>
<td>LNN1.5</td>
</tr>
<tr>
<td></td>
<td>N5-4</td>
<td>4.6</td>
<td>1.5</td>
<td>8.5</td>
<td>Z</td>
<td>LNN1.5</td>
</tr>
<tr>
<td></td>
<td>N5-5</td>
<td>4.6</td>
<td>-1.5</td>
<td>8.5</td>
<td>Z</td>
<td>LNN1.5</td>
</tr>
</tbody>
</table>
Highlights

- Optimal distance between two nozzles at the same plane is in the range of 1-1.6m.
- The increment of $\frac{m_e}{m_a}$ can improve the cooling efficiency and spray cover ratio.
- Hot air is cooled by 6.3 °C using 5 nozzles, achieving cooling efficiency of 51.2%.
- A correlation between cooling efficiency with dimensionless variables is presented.