Applied Thermal Engineering 131 (2018) 522–530

Contents lists available at ScienceDirect

Applied Thermal Engineering

journal homepage: www.elsevier.com/locate/apthermeng

Research Paper

A quantitative approach identifies the critical flow characteristics in a natural draft dry cooling tower

Weiliang Wang *, Junfu Lv, Hai Zhang, Qing Liu, Guangxi Yue, Weidou Ni

Key Laboratory for Thermal Science and Power Engineering of Ministry of Education, Department of Thermal Engineering, Tsinghua University, Beijing 100084, China

HIGHLIGHTS

- A half-cylindrical NDDCT CFD model is built and validated by a hot state test rig.
- FLF is derived and verified to quantitatively describe flow characteristics effect.
- Rear side flow separation area and main stream vortices are critical to ventilation.
- The contributions of each flow field region on ventilation degradation is exhibited.
- Flow convergence are transformed to be vortices barrier under high crosswind.

ARTICLE INFO

Article history:
Received 5 September 2017
Revised 6 November 2017
Accepted 9 December 2017
Available online 9 December 2017

Keywords:
NDDCT
Hot state experiment
Crosswind
FLF
Quantitative description
Flow separation

ABSTRACT

The natural draft dry cooling tower (NDDCT) is a critical facility for an indirect dry cooling power plant in arid area for its merit of excellent water-saving. While crosswind degrades the performance of a NDDCT by changing the flow field inside and outside. In order to quantitatively study the influence of different flow characteristics on the performance of a NDDCT, hence to grasp the affecting mechanism of crosswind, a half-cylindrical computational fluid dynamics (CFD) model of a Heller type 660 MW NDDCT is developed and validated by a hot state modelling test rig. A flow loss factor (FLF) is derived and verified to linearly describe the effect of local flow field changing on the overall performance of a NDDCT. Based on the conjoint studies of the local FLF variation trends and the changing processes of correspondent flow characteristics in each specific flow segments, the critical factors influencing the performance of a NDDCT are identified under different crosswind conditions.

0. Introduction

Dry cooling technology has been increasingly used for power generation in arid countries and regions for its merit of excellent water saving [1]. While dry cooling is found to be sensitive to ambient crosswind [2]. Natural draft dry cooling tower (NDDCT) is the main facility of an indirect dry cooling system, with air-cooled heat exchanger bundles vertically arranged around the circumference of cooling tower, or horizontally configured inside cooling tower [3]. It is found that crosswind at 20 m/s may decreases the ventilation rate of a NDDCT by 36%, and increases the temperature of circulating water by 7 °C [4]. It is also reported that crosswind could cause 7.5 °C increment of the air temperature inside the tower [5], or more than 25% decrement of the heat transfer efficiency [6].

Crosswind degrades the cooling performance of a NDDCT by affecting the air flow field at the inlet and outlet and inducing complex vortices inside and outside the tower [7]. As found by Wei et al. [5], crosswind affects the cooling efficiency of a NDDCT by forming an unfavourable pressure distribution at the tower inlet, disturbing the hot plume flow field, and causing cold air back flow at the leading edge of the tower outlet. Tang et al. [8] found that crosswind lead to a horizontal air flow or even a cross ventilation inside the tower. Zhai et al. [9] found that inlet air flow from the leading and rear radiators converge to produce complex vortices. Zhang et al. [10] and Goodarzi [11] both reported that crosswind squeezes the plume flow. Wang et al. [7] found that the influence of the flow field deformation of different sections on the performance of a NDDCT changes as the crosswind increases.

In the past decades, the influence of crosswind on the performance of a NDDCT are investigated experimentally and numerically. As to the experimental studies, cold state test is favoured for its easy control and measurement, while on sacrifice of

* Corresponding author.
E-mail address: wang_wl@tsinghua.edu.cn (W. Wang).

https://doi.org/10.1016/j.applthermaleng.2017.12.038
1359-4311/

Nomenclature

NDDCT	Natural draft dry cooling tower
CFD	Computational fluid dynamics
FLF	Flow loss factor
g	gravitational acceleration, m/s^2
P	pressure, kPa
q	ventilation rate, kg/s
S	sectional area, m^2
V	crosswind velocity, m/s
v	velocity, m/s
U	potential flow, kg/(m^2·s)
Z	the vertical height, m
Greek letters	
Δ	delta, variable increment
ξ	local resistance coefficient
Ω	flow resistance 1/m^2
ρ	air density, kg/m^3
ν	kinematic viscosity, m^2/s
Superscripts	
$*$	summarised value
Subscripts	
0	baseline value
f	flow
m	mass
-r	reference
ref	reference
t	tunnel
-t	total

destroying the flow field inside the tower by adopting mechanically forced draft device to mimic the natural draft upward flow [12]. Hot state test directly model the natural draft ventilation of the NDDCT by introducing hot circulating water system [5]. However, limited by the bench scale and water temperature, it is even hard to generate a measureable natural draft upward in-tower flow, not to mention meeting the required scaling laws [13]. Consequently, as the fast development of computational fluid dynamics (CFD), more people tend to adopt CFD to study the performance of a NDDCT under crosswind condition without sound validations [14].

The cooling performance of a NDDCT was investigated previously vie CFD methods. Due to the natural draft, an axial symmetrical air flow field is formed inside and outside the NDDCT. An inner wall vortex is found right above the lower chimney borderline [4]. Under crosswind condition, the tangential flow acceleration is found at the side sections, encouraging the uneven air flow intake, and inducing two symmetrical mainstream vortices inside the NDDCT. Besides, air inhalation at the centre areas on both the windward and leeward sides are reinforced, thanks to the positive windward enhancement and the convergence of the leeward flow respectively [7]. When crosswind rises to ~12 m/s, a suck-back vortex arises inside the outlet of the tower at the windward side. When the crosswind rise to ~14 m/s, a heated air suck-out phenomenon is found around the side radiators [15].

As crosswind increases, the tangential velocity in the side sections increases in an approximately linear way. The inner wall vortex grows bigger and stronger at the windward side [4]. The mainstream vortices grow stronger, with their centres moving gradually from the outlet to the bottom of the tower [7], and separating apart from the centre of the tower to the side [16]. When crosswind rises to ~14 m/s, the enhanced air intake on the leeward side is broken down gradually, generating a batch of small vortices around the leeward radiators. Meanwhile, the upward flow channel is suppressed to a minimum at ~14 m/s, and then expands [7].

However, up to now, the influence of the flow characteristics on the performance of a NDDCT can only be analysed qualitatively. In order to quantitatively evaluate the effects of the flow field variations, this paper presents a flow loss factor (FLF) as an indicator to linearly describe the effect of the local flow field changing on the overall performance of a NDDCT based on validated CFD simulation. Hence, the affecting mechanism of crosswind on the performance of a NDDCT can be identified.

1. Methods and validation

1.1. Basic assumptions

Based on the knowledge that, on a certain power load, as the back pressure increases in a certain range, the steam rate per unit power generation increases slightly, while the latent heat of the saturated vapour decreases correspondently [17], resulting a negligible heat rejection change in the condensing process. In a case study, a 5 kPa increment of back pressure results only ~2% increment of the exhaust heat. Hence, the exhaust heat released in a condenser (same as that released in NDDCT radiators) is regarded as a constant under different crosswind condition. Consequently, the heat rejection of the radiator bundle is simplified as a constant heat source in CFD model, and mimicked by evenly assembled heating rods in the experiment.

1.2. Modelling of the CFD approaches

The NDDCT of interest is of Heller type and installed in a 660 MW power plant in China. The NDDCT has a total height of 170 m, a radiator height of 24 m, an outlet diameter of 84.466 m, and a base radiator diameter of 146.17 m.

On the ground of a specific crosswind direction, the horizontal fluid field around the circle of the radiator bundle should be symmetrical to the axis connecting the centre points of the windward side and the leeward side. Then, a plane could be obtained if the symmetry axis is extended along the vertical direction. Consequently, this plane should be the symmetry plane of the whole flow field around the NDDCT. Based on the symmetry of the flow field around a NDDCT under crosswind condition, a half-cylindrical computational field, with a dimension of 1200 m (diameter) × 1700 m (height), is set to numerically investigate the NDDCT performance as shown in Fig. 1(a). The computational field is 8–10 times large in each direction to eliminate the unrealistic effect of the domain boundaries. The structure of the cooling deltas, tower shell, support base and joint faces between adjacent radiators are all constructed in accordance with the real conditions, as shown in Fig. 1(b).

By adopting large computational domain to let the crosswind develop a reasonable velocity profile, the crosswind is assumed to be constant at the inlet. The outflow boundary is appointed to the outlet. The other surfaces including the ground, the