Energy & Buildings 175 (2018) 87–101

Contents lists available at ScienceDirect

Energy & Buildings

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

Thermal buoyancy driven flows inside the industrial buildings primarily ventilated by the mechanical fans: Local facilitation and infiltration

Shuo-Jun Mei[a,b,c], Jiang-Tao Hu[a,b,c], Di Liu[d,*], Fu-Yun Zhao[a,b,c,**], Han-Qing Wang[e]

[a] Key Laboratory of Hydraulic Machinery Transients, Ministry of Education Wuhan University, Wuhan, Hubei, China
[b] Hubei Key Laboratory of Waterjet Theory and New Technology, Wuhan University, Wuhan, Hubei, China
[c] School of Power and Mechanical Engineering, Wuhan University, Luo-Jia-Shan, 430072 Wuhan, Hubei, China
[d] College of Pipeline and Civil Engineering, China University of Petroleum, Qingdao, Shandong, China
[e] School of Civil Engineering, University of South China, Hengyang, Hunan Province, China

ARTICLE INFO

Article history:
Received 28 November 2017
Revised 29 June 2018
Accepted 5 July 2018
Available online 19 July 2018

Keywords:
Natural ventilation of industrial buildings
On-site measurements
Local ventilated cooling
Mechanical assisted natural ventilation

ABSTRACT

The turbine hall houses a number of components with intense heat release, resulting in a hot and humid air environment and making it very unpleasant for plant staffs to conduct maintenance work there, particularly at a hot outdoor environment. The traditional pure buoyancy driven natural ventilated could fulfil the overall heat exhausting requirement, however, creating hot environment at several positions due to the vortex airflow structure. As only part of the building need to be cooled, a localized cooling system with assisting mechanical fans is used. Buoyancy driven natural ventilated flows primarily occupying this turbine hall have been investigated by a computational fluid dynamics (CFD) model and an on-site measurement. The ventilation airflow is solved by 3D steady state Reynolds Averaged Navier-Stokes (RANS) equations in conjunction with the SST k-ω turbulence model. The numerical codes and procedures were validated by comparing with on-site measurements. The concept of mean age of air is used to evaluate the natural ventilated efficacy. Following that, the potential of using mechanical vents and slabs to cool equipment work area at this buoyancy driven natural ventilated hall is analysed. The simulation results show that the mechanical vents could decrease air temperature and increase air exchange efficiency at work area effectively on condition that the local airflow pattern is well organized, which is strongly related to the ejection direction and velocity. Present research could benefit future development of the ventilation design in the power plant buildings.

1. Introduction

Turbine hall, which is the most important part of a thermal power plant, houses a number of components vital to the generation of electricity from the steam that comes from the boiler. Therefore, heat releasing from this hall is very intensive, resulting in a hot and humid indoor environment, which makes it almost impossible for plant staffs to conduct maintenance work there, particularly in the regions with hot climate. Proper ventilation is needed to provide adequate fresh air and remove heat.

As the heat release is intense in the turbine hall, thermal buoyancy driven natural ventilation could be generally induced and then dominate the whole room airflow patterns, which also has great potential for reducing the energy consumption in buildings [1–4]. However, several studies have found that natural ventilation could not provide sufficient ventilation during a certain time of one calendar year in many regions [5] especially for buildings with large spaces [6]. Although full mechanical ventilation system could achieve sufficient ventilation, the energy consumption is tremendous at such a large space with intense heat release [7,8]. Hybrid ventilation (or mixed-mode ventilation) combining the natural and mechanical cooling modes is a potential solution [9].

1.1. Hybrid ventilation

The hybrid ventilation system exploited the natural cooling mode when the outdoor climate is suitable; therefore, it not only saves energy but also provides better indoor air quality than a unique mechanical system [6,10]. Furthermore, the system could provide better thermal comfort than a pure natural ventilation

* Corresponding author.
** Corresponding author at: School of Power and Mechanical Engineering, Wuhan University, Luo-Jia-Shan, 430072 Wuhan, Hubei, China.
E-mail addresses: liudi@upc.edu.cn (D. Liu), fyzhao@whu.edu.cn, zhao@bv.tum.de (F.-Y. Zhao).

https://doi.org/10.1016/j.enbuild.2018.07.014
0378-7788/

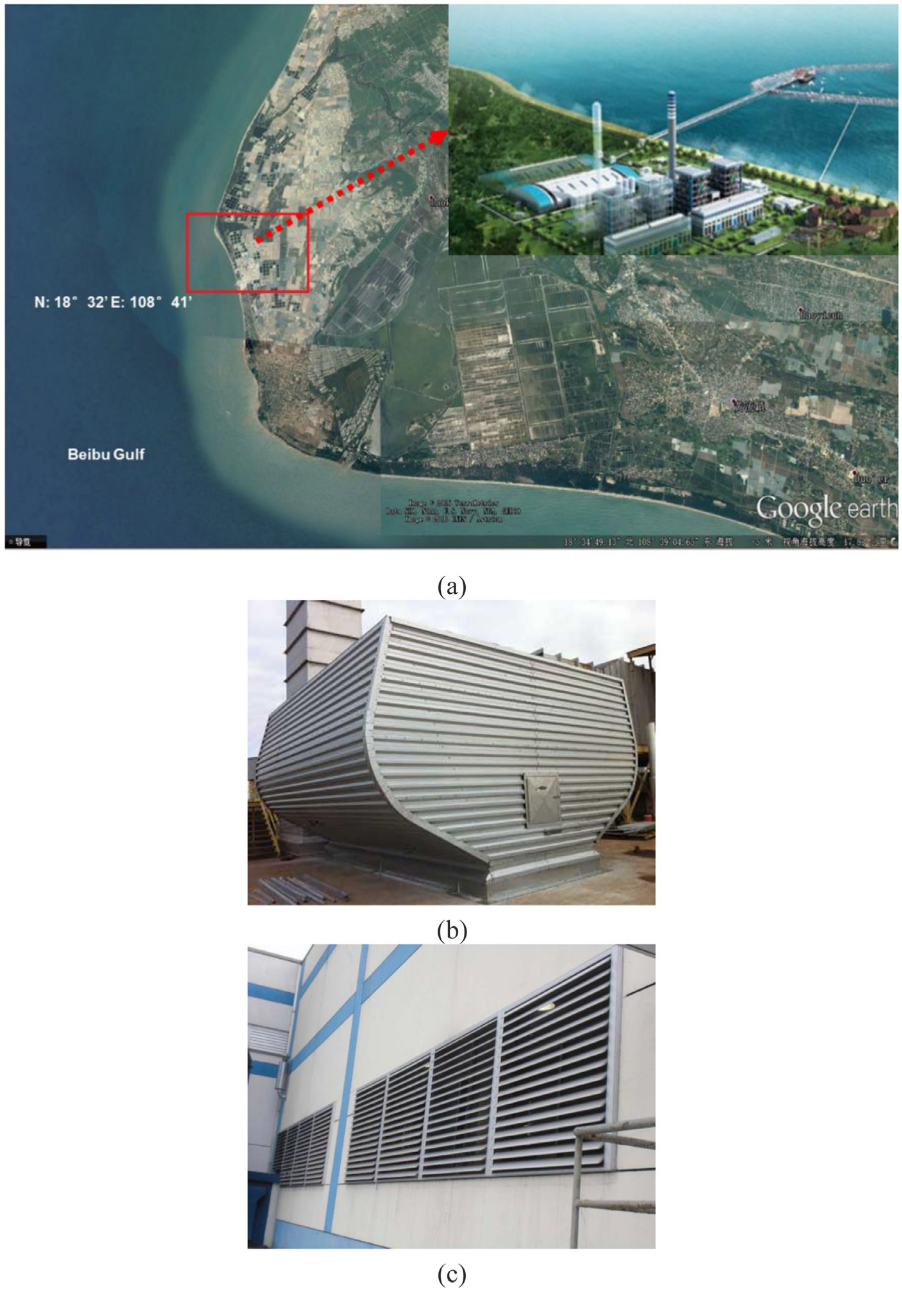

Fig. 1. Spatial location and eye-bird view on the thermal power plant and surroundings from northern direction. Configurations of the openings roof ventilators (b) and alloy shutters (c) for natural ventilation were also included.

system [10]. Although this hybrid ventilation system has great potentials in reducing energy consumption and improving indoor air quality simultaneously, systematic design optimisation is still needed to ensure the optimal performance of this hybrid ventilation system [6].

Regarding of the built environment, appropriate thermal distributions and room airflow patterns are significant for improving indoor air quality and reducing energy usage simultaneously [11,12]. Particularly, when the cooling is only needed in specific regions (for example, the working area), well-designed hybrid ventilation airflows could further reduce the energy cost and enhance the indoor air quality simultaneously [13–15].

1.2. Air flow patterns in hybrid ventilation

For the safety of thermal units and comfort of dwellers, it is important for building operators to evaluate the airflow patterns and indoor air temperature distributions caused by the hybrid ventilation across buildings [16]. In the past, a number of methodologies and models, including analytical models [17,18], empirical models [6], multi-zone models [7], zonal models, small-scale experimental models, full-scale experimental models and computational fluid dynamics (CFD) models [1,16], have been proposed to evaluate the airflow patterns and indoor air temperature distributions, which covers. For buildings with simple geometries, such as a single zone with one lower and one upper opening, analytical models could be developed to evaluate the ventilation rate depending on density differences, temperature differences, or surplus heat. These analytical models provide a basic understanding of the ventilation process. For buildings with complex geometries, CFD is a useful tool to model the detailed performance of natural ventilation and hybrid ventilation [19]. Compared to experimental and mathematical analysis, CFD can provide the velocity, turbulence and temperature distributions in the whole flow field [20]. At early design stage, the