Study on Natural Ventilation in Gridiron of High-pile Wharf to Enhance The Durability Performance

Qin Zhang¹, Yubao Zhou¹ and Yibin Zhang¹, ²*

¹ Department of Hydraulic Engineering, Civil Engineering college, Tongji University, Shanghai, 200091, China.
² Quanzhou Port Authority of Fujian Province, Quanzhou, Fujian, 362000, China.
*Corresponding author’s e-mail: zybdy@126.com

Abstract. The salt which origins from offshore zone is easy to gather in the semi-closed space formed by beams and slabs of pile-supported wharf called gridiron space, and that could induce the premature failure of durability of the wharf’s substructure. Previous work commonly focused on material-level measures which will lead to increased costs such as anti-corrosion concrete and anti-corrosion coatings to eliminate the adverse effects of that gathering of salt fog. In fact, the salt fog could be extracted from the gridiron by natural ventilation in some cases, so it is necessary to examine the flow characteristics in the gridiron space. In this work, the numerical simulation has been conducted with natural ventilation and auxiliary ventilation group. Critical value of the blocking ratio (defined as height ratio of edge stringer to air inlet) that makes the airflow inside the gridiron reach the strongest turbulence is found in natural ventilation group. In addition, the effect of different auxiliary ventilation schemes is compared in auxiliary ventilation group. Some recommendations for designing were proposed to enhancing pile-supported wharf’s durability.

1. Introduction

As one of the common structures using in the port, the high-pile wharf bears important transportation tasks, so enhancing the durability performance of harbour structure is vital. However, high-pile wharves are vulnerable to chloride penetration. The degradation mainly occurs in cyclic wet-dry zone of the pile foundation and beam-slab structure of the superstructure. This study focuses on the latter.

The gridiron is constituted by beams at the bottom of superstructure of high-pile wharf formed a considerable space to collect the marine aerosol salinity (figure 1), which is a primary source of chloride ions. These ions are transported with the aerosol and, when in contact with concrete surfaces, they are partially accumulated and transported into concrete [1]. The concentration of marine aerosol salinity may be especially obvious in the high-level water condition, because of low air inlet side (figure 2).
Theoretically, the concentration of marine aerosol salinity will not happen if inward new wind could distribute the flow field inside the in gridiron. However, to authors’ best knowledge, there is almost no research conducted on enhancing durability of high pile wharves’ superstructure by optimizing ventilation in the gridiron. Therefore, this paper will study on air flow field characteristic in the gridiron under the several different vents arrangement based on numerical method.

2. Methodologies
In this section, the models applied to describe air flow driven by natural wind in the gridiron of high pile wharf are introduced first, followed by an introduction of the governing equations to solve airflow field based CFD and details of solution.

2.1. Geometrical model
To represent airflow through real gridiron, as in the case of single-sided buoyancy-driven natural ventilation, the geometrical model should be determined carefully. Generally, the airflow is affected by factors such as gridiron style, inlet velocity of fluid, heat sources and the building envelope. However, for computational modelling, the effects of the domain size (especially if the exterior environment is modelled) and applied boundary conditions are also important factors. Accordingly, a balance is needed between the computing efficiency and the modelling complexity in establishing the geometrical model. A compromise may simplify the model into a beam plated construction, which can represent a superstructure in a wharf. The strategy has been widely applied in other studies on single-sided natural ventilation. [2–7] Therefore, the study herein will only consider the external conditions for the littoral environment and the configurations of different vent holes. Different computational domains and boundary conditions are test to establish the geometrical model here.

The geometrical model was established in accordance with a typical high pile wharf structure in southern China, where measurements for average wind velocity took place during March.

Three ventilation schemes have been designed, in which vent holes on the slab or beams is shown in the figure. The holes are near the corner because the salt fog is most likely to accumulate in the corner, and corners are also observed to be the most corroded Figure 3 presents the distribution of the vent holes. Table 1 shows the parameters of the three schemes.
Table 1. Three schemes.

| Parameter(mm) | R  | a  | b  | c   | d   | Physical open area |
|---------------|----|----|----|-----|-----|--------------------|
| Scheme1       | 150| /  | /  | 1200| 300 | 4.239×10^6         |
| Scheme2       | 150| 1500| 1150| / | / | 3.3912×10^6       |
| Scheme3       | 150| 1500| 1150| 1200| 300 | 7.6302×10^6       |

2.2. Governing equations

In order to model airflow, the governing conservation equations for fluid flow are solved. In the study, the RANS model is used with the Boussinesq model and k-ω turbulence equations. Since these are very widely adopted equations available from the literature [8-11], they will be discussed briefly here.

The continuity, energy and momentum equations for an incompressible fluid are written as:

\[
 \nabla \cdot \rho = 0
\]

\[
 \rho \left( \frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right) = -\nabla p + \mu \nabla^2 \mathbf{u} + \rho g
\]

\[
 \frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T = \frac{K}{\rho c_p} \nabla^2 T
\]

where \( \mathbf{u} \) is the air velocity, \( K \) is the thermal conductivity, \( T \) is the air temperature, \( g \) is the gravitational acceleration, and \( \mu \) is the absolute viscosity, \( \rho \) is the fluid density.

So as to consider buoyancy effects caused by variation of air density, the Boussinesq model is applied. The model approximates buoyancy term in Eq. (2) as:

\[
 \rho g = \rho_{ref} \left[ 1 - \beta (T - T_{ref}) \right] g
\]

where \( \rho_{ref} \) is the reference density, and \( \beta \) is the air thermal expansion coefficient, \( T_{ref} \) is the operating temperature.

Among various turbulence models, two-equation eddy viscosity models are generally better than zero- or one-equation models. The \( k-\omega \) model is a two-equation eddy-viscosity model that has been found to yield satisfactory performance in accuracy and stability when modeling airflow [10]. The BSL \( k-\omega \) model is employed to describe the turbulent airflow in the study. Compared to the standard \( k-\omega \) model, an additional cross-diffusion term is incorporated in the \( \omega \) equation and the modeling constants are modified. The specific forms are given by [11]:
\[ \frac{\partial}{\partial t} (p k) + \nabla \cdot (\rho u_k) = \nabla \cdot (\Gamma_k \nabla k) + G_k - Y_k \]  
\[ \frac{\partial}{\partial t} (p \omega) + \nabla \cdot (\rho u_\omega) = \nabla \cdot (\Gamma_\omega \nabla \omega) + G_\omega - Y_\omega + D_\omega \]  

where $k$ is the turbulence kinetic energy, $\omega$ is the specific dissipation rate, $\Gamma_k$ and $\Gamma_\omega$ represent the effective diffusivity of $k$ and $\omega$, $G_k$ and $G_\omega$ represent the generation of $k$ and $\omega$, $Y_k$ and $Y_\omega$ represent the dissipation of $k$ and $\omega$, and $D_\omega$ represents the cross-diffusion term.

The boundary conditions are summarized in Table 2 for the building and the external domain that is coincident with the computational boundary. The no-slip boundary is imposed on all solid surfaces. Temperatures are specified for the side walls and the sea, but the wharf are modeled as adiabatic.

| Location     | Slab     | Longitudinal Beams | Cross beams | Vent holes | Sides     | Sea      | Top      |
|--------------|----------|--------------------|-------------|------------|-----------|----------|----------|
| Type/value   | No-slip/adiabatic | No-slip/adiabatic | No-slip/adiabatic | No-slip/adiabatic | No-slip/16-20°C | No-slip/16-20°C | No-slip/16-20°C |

3. Results and Discussion

Figure 4. presents the computed velocity profiles along the vertical centerline of the gridiron for each scheme. The parameters for window height are normalized by the following relationships:

\[ h^* = y / h \]  

where $y$ is the vertical coordinate, $h$ is the height of the gridiron, $v$ is the z-velocity.

Figure 4. The velocity along the vertical centerline of the gridiron for each scheme.

Figure 4 presents the air flow profiles in the center of the wharf for different schemes with similar physical openings. The non-dimensional air velocity varying with height ($H^* = y/H$) is illustrated in Figure 4. The air velocity is higher as air goes through at the lower part of the gridiron. There is not much difference among the velocity profiles, which are all relatively uniform, except for the scheme 1 case, which shows fluctuations near the middle part of the gridiron. Air stratification exists in all cases and the velocity varies parabolic to the height.

Due to the opening hole in the longitudinal beam, the air flow from the hole into the interior of the gridiron has disturbed the original velocity field distribution. Especially in the first two holes, it has an excellent effect, completely affecting the internal air flow. The swirl caused the salt fog, which had gathered in the corner, to blow away and the stagnant air zone disappeared.
According to the different performances of schemes, Figure 5 further presents detailed air flow profiles at the central z-plane for 3 cases.

In scheme 1, the ventilation vent is generally arranged on the top of the longitudinal beam, forming a adherent jet along the top. After the air flow is ejected from the air duct, the air flow develops according to the free jet. However, due to the limitation of finite space of gridiron and the continuity of wind flow, the flow opposite the direction of jet flow is formed at the bottom of gridiron.

In scheme 2, at the opening of the panel, the pressure difference is caused by the velocity difference between the upper and lower surfaces. Because the velocity of the upper surface is much higher than that of the lower surface, the pressure on the upper surface is relatively small. The air in the panel flows out, removing the salt mist in the corner and affecting the flow distribution in the gridiron.

In scheme 3, good ventilation was achieved in the first half, which was better than the first two schemes. However, improvement was not evident in the second half. The reason may be that both types of openings are mainly used in the upper part, and the effect is repeated. Consider the cost and impact on the strength of the structure, scheme 3 turns out not to be particularly reasonable.

4. Conclusion
This paper discusses the detailed airflow characteristics inside and outside the high-pile wharf for each ventilation scheme. The performance of CFD was compared to quantify the ventilation efficiency through variable vent configurations.

The air flow profiles inside the wharf varied with vent layout, though the physical vent openings were identical. It was shown that the 3 schemes can have a reasonable effect on the ventilation rates, and the scheme 2 and scheme 3 performed better because they function equally in each region. Considering practical operation, scheme 2 is obviously more suitable for high pile wharf because it has smaller area, less impact on structural strength and lower cost. Simulation and experience gained so far clearly demonstrate that vents on the slab can relatively reduce the adverse effects of salt spray under appropriate conditions effectively and increase the efficiency of ventilation. This research can contribute to the practical application of wharf ventilation.

References
[1] Thomas, J.E., Margeson, G.R. (2001) Impacts of Sediment Mounds under Pile Supported Wharf Structures. In: Ports Conference. 1-11.
[2] Bangaleea, M.Z.I., Lina, S.Y., Miaua, J.J. (2012) Wind driven natural ventilation through multiple windows of a building: a computational approach. Energy Build, 45: 317-325.
[3] Caciolo, M., Stabat, P., Marchio, D. (2011) Full scale experimental study of single-sided ventilation: analysis of stack and wind effects. Energy Build, 43: 1765-1773.
[4] Wang, H.W., Karavab, P., Chen, Q. (2015) Development of simple semi-empirical models for calculating airflow through hopper, awning, and casement windows for single-sided natural ventilation. Energy Build, 96: 373-384.
[5] Grac, G.C., Linden, P. (2016) Ten questions about natural ventilation of non-domestic buildings. Energy Build, 107: 263-273.
[6] Ai, Z.T., Mak, C.M. (2014) Determination of single-sided ventilation rates in multistory buildings: evaluation of methods. Energy Build, 69: 292-300.
[7] Freirea, R.Z., Abadieb, M.O., Mendes, N. (2013) On the improvement of natural ventilation models. Energy Build, 62: 222-229.

[8] Hauser, M.B. (1994) A Handbook of Natural, Climatic Control. Journal of Bioscience & Bioengineering, 110: 582-587.

[9] Versteeg, H., Malalasekera, W. (2007) An Introduction to Computational Fluid Dynamics: The Finite Volume Method, second ed. Harlow Publishing. Prentice Hall.

[10] Zhai, Z., Zhang, Z., Zhang, W., Chen, Q. (2007) Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD: Part-1: summary of prevalent turbulence models. Hvac & R Research. 853-870.

[11] Menter, F.R. (1994) Two-equation eddy-viscosity turbulence models for engineering applications. Am. Inst. Aeronaut. Astronaut. J, 32: 1598-1605.