

Article

Characteristics of the Wind Environment above Bridge Deck near the Pylon Zone and Wind Barrier Arrangement Criteria

Dalei Wang , Yiquan Zhang, Mengjin Sun * and Airong Chen

Department of Bridge Engineering, Tongji University, 1239 Siping Road, Shanghai 200092, China; wangdalei@tongji.edu.cn (D.W.); zhangyq987@tongji.edu.cn (Y.Z.); a.chen@tongji.edu.cn (A.C.)

* Correspondence: sunmengjin@tongji.edu.cn

Received: 6 December 2019; Accepted: 17 February 2020; Published: 20 February 2020



Abstract: Due to the complex arrangement of structural components in the vicinity of bridge pylon zones, the wind environment above bridge decks is very complicated. A sudden change in wind speed exerts an adverse effect on vehicle control stability. In order to investigate the characteristics of the flow field in the vicinity of the bridge pylon, the wind environment near an inverted Y-shaped pylon is studied by experimental and numerical methods. From the flow visualization and the wind speed measurement in the wind tunnel and the numerical simulation created using Fluent software, specific patterns of the direction and magnitude of wind speed at a range of vehicle height above the bridge deck near the pylon zone were observed along the longitudinal direction. This distribution pattern of the wind environment can effectively guide the wind barrier arrangement near the bridge pylon zone. Combined with the two safety evaluation indicators proposed in this paper, the optimal arrangement scheme of wind barriers in the bridge pylon zone of Sutong Bridge is determined. This paper deepens the understanding of the wind environment near the pylon zone and proposes an evaluation method for the wind environment near the pylon zone, which can serve as the basis for wind barrier arrangement in similar research projects.

Keywords: wind environment; bridge pylon zone; wind tunnel test; numerical simulation; wind barriers; evaluation method

1. Introduction

Since the flow field around the bridge pylon is unevenly distributed [1], the vehicle is inevitably affected by the change of crosswind speed whilst traveling on the bridge deck. Under the action of strong dynamic disturbances, high-speed vehicles may have control instability problems such as side slip and rollover [2–4].

In order to evaluate the driving safety on a bridge under a crosswind, numerous studies have been carried out on the aerodynamic forces acting on vehicles on bridges. Baker and Humphreys [5] obtained the aerodynamic coefficients of automobiles and discussed the accuracy of the different types of wind tunnel experimental results. Zhu et al. [6] compared the aerodynamic coefficients of vehicles in open area and on bridge using wind tunnel tests. In addition, Han and Cai [7,8] found that the wind turbulence, vehicle interference, and vehicle distance from the windward edge significantly affected the aerodynamic coefficients of vehicles. These studies were focused on the aerodynamic coefficients of vehicles on bridge girders, while for cable-stayed bridges and suspension bridges, the flow separation around the pylon led to a sharp increase in wind speed, which is more unfavorable for vehicle stability. Salati et al. [9] studied the aerodynamic performance of high-sided vehicles as well as the flow field around the pylon when the vehicle was passing through the wake of bridge

pylon zone by the numerical method. The flow field near the bridge pylon zone is complicated, since it contains the flow around the girder and the pylon, which are dominated by different 2D planes.

From far away to the center of bridge pylons, the wind speed changes from the normal state to the amplification state and then quickly to the low-velocity state. This sudden change of wind speed in a short time period poses a great threat to driving safety. Some studies have examined this problem. In [10–12], the effects of wind on stationary vehicles and vehicles moving adjacent to the bridge pylon were studied, and the influence of the wind barrier on the wind environment above the bridge deck was determined through a wind tunnel experiment and numerical simulation. Argentini et al. [13] measured the aerodynamic force of a stationary vehicle near the pylon zone with and without wind barriers through a wind tunnel experiment, and applied the smoke display method to observe the flow field near the pylon zone. Similar research was conducted by Li et al. [14], who measured the sudden changes of the aerodynamic forces of a train when it was passing through the bridge pylon. Wang et al. [15] compared the aerodynamic coefficients of three different types of vehicles with different sizes in an open area and in the pylon zone, and analyzed the changing patterns of the aerodynamic coefficients under the influence of wind barriers. The vehicle driving stability near the pylon zone was also studied and a multiparameter evaluation method was proposed, based on the drivers' reactions [16]. Most studies are concerned with the aerodynamic force on the vehicles near the pylon zone [17], and there is little research revealing the characteristics of the wind environment near the pylon zone. However, understanding the distribution pattern of the flow field in the vicinity of the bridge pylon zone is the foundation for evaluating the driving safety of vehicles passing by the bridge pylon zone.

A large number of studies have shown that a sudden change in wind speed near the bridge pylon is unfavorable for driving stability [18], and countermeasures have been explored for improving the wind environment at the bridge deck and ensuring driving safety [19,20]. Wind barriers, as an effective countermeasure, have had increasing attention from scholars [21–23]. Alonso-Estebanez et al. [24] numerically studied and compared the aerodynamic coefficients of a truck on an embankment both with and without the wind fences installed. Kozmar et al. [25] obtained the optimal height and porosity of wind barriers that can effectively improve the deck wind environment of the viaduct by a wind tunnel experiment and numerical simulation. Telenta et al. [26] used numerical simulation and wind tunnel tests to analyze the bar inclination effect on the flow characteristics behind the wind barrier. However, there are limited methods available that consider the control stability of the vehicles near the pylon zone when evaluating wind barrier arrangement. Therefore, a quantitative study is very important for the evaluation of the wind environment near the pylon zone.

In this study, the detailed features of the flow field above the deck near the bridge pylon are investigated based on a case study of Sutong Bridge. Sutong Bridge is a cable-stayed bridge with a main span of 1088 m, and its pylon is an inverted Y-shaped reinforced concrete structure with one connecting girder between two pylon legs. The characteristics of the wind environment above the deck near the pylon zone are studied by wind tunnel experiments and numerical simulation methods. In order to deeply investigate the influenced range and the wind environment characteristics dominated by the flow around a bridge pylon or bridge girder, the experiments of wind speed measurement and flow visualization were applied in wind tunnel tests, then the main characteristics of wind environment are reproduced by the numerical simulation method. Based on a comprehensive knowledge of the flow field pattern in the vicinity of the bridge pylon zone, the evaluation method and specific indicators of wind barrier arrangement are proposed based on the driving stability of the vehicle under a crosswind, which can provide guidance for decision-making in wind barrier design.

2. Theoretical Basis

A bridge pylon consists of a blunt body, which can cause great changes in the wind speed at the bridge deck. The amplification of wind speed around the bridge pylon will seriously affect the wind environment in terms of driving safety. Since the bluff body flow caused by the main girder and the

pylon is dominant in two different 2D planes, it is difficult to determine the dominant flow in the mutual influencing range.

This research is conducted based on the wind tunnel test and Computational Fluid Dynamics (CFD). For the CFD method, the basic equations mainly include the continuity equation, momentum equation, and energy equation [27]. Since the heat transfer effect of air is not considered in this study, the energy equation is not included.

2.1. Continuity Equation

The mass conservation equation for fluid microelement is referred to the continuity equation. That is, the velocity of fluid mass in a fixed volume region is equal to the mass difference between the inflow and the outflow, expressed by Equation (1):

$$\frac{\partial \rho}{\partial t} + \sum_{i=1}^3 \frac{\partial(\rho u_i)}{\partial x_i} = 0 \text{ or } \frac{D\rho}{Dt} + \nabla \cdot \rho \mathbf{U} = 0, \quad (1)$$

where ρ is the air density, t is the time, and u_i is the velocity component of air in the x , y , and z directions in a Cartesian coordinate system.

2.2. Momentum Equation

According to Newton's second law:

$$\frac{Du_i}{Dt} \rho dV = f_i \rho dV + \sum_{i=1}^3 \frac{\partial \sigma_{ij}}{\partial x_i} dV, \quad (2)$$

where dV is the volume in a fixed closed area of the fluid, and its mass is ρdV , f_i is the force component acting on the unit mass; and σ_{ij} is the stress in unit volume. If the fluid is unsteady, the variables ρ and u_i , are functions of time and space. Low-speed air is incompressible fluid, and its momentum equation can be obtained by Equation (3):

$$\frac{Du_i}{Dt} = f_i + \frac{1}{\rho} \sum_{i=1}^3 \frac{\partial \sigma_{ij}}{\partial x_i}. \quad (3)$$

Equations (2) and (3) can be interpreted as the acceleration of the fluid microelement equal to the resultant force of unit mass.

2.3. Navier-Stokes Equation

In the study of structural wind engineering, lower-speed air is generally considered as a Newtonian fluid. According to the constitutive equation of a Newtonian fluid, the equation of motion of the fluid can be obtained, that is, the Navier-Stokes equation [28].

The constitutive relationship of Newtonian fluids is expressed in Equation (4):

$$\sigma_{ij} = (-p + \lambda S_{kk}) \delta_{ij} + 2\mu S_{ij}, \quad (4)$$

where σ_{ij} and δ_{ij} are second-order symmetric tensors, p is the fluid pressure, S_{kk} is the trace of fluid, μ is the viscosity coefficient, and λ is a scale. According to the Navier-Stokes hypothesis, a simplified form can be obtained:

$$\sigma_{ij} = -p \delta_{ij} + 2 \left(S_{ij} - \frac{1}{3} S_{kk} \delta_{ij} \right). \quad (5)$$

Substituting Equation (5) into the momentum Equation (3), the resultant equation of motion is called the Navier-Stokes equation:

$$\frac{Du}{Dt} = f - \frac{1}{\rho} \nabla p - \frac{1}{\rho} \nabla \cdot \left(\frac{2}{3} \mu \nabla \cdot \mathbf{U} \right) + \frac{1}{\rho} \nabla \cdot (2\mu \mathbf{S}), \quad (6)$$

where the left side is the acceleration of the fluid particle, f is the body force acting on unit mass, $\frac{1}{\rho} \nabla p$ is the resulted pressure per unit mass acting on a microelement, $\frac{1}{\rho} \nabla \left(\frac{2}{3} \mu \nabla \cdot U \right)$ is the viscous volumetric expansion force acting on the unit mass of fluid, and $\frac{1}{\rho} \nabla \cdot (2\mu S)$ is the resultant force of the viscous partial stress tensor of a unit mass fluid.

Due to the nature of the incompressible fluid and ignoring the expansion of its volume, u is a constant. According to the above equation, the continuity equation and momentum equation can be expressed in Equations (7) and (8) respectively:

$$\frac{\partial u_i}{\partial x_i} = 0 \quad (7)$$

$$\frac{Du_i}{Dt} = f_i - \frac{1}{\rho} \frac{\partial p}{\partial x_i} - \frac{u}{\rho} \frac{\partial}{\partial x_i} \left(\frac{\partial u_i}{\partial x_i} \right) \quad (i = 1, 2, 3). \quad (8)$$

3. Experimental and Numerical Simulation Setup

3.1. Experimental Setups

The wind tunnel experiment is carried out at TJ-3 Wind Tunnel of the State Key Laboratory for Disaster Reduction in Civil Engineering at Tongji University in China. The segmental model is 4.5 m long and 1.7 m high (shown in Figure 1), the girder model width is 1.025 m, and the pylon model width is 1.08 m at the top and 1.49 m at the bottom. The scale ratio of 1:40 is applied to this experiment to improve the accuracy of the test model and the results. The wind attack angle is zero, and the inflow wind speed is 8.0 m/s and set to be uniform. Figure 1b shows the steel box girder section and the cross section of the pylon at the bridge deck elevation.

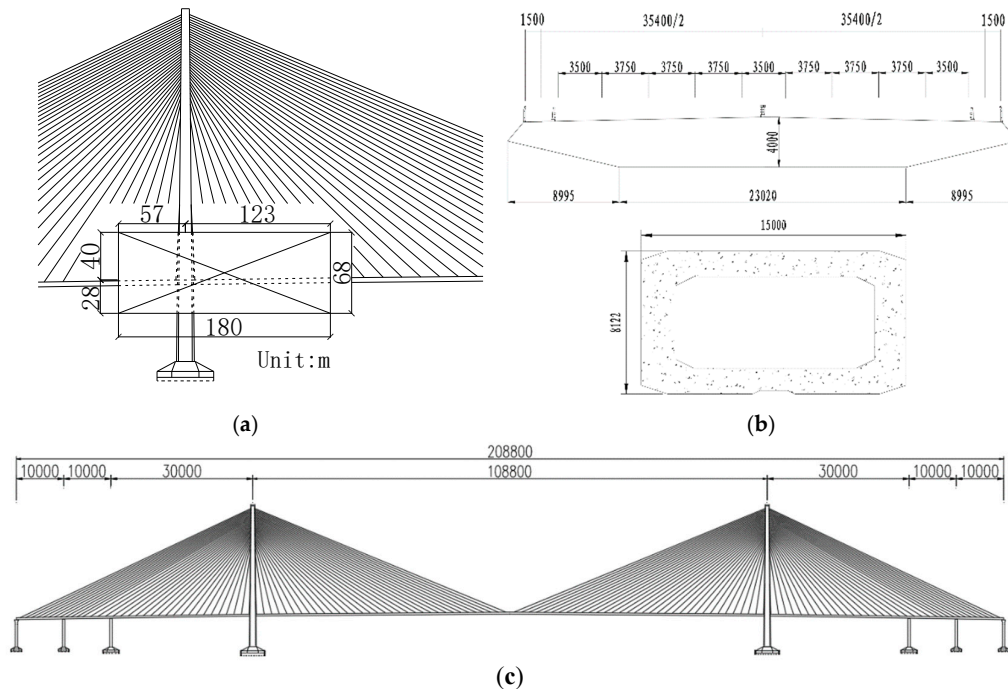


Figure 1. The size of Sutong Bridge. (a) Segmental model region. (b) Cross sections of the girder and the pylon column (unit: mm). (c) The profile image of the bridge (unit: mm).

The experiment was conducted under two conditions. C1 represents the bridge section with guardrails, while C2 represents the section without any ancillary facility. The segmental model in the wind tunnel test is shown in Figure 2.

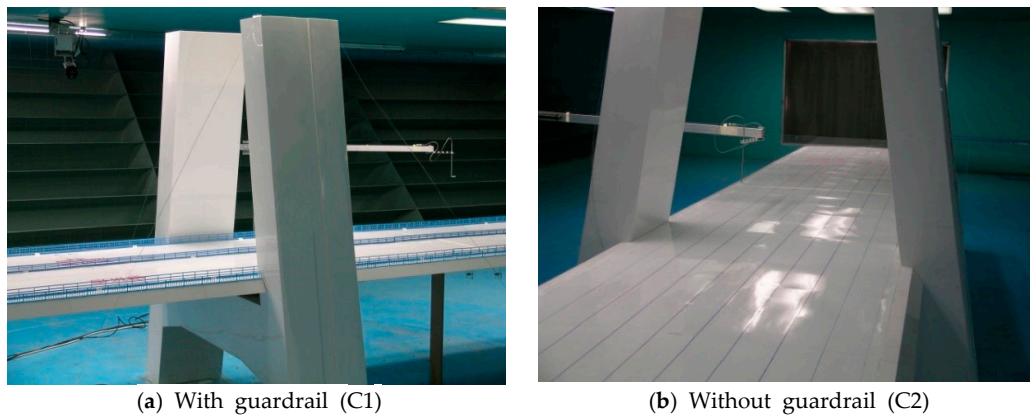


Figure 2. Segmental model.

3.1.1. Flow Pattern Observation

In order to study the flow fields above the bridge deck near the pylon zone, the tufts are used for the visualization of the flow characteristics. One end of the tuft is fixed to the steel wireline, and the other end can float freely under the crosswind. In addition, the wireline is stretched along the longitudinal direction and anchored at two different heights above the bridge deck.

3.1.2. Wind Speed above the Bridge Deck

In addition to the study of the flow pattern above the bridge deck near the pylon zone, the wind speed profile is also measured in the wind tunnel experiment. The results are used to analyze the influence of the pylon on the wind speed distribution above the bridge deck. Dantec's hot-wire anemometers (DENTEC Dynamics company, model number of 55P11, Denmark) are applied to measure the wind speed. According to the Cartesian coordinate system defined in Figure 3, the detailed positions of the measuring points in test model are given in the following.

- ① Upstream main lane ($Y = -0.184$ m): $x = 0, 0.05, 0.1, 0.15, 0.2, 0.25, 0.3, 0.4, 0.5, 0.6, 0.8, 1.0, 1.2, 2.0$ m;
- ② Downstream main lane ($Y = +0.184$ m): $x = 0, 0.05, 0.1, 0.15, 0.2, 0.25, 0.3, 0.4, 0.5, 0.6, 0.8, 1.0, 1.2, 2.0$ m;
- ③ For each longitudinal measuring position, the height $z = 0.005, 0.01, 0.015, 0.02, 0.025, 0.03, 0.04, 0.05, 0.06, 0.08, 0.1, 0.12$ m.

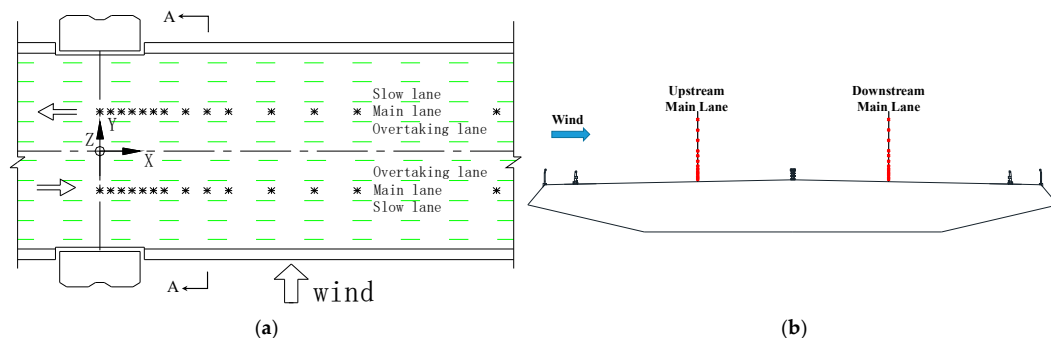


Figure 3. The positions of measuring points. (a) Longitudinal measuring point arrangement. (b) Vertical measuring point arrangement (A-A Section).

There are 336 measuring points in total. These include 14 measuring positions in the upstream main lane and also the relative 14 measuring positions in the downstream main lane, and each position has 12 measuring points along the height. The upstream lanes represent vehicle lanes that

are close to the inlet, and the downstream lanes represent the lanes that are far from the inlet flow. Furthermore, the central railing is the separation of upstream and downstream lanes.

3.2. Numerical Simulation Setups

In order to obtain more wind speed data for different lanes, the computational fluid dynamics (CFD) simulation software FLUENT (Canonsburg, Pennsylvania, U.S.) was used. In order to validate the numerical simulation method, the numerical simulation results are compared with the wind tunnel experiment results in the following section. The error between these two results are acceptable, and the reason that might cause the differences is also discussed. Through the grid independency study and the validation, the most suitable set of mesh is adopted for this study. The computational domain is shown in Figure 4. The top and the bottom parts of the pylon, which are vertically far away from the main girder are not considered in the numerical model, since these almost have no effect on the wind environment at the bridge deck. Moreover, the size of the flow field should be large enough that the boundary conditions will not affect the flow pattern at the bridge deck. In order to ensure that the flow field can be fully developed without squeezing the air flow, the dimension of the calculation domain is determined.

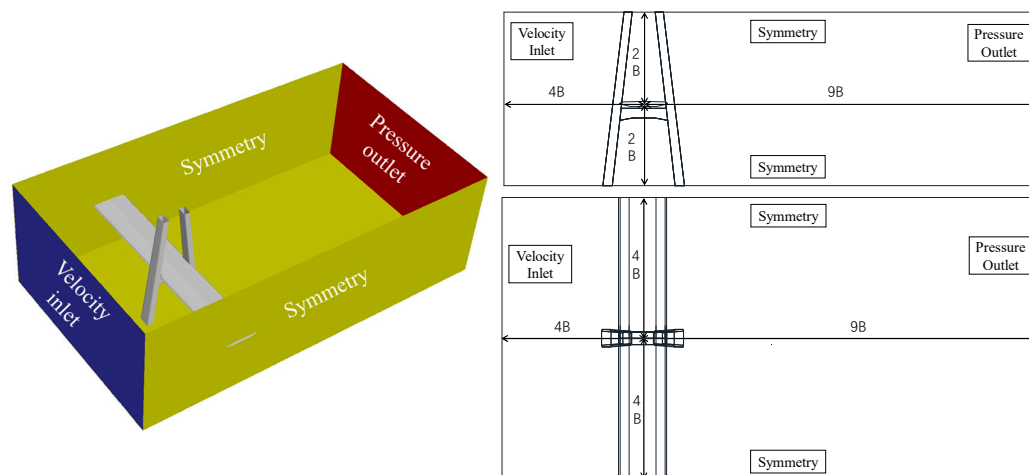


Figure 4. Boundary conditions of computational domain (B is the girder width 41 m).

The steady Reynolds-averaged Navier-Stokes (RANS) simulation method is applied, and the discretization numerical method is used to solve the flow field. The turbulence model is the Realizable $k-\epsilon$ model. The semi-implicit method for pressure-linked equations (SIMPLE) of the pressure-velocity coupling algorithm is used to solve the discretized problem. The discrete format is second-order upwind difference.

The inlet is uniform inflow with the speed of 8 m/s, and the inlet is set to be the same as that of the wind tunnel test. The outlet is set to be pressure outlet, the other four boundaries are set to be symmetry boundary condition. The pylon, bridge deck, wind barriers, and guardrails are all set to be no-slip wall boundary conditions (see Figure 4).

The number of grids is approximately 20 million, and the geometric model is shown in Figure 5. The mesh near the pylon and bridge deck is locally refined, which is shown in Figure 6. The near-wall mesh is quite dense, but it is relatively sparse in a less important area, which improves the accuracy of the calculation and saves on limited computer resources at the same time.

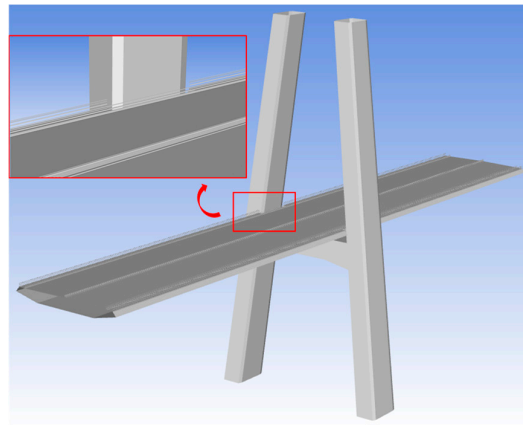


Figure 5. Geometric model.

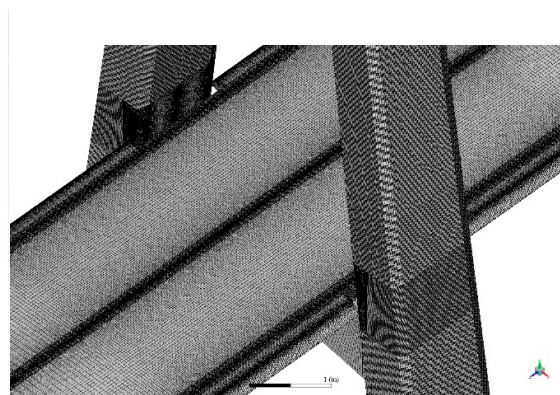


Figure 6. Near wall mesh.

4. Wind Environment around the Bridge Pylon

4.1. Flow Visualization at the Bridge Pylon Zone

In order to accurately reflect the wind environment above the deck under the effects of the bridge pylon and ancillary facility, the flow visualization experiment is conducted under the C1 condition (with guardrails) and C2 condition (without guardrails). According to the flow field visualization at the bridge pylon zone, the certain distribution pattern of the flow field along the bridge's longitudinal direction is revealed.

4.1.1. The Section with Guardrails

Figure 7 shows the flow field at $0.25H$ and $0.75H$ above the bridge deck in the wind tunnel experiment under C1 condition, where H is the girder height of 4 m. In the analysis of the flow observation experiment, several flow patterns are observed. Since the wake effect of the bridge pylon causes the wind speed behind the pylon column to be very small, the tufts are not floating at this zone for both measuring heights above bridge deck. At the height of $0.25H$, the vortex caused by the flow around the bridge deck makes the reverse flow and the flowing trace is opposite to the incoming flow, this is Zone I. At the same height, the flow is affected by the shelter effect of the bridge pylon, and the flowing direction near the pylon is consistent with the incoming flow in Zone II, which is shown in Figure 7a. Further analysis shows that the flowing direction of Zone I is dominated by the flow around the bridge girder and is located at the turbulent flow and recirculation zone after the separation point, while Zone II is dominated by flow around the bridge pylon. In Figure 7b, the flowing direction in both Zone I and Zone II is same as the inflow direction, which means that the flow field at the height of

$0.75H$ above bridge deck is higher than the turbulent boundary layer. The flowing direction of tufts changes from parallel to slight turning, which is also caused by the flow separation of the bridge pylon.

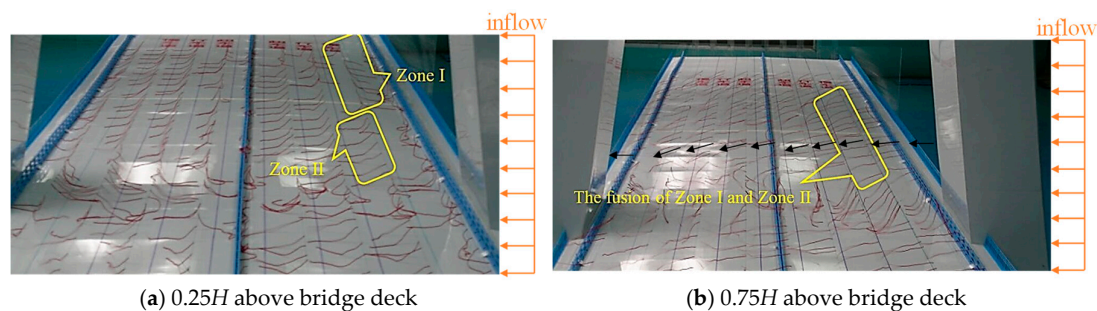


Figure 7. Flow field from experimental results (C1 condition).

In order to further understand the flow field above the bridge deck near pylon zone, the numerical method is applied to obtain more information under the same inflow condition. According to the flow field shown in Figure 8, the magnitude of wind speed first increases then decreases along the longitudinal direction from the center of the bridge pylon; the increase is abrupt, which is unfavorable for vehicle driving stability. Furthermore, the disturbance near the bridge pylon is obvious and the wind speed behind the bridge pylon is small at both height above the bridge deck. These are consistent with the experimental results. It is also obvious that the wind speed is generally larger at higher locations, which is caused by the disturbance of the bridge girder. The reverse flowing direction is not detected near girder edges at $0.25H$ above the deck in the numerical results, possibly because the simulation is a steady calculation while the experimental results are transient.

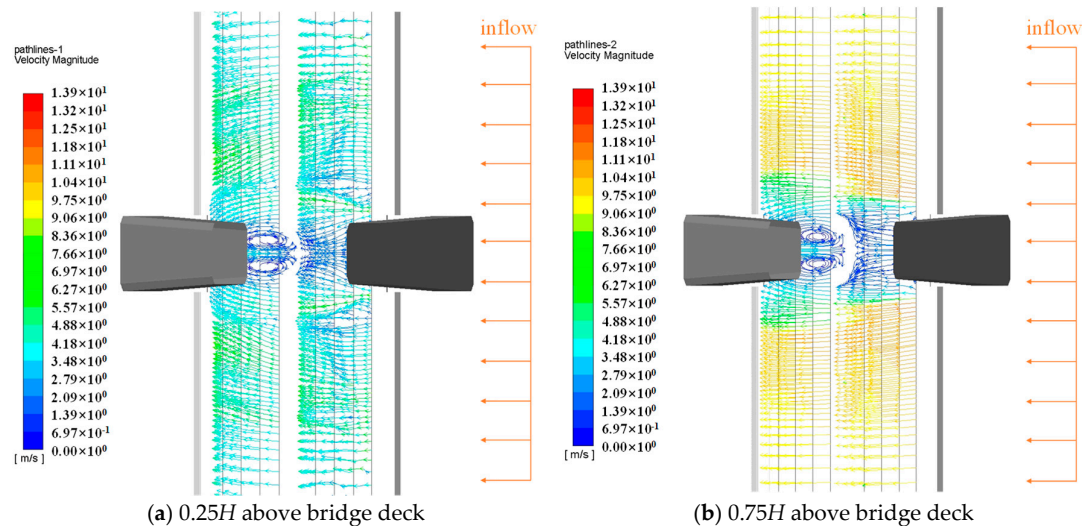


Figure 8. Flow field from numerical simulation results (C1 condition).

4.1.2. The Section without Any Ancillary Facility

In Figure 9, the flow pattern under the C2 condition is similar to that under the C1 condition. The tufts are not floating behind the pylon, which indicates that the wind velocity at this zone is very low. At $0.25H$ above the bridge deck, the tufts float in opposite directions in Zone I and Zone II, caused by different dominations of flow around the bridge deck or pylon. Moreover, at the higher elevation above the bridge deck, the flowing direction in Zone I is the same as that in Zone II, which illustrates the flow field at $0.75H$ is higher than the turbulent boundary layer.

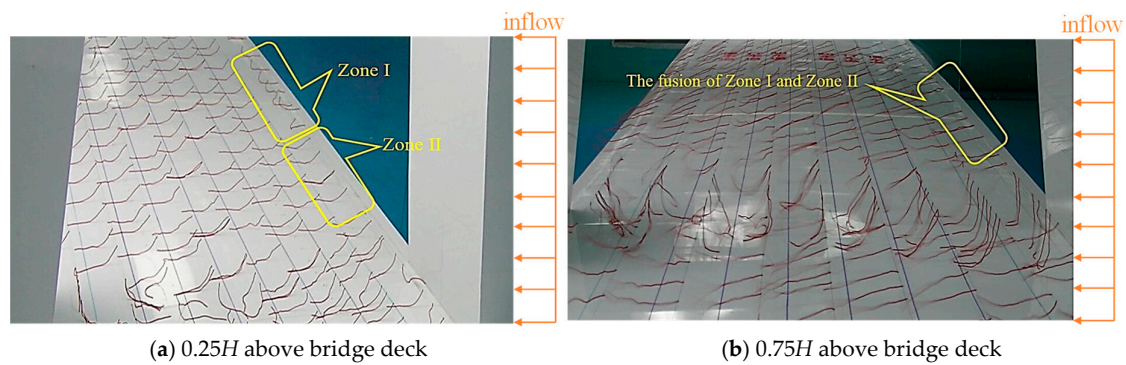


Figure 9. Flow field from experimental results (C2 condition).

The numerical results are shown in Figure 10; the wind velocity increases at first then decreases along the longitudinal direction from the center of the bridge pylon. The velocity magnitude scale clearly indicates that the wind speed between the two pylon columns is very low. In addition, the wind velocity is larger in higher locations. These are consistent with the experimental results and similar to the characteristics of flow field in the C1 condition. However, the reverse flow direction is not detected in the numerical results, and the reason could be that the numerical simulation is a steady calculation while the experimental result is transient.

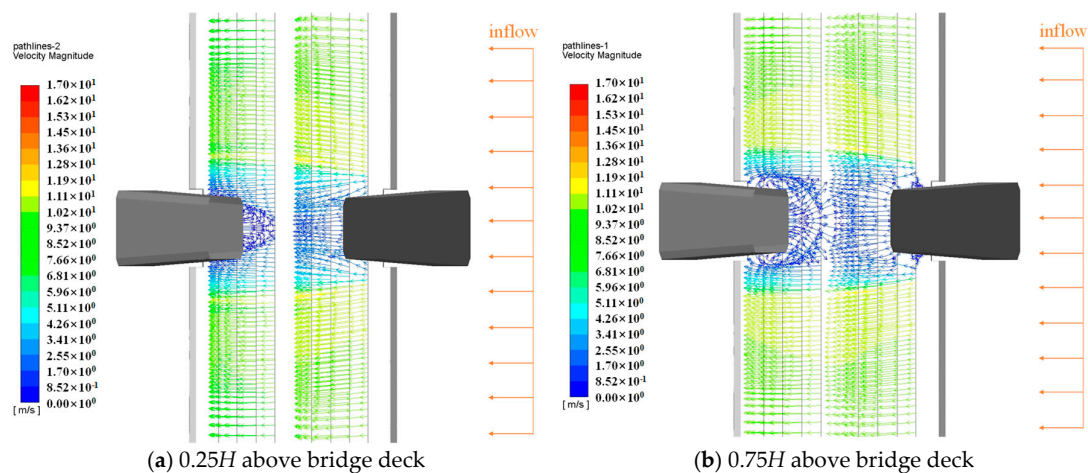


Figure 10. Numerical simulation results (C2 condition).

Compared with the result in the C1 condition, it is obvious that the overall magnitude is larger in the C2 condition, and this implies that the guardrails have a great influence on the wind environment above the bridge deck.

The vortex zone can be clearly observed in both experimental and numerical results shown in Figure 11, and flow pattern is very similar for both methods. This also demonstrates that the numerical simulation is reliable.

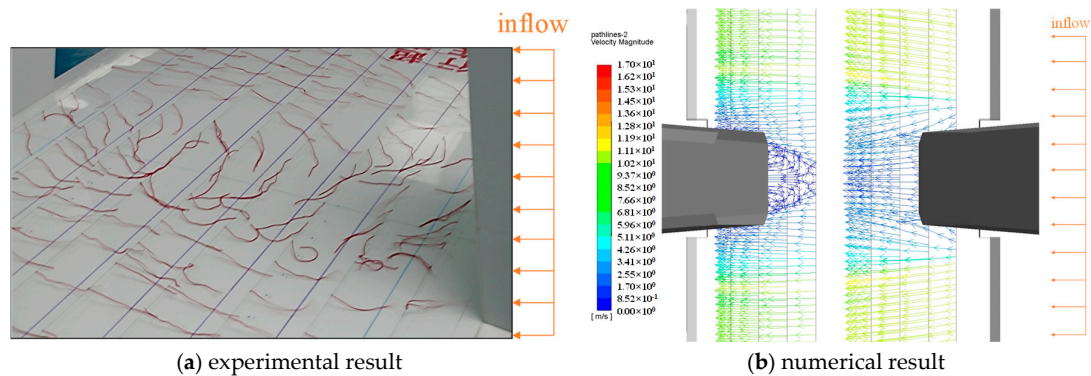


Figure 11. Vortex zone (at $0.25H$ above the deck, C2 condition).

4.2. Wind Speed Distribution at the Bridge Pylon Zone

4.2.1. Nondimensional Wind Speed Definition

To investigate the change in wind speed at different spatial points above the bridge deck, the nondimensional wind speed coefficient α is given by Equation (9) [29]:

$$\alpha = V_{mean} / V_{in}, \quad (9)$$

where V_{mean} is the mean wind speed at a spatial point and V_{in} is the inlet wind speed.

To compare the wind speed profile at different positions with the same criterion, equivalent wind speed V_{eff} and the influence coefficient λ_s are introduced. Equation (10) and Figure 12 performs the transformation of the boundary layer flow profile to an equivalent uniform profile, which is based on retaining the same magnitude of the side forces of the vehicle on the bridge. Since the wind load per unit area is proportional to the square of the wind speed, the square of the wind speed profile has to be integrated.

$$V_{eff} = \sqrt{\frac{1}{z_r} \int_0^{z_r} v^2(z) dz}, \quad (10)$$

where V_{eff} is the uniform equivalent wind speed, z_r is the height of the vehicle (usually the vehicle height is no more than 5 m, therefore, z_r is 5 m in this paper), and $v(z)$ is the absolute value of wind velocity vector at the height of z above the deck.

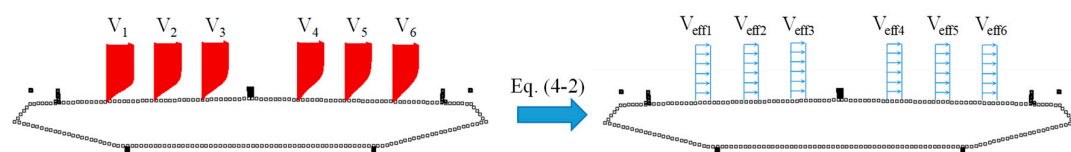


Figure 12. Boundary layer flow profile to equivalent uniform profile.

The influence coefficient λ_s is defined in Equation (11), which indicates the influence of the flow around the bridge pylon and girder on the wind environment above the bridge deck:

$$\lambda_s = V_{eff} / V_{in}. \quad (11)$$

4.2.2. Wind Speed Coefficient Result

The experimental and numerical results of the wind speed coefficient at the upstream and downstream main lanes under C1 and C2 conditions are shown in Figures 13–16. In these figures, the x-axis represents the wind speed coefficient, and the y-axis represents the distance in the bridge's longitudinal direction from the central of two pylon columns. For comparison, the mean wind speed at

locations 5 m above the upstream and downstream main lane, and 100 m in from the center of the pylon columns, is given. It is 8.68 m/s upstream and 8.67 m/s downstream in the C1 condition, and 8.60 m/s for both upstream and downstream in the C2 condition. The study on wind speed reveals that its distribution has a certain pattern along the longitudinal direction; the common details are as follows:

- (1) The wind speed coefficient increases at first, then decreases along the longitudinal direction, consistent with the analysis in the flow visualization section.
- (2) The wind speed coefficient is very small from the bridge pylon center to $0.5W$, and has a sudden increase from $0.5W$ to W , which is unfavorable for bridge driving safety. W represents the width of the bridge pylon of 15 m.
- (3) α tends to be static at a distance further than $3W$, and the largest α_{max} appears near the distance of W distant from the pylon column's center, which indicates that the flow around the pylon is dominant in this range.
- (4) The magnitude of α_{max} increases as the height above the bridge deck increases, which is caused by the boundary layer flow of the bridge deck.
- (5) α is a unimodal curve at higher locations above the bridge deck, which means that the flow around the bridge pylon is dominant at higher locations.
- (6) In the range of 0.8 to $1.2H$ above the bridge deck, α is basically coincident. In addition, the magnitudes of α_{max} are very close in the C1 and C2 conditions. This indicates that the wind environment within $0.8H$ is controlled by the flow around the bridge girder.
- (7) The numerical simulation result has the same trend and same location of α_{max} , and corresponds with the experimental result, though its value is generally larger than in the experimental results. A possible reason for this is that the wind speed is considered as a 3D resultant velocity in the numerical simulation, but as 1D along the inflow direction in the experimental results. If the measuring points are in the vortex zone, the velocity cannot be accurately obtained from the wind tunnel experiment.

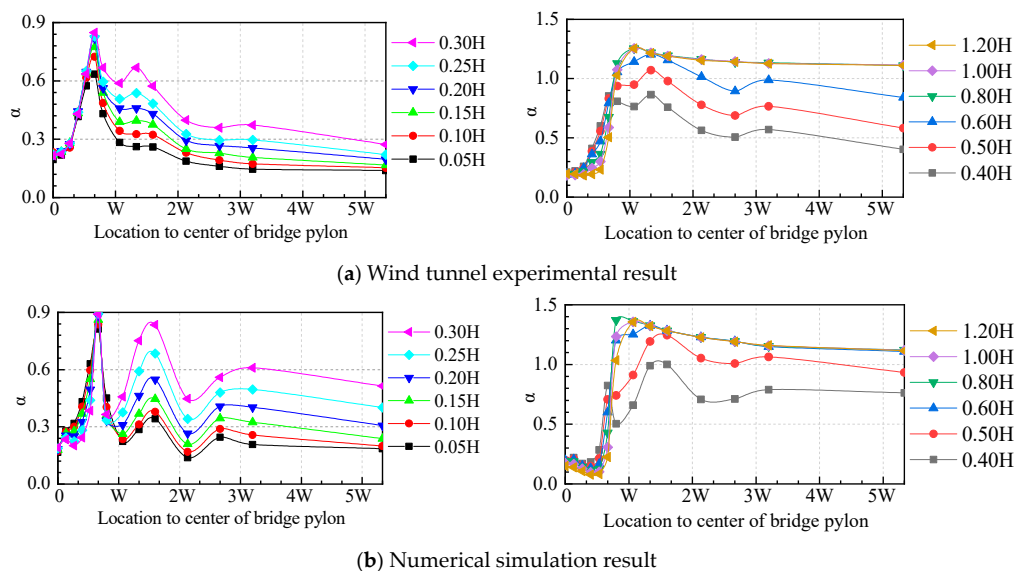


Figure 13. Wind speed coefficient at upstream main lane in C1 condition.

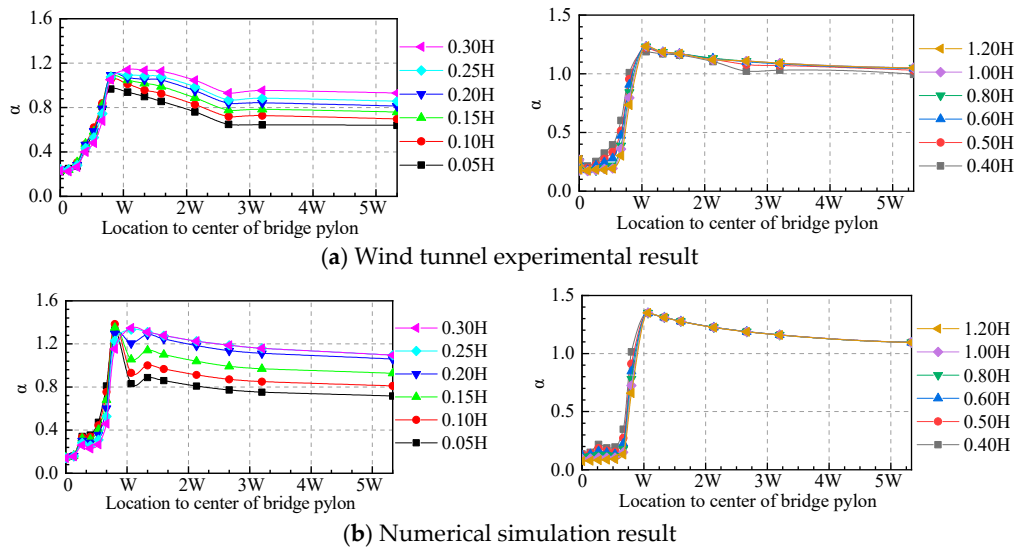


Figure 14. Wind speed coefficient at upstream main lane in C2 condition.

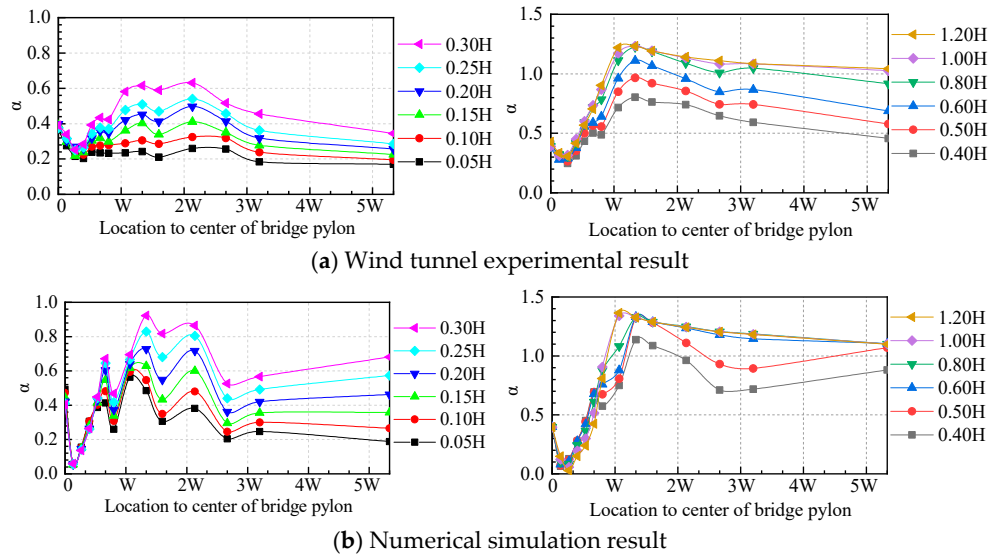


Figure 15. Wind speed coefficient at downstream main lane in C1 condition.

i. Upstream Main Lane

For both the numerical and experimental results, the pattern of the wind speed coefficient of the upstream main lane in the C1 condition can be summarized as follows:

- (1) α_{max} is located $0.67W$ from the center of the pylons, and within $0.3H$ above the deck (H is the girder height 4 m and W is the bridge pylon width 15 m at bridge deck elevation). The magnitudes of α within $0.3H$ are all smaller than the inflow velocity.
- (2) From 0.4 to $0.6H$ above the bridge deck, α_{max} appears at $1.33W$ from the center of the pylon columns.

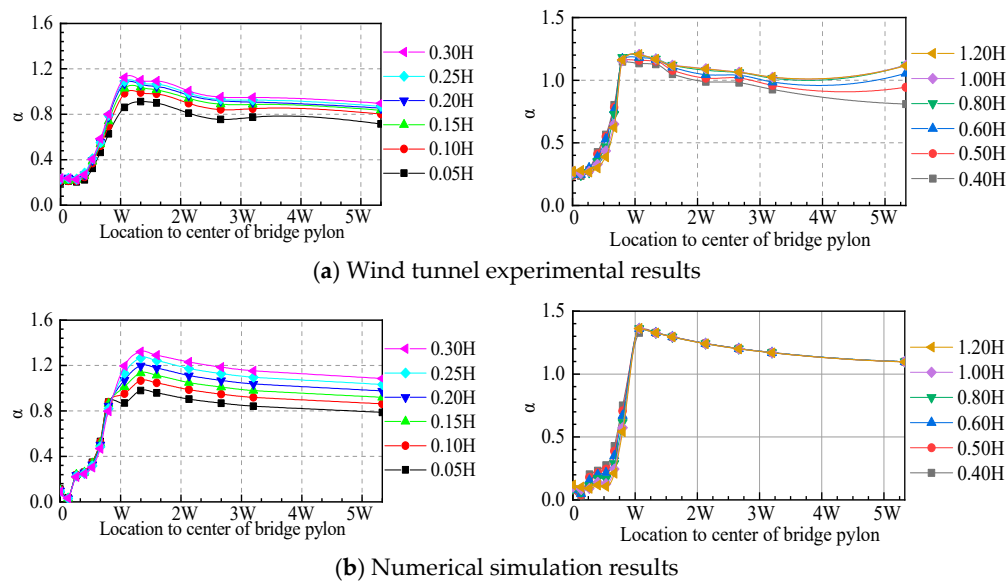


Figure 16. Wind speed coefficient at downstream main lane in C2 condition.

For the upstream main lane in C2 of the numerical and experimental results, similar conclusions can be drawn. The differences are listed as follows:

- (1) Under the C2 condition, α_{max} for all heights above the bridge deck of the upstream main lane is located near the distance of W from the center of pylon columns.
- (2) The magnitude of the wind speed coefficient within $0.3H$ above the deck is larger than that of the C1 condition, and the curve is more complicated in the C1 condition. In addition, for locations higher than $0.4H$ above the bridge deck, these curves are basically overlapping in the C2 condition, which occurs higher than $0.8H$ above the bridge deck in the C1 condition. The main reason is that the guardrails can disturb the inflow and cause airflow detouring.

ii. Downstream Main Lane

For the downstream main lane of both numerical and experimental results in the C1 condition, α_{max} for all heights above the bridge deck is located near $1.33W$ from the center of the pylons, and the magnitudes of α within $0.3H$ are all smaller than the inflow velocity.

For the downstream main lane in C2 of the numerical and experimental results, α_{max} for all heights above the bridge deck has a similar location as in the C1 condition. Similar to the comparison of C1 and C2 upstream, the magnitudes of α_{max} within $0.3H$ above the bridge deck are larger, and the α curves are unimodal in the C2 condition. In addition, at heights greater than $0.4H$ above the bridge deck, the curves are basically overlapping in the C2 condition, in contrast to the C1 condition, where this overlap appears at heights greater than $0.8H$ above the bridge deck. Comparing the wind speed coefficient α in Figures 13 and 14 (as well as Figures 15 and 16), the magnitudes of α are obviously larger in the C2 condition (without guardrails). This is caused by the shelter effect of the guardrail.

4.2.3. Influence Coefficient Result

Figure 17 shows a comparison of the results of the influence coefficient in the range of 5 m above the bridge along the longitudinal direction. The following conclusions can be obtained from the analysis:

- (1) The peak value of λ_s appears around the distance of W from the center of the pylon.
- (2) λ_s is very small from the pylon center to $0.5W$, and it has a sudden increase from $0.5W$ to W , which is unfavorable for bridge driving safety.

- (3) λ_s for distances larger than $3W$ tends to be static up to a certain value, which means that the wind environment is dominated by the flow around the pylon within the range of $3W$ from the pylon center, outside this range, the flow around the bridge girder is dominant.
- (4) Compared with the C2 condition, the peak value of λ_s in the C1 condition is obviously smaller. This implies that the guardrail and other ancillary facilities are helpful for reducing the wind speed above the deck.
- (5) The values of λ_s from the numerical simulation and the wind tunnel test are very close, and the trend of λ_s along the longitudinal direction is same for both methods. The maximum value of λ_s is generally larger in the numerical simulation, possibly for the same reason discussed with regard to the wind speed coefficients.

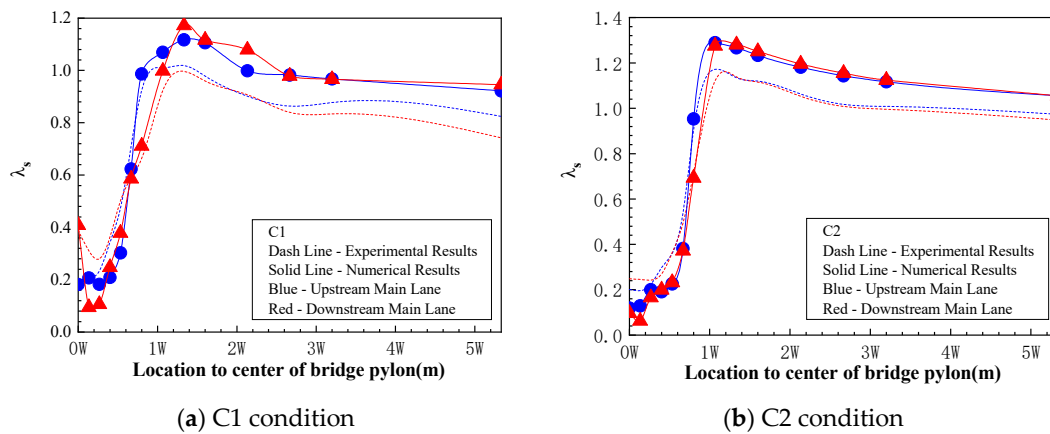


Figure 17. Comparison of influence coefficient for the experimental and numerical results.

5. Evaluation Method of Wind Barrier Arrangement

5.1. Evaluation Indicators Definitions

The characteristics of the flow field in the vicinity of the bridge pylon zone and the quantitative description both supply the basic information and data for setting up the evaluation method of wind barrier arrangement. In order to arrange the wind barrier in an effective way and improve driving safety near the bridge pylon zone, the influence coefficient ratio R and the safety factor of response time K are defined in Equations (12) and (13) to quantitatively evaluate the effect of wind barriers:

$$R = \lambda_{smax} / \lambda_{stand} \quad (12)$$

$$K = \frac{L_d / V}{T_s}, \quad (13)$$

where λ_{smax} is the maximum influence coefficient at the pylon zone and λ_{stand} is the standard influence coefficient far from pylon area. L_d is the distance along the axial direction from the center of the pylon to the point where λ_{smax} shows up, V is the vehicle limit speed on the bridge, and T_s is the reaction time for a driver to deal with emergencies.

R reflects the correlative relationship of wind speed between the area near and far from the pylon. If R is close to 1, it indicates that the wind velocity is almost the same both near and far from the pylon, which is conducive to maintaining straight driving. Therefore, a smaller R means a better effect of the wind barriers in reducing the wind speed.

K reflects the relationship between the time drivers have and the time drivers need to deal with the sudden change in wind speed. In previous experiments, the shortest reaction time for the driver to correct deflection after feeling a crosswind was 0.2 s. However, vehicles actually get the reaction from drivers 0.8 s later, since steering involves a change in gap, elastic force, and lateral force [30].

Considering the discrete and stochastic nature of reaction time, it is considered safe when K is larger than 2.5, which is a common number for the safety factor in many standards.

5.2. Evaluating Different Wind Barrier Arrangements near a Bridge Pylon Zone

According to the analysis in Section 4, the wind environment above bridge deck at the influencing zone of the pylon is unfavorable for vehicle stability under crosswinds. Setting wind barriers in a certain range around the pylon is an effective countermeasure to improve the wind environment above the bridge deck and the driving safety. To explore the improvement effect of wind barriers, three kinds of wind barriers were proposed. The cross section of wind barriers is a circular arc, and the barriers with the width of 220 mm are set in a curved shape vertically. The distance between the centers of the adjacent bars is 420 mm. Figure 18 shows the layout of the wind barriers. For all schemes, $H1 = 1.3$ m, $H2 = H3 = H4 = 0.84$ m. Details of L are shown in Table 1.

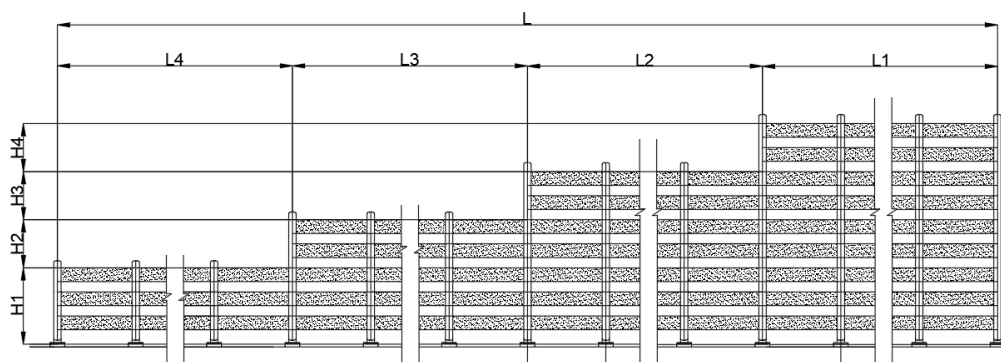


Figure 18. Wind barriers layout.

Table 1. The concrete size of wind barriers (unit: m).

	L^*	$L1$	$L2$	$L3$	$L4$
Scheme 1	25	7	6	6	6
Scheme 2	41	11	10	10	10
Scheme 3	57	15	14	14	14

where L^* is the total length outside the pylon; $L1$ is the length of nine-bar wind barriers area; $L2$ is the length of seven-bar wind barriers area; $L3$ is the length of five-bar wind barriers area; $L4$ is the length of three-bar wind barriers area.

The numerical simulation of wind speed above the bridge deck in the pylon zone under different conditions is carried out, and the influence coefficients at different special points are calculated. The distributions of the λ_s under different conditions are shown in Figure 19.

From Figure 19a–e, it can be seen that the wind speed increases significantly outside the pylon without wind barriers. After the wind barriers were installed, the velocity increase outside the pylon was effectively eliminated, and the changing gradient becomes more and more smooth.

The Sutong Bridge is designed for driving speeds of 100 km/h, with the speed limits for safe operation being 80 km/h on a normal day, and 60 km/h on a windy day. The results of K under different conditions are shown in Table 2. By analyzing the data in this table, it can be seen that K dramatically improves after the installation of wind barriers and that this increase is proportional to the wind barrier length. However, up to a certain threshold, additional lengthening of the wind barriers no longer generates any further effects. The values for K in Schemes 2 and 3 are larger than 2.5, which indicate a good wind environment for the operating stability of a vehicle in the crosswind. With all factors considered, Scheme 2 was adopted for Sutong Bridge, which is more economical, as Figure 20 shown. Figure 19 shows the influence coefficients in the longitudinal direction, and the influence coefficient ratio R is shown in Table 2. It can be seen that R is close to 1 in Schemes 2 and 3, which means these two schemes are optimal in terms of their wind barrier arrangement.

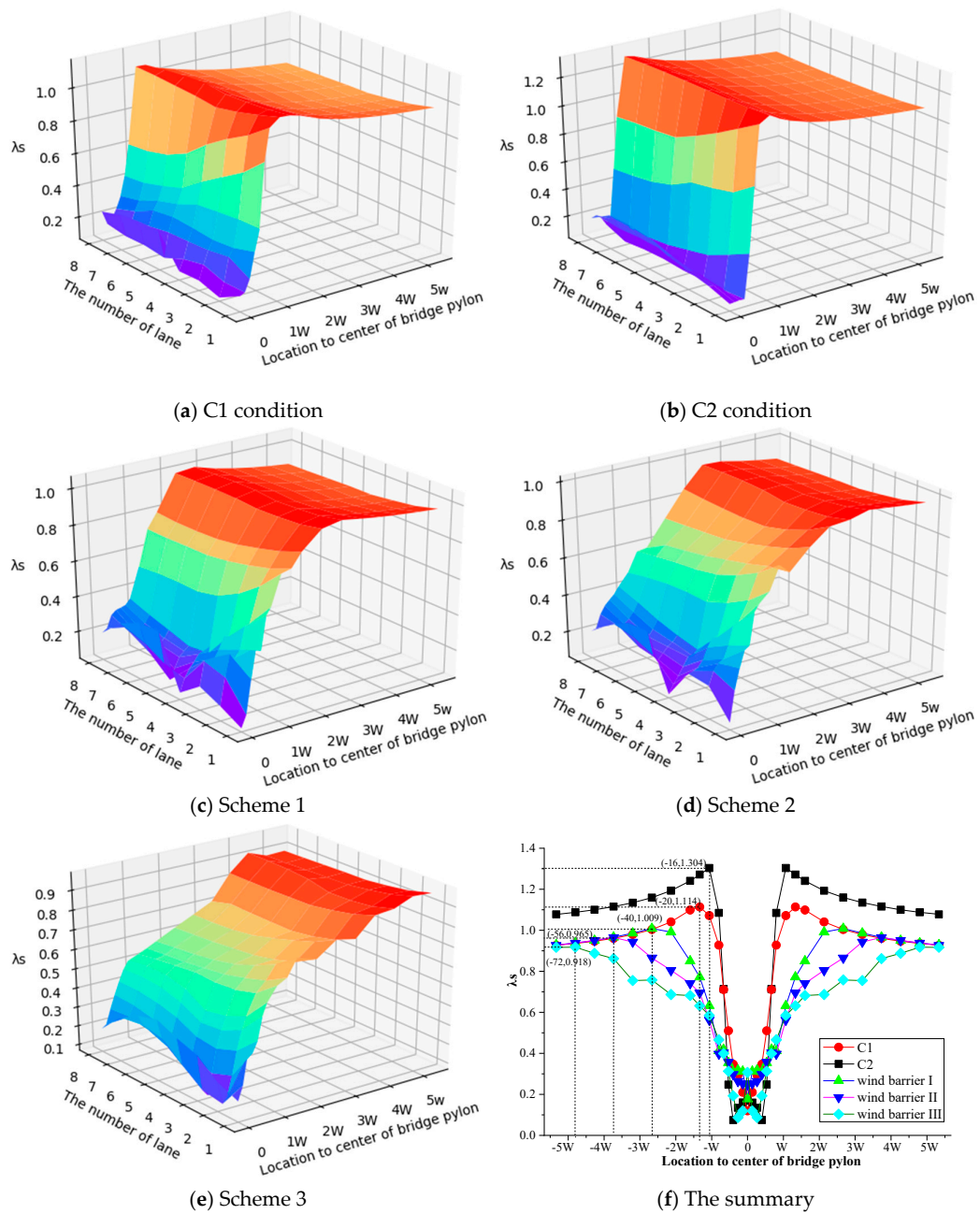
Figure 19. The influence coefficient λ_s .

Table 2. The concrete size of wind barriers.

Condition	λ_s	R	L_d (m)	K		
				V_1^*	V_2	V_3
C1	1.304	1.427	16.00	0.7	0.9	1.2
C2	1.114	1.219	20.00	0.9	1.1	1.5
Scheme 1	1.009	1.104	40.00	1.8	2.3	3.0
Scheme 2	0.965	1.056	56.00	2.5	3.2	4.2
Scheme 3	0.918	1.004	72.00	3.2	4.1	5.4

$V_1^* = 100$ km/h, $V_2 = 80$ km/h, $V_3 = 60$ km/h.



Figure 20. The final arrangement scheme of wind barriers on Sutong Bridge.

6. Conclusions

In order to explore the characteristics of the wind environment above the bridge deck near the pylon zone and guide the arrangement of wind barriers for improving driving safety in this zone, the flow visualization and wind speed near the inverted Y-shaped bridge pylon zone were studied. Combined with the wind tunnel test and numerical simulation, specific distribution patterns were found, and based on these characteristics, two safety evaluation indicators that consider vehicle driving stability were proposed. The main conclusions are summarized as follows:

- (1) The wind environment above the bridge deck near the pylon zone has the obvious characteristics of a typical three-dimensional flow field controlled by the flow around the pylon and girder. The influence range of the flow around the girder is dominant within 0.8 times the girder height in the vertical direction, and the flow around the bridge pylon is dominant within 3 times the pylon width in the longitudinal direction.
- (2) The wind velocity above the bridge deck is small behind the pylon columns and suddenly increases outside the pylon, and the maximum value appears near a distance equals to the pylon column width from the center of the bridge pylons. The sudden change caused by the flow around the pylon is unfavorable for driving safety under a crosswind. The wind tunnel experiment and the numerical simulation results both confirm the flow pattern and wind speed distribution. This distribution pattern of the wind environment can effectively guide the arrangement of wind barriers near the bridge pylon zone.
- (3) To effectively evaluate the wind barrier effects, this paper proposes a double-index evaluation method that contains the ratio of maximum influence coefficient R and the safety factor of response time K . With the wind distribution pattern and the proposed evaluation scheme, the optimal scheme for the wind barrier arrangement of Sutong Bridge can be determined, and the driving safety significantly improved.

The ongoing researches should be investigated considering the change of vehicle aerodynamic load influenced by the wake of the pylon column under crosswinds.

Author Contributions: Conceptualization, D.W. and A.C.; Data curation, D.W.; Formal analysis, Y.Z. and M.S.; Funding acquisition, D.W. and M.S.; Investigation, Y.Z.; Methodology, D.W. and M.S.; Project administration, A.C.; Resources, D.W.; Software, Y.Z. and M.S.; Supervision, A.C.; Validation, Y.Z., M.S. and A.C.; Visualization, Y.Z.; Writing – original draft, D.W. and M.S.; Writing – review & editing, D.W., Y.Z. and M.S. All authors have read and agreed to the published version of the manuscript.

Funding: This research was funded by the National Natural Science Foundation of China (grant number [51778472]) and the program of China Scholarships Council (grant number [201906260185]). The authors would also like to express their thanks to the Jiangsu Sutong Bridge Co., Ltd., for support and assistance during the experiments.

Conflicts of Interest: The authors declare no conflict of interest.

References

- Kim, S.J.; Kim, H.K. Feasibility of a Quasi-Static Approach in Assessing Side-Wind Hazards for Running Vehicles. *Appl. Sci.* **2019**, *9*, 3377. [\[CrossRef\]](#)
- Baker, C.J. The quantification of risk for road vehicles in cross winds. *J. Wind Eng. Ind. Aerodyn.* **1994**, *52*, 93–107. [\[CrossRef\]](#)
- Baker, C.J.; Reynolds, S. Wind-induced accidents of road vehicles. *Accid. Anal. Prev.* **1992**, *24*, 559–575. [\[CrossRef\]](#)
- Snaebjornsson, J.T.; Baker, C.J.; Sigbjornsson, R. Probabilistic assessment of road vehicle safety in windy environments. *J. Wind Eng. Ind. Aerodyn.* **2007**, *95*, 1445–1462. [\[CrossRef\]](#)
- Baker, C.J.; Humphreys, N.D. Assessment of the adequacy of various wind tunnel techniques to obtain aerodynamic data for ground vehicles in cross winds. *J. Wind Eng. Ind. Aerodyn.* **1996**, *60*, 49–68. [\[CrossRef\]](#)
- Zhu, L.D.; Li, L.; Xu, Y.L.; Zhu, Q. Wind tunnel investigations of aerodynamic coefficients of road vehicles on bridge deck. *J. Fluids Struct.* **2012**, *30*, 35–50. [\[CrossRef\]](#)
- Han, Y.; Cai, C.S.; Zhang, J.R.; Chen, S.R.; He, X.H. Effects of aerodynamic parameters on the dynamic responses of road vehicles and bridges under cross winds. *J. Wind Eng. Ind. Aerodyn.* **2014**, *134*, 78–95. [\[CrossRef\]](#)
- Han, Y.; Hu, J.X.; Cai, C.S.; Chen, Z.Q.; Li, C.G. Experimental and numerical studies of aerodynamic forces on vehicles and bridges. *Wind Struct.* **2013**, *17*, 163–184. [\[CrossRef\]](#)
- Salati, L.; Schito, P.; Rocchi, D.; Sabbioni, E. Aerodynamic Study on a Heavy Truck Passing by a Bridge Pylon under Crosswinds Using CFD. *J. Bridge Eng.* **2018**, *23*. [\[CrossRef\]](#)
- Cao, J.F.; Zhu, W.J.; Shen, W.Z.; Sorensen, J.N.; Wang, T.G. Development of a CFD-Based Wind Turbine Rotor Optimization Tool in Considering Wake Effects. *Appl. Sci.* **2018**, *8*, 1056. [\[CrossRef\]](#)
- Chen, W.L.; Gao, D.L.; Laima, S.J.; Li, H. A Field Investigation on Vortex-Induced Vibrations of Stay Cables in a Cable-Stayed Bridge. *Appl. Sci.* **2019**, *9*, 4556. [\[CrossRef\]](#)
- Huang, Z.W.; Li, Y.Z.; Hua, X.G.; Chen, Z.Q.; Wen, Q. Automatic Identification of Bridge Vortex-Induced Vibration Using Random Decrement Method. *Appl. Sci.* **2019**, *9*, 2049. [\[CrossRef\]](#)
- Argentini, T.; Ozkan, E.; Rocchi, D.; Rosa, L.; Zasso, A. Cross-wind effects on a vehicle crossing the wake of a bridge pylon. *J. Wind Eng. Ind. Aerodyn.* **2011**, *99*, 734–740. [\[CrossRef\]](#)
- Li, Y.L.; Hu, P.; Cai, C.S.; Zhang, M.J.; Qiang, S.Z. Wind Tunnel Study of a Sudden Change of Train Wind Loads due to the Wind Shielding Effects of Bridge Towers and Passing Trains. *J. Eng. Mech.* **2013**, *139*, 1249–1259. [\[CrossRef\]](#)
- Wang, D.; Chen, A.; Ma, R. Influence of wind barrier on aerodynamic characteristics of automobiles on bridge girder near pylon. *Eng. Mech.* **2013**, *30*, 244–250. (In Chinese)
- Wang, D.; Chen, A. Assessment of lateral driving stability of automobiles passing by the pylon zone under cross wind. *Res. J. Appl. Sci. Eng. Technol.* **2012**, *4*, 2735–2739.
- Bettle, J.; Holloway, A.G.L.; Venart, J.E.S. A computational study of the aerodynamic forces acting on a tractor-trailer vehicle on a bridge in cross-wind. *J. Wind Eng. Ind. Aerodyn.* **2003**, *91*, 573–592. [\[CrossRef\]](#)
- Zhang, N.; Ge, G.H.; Xia, H.; Li, X.Z. Dynamic analysis of coupled wind-train-bridge system considering tower shielding and triangular wind barriers. *Wind Struct.* **2015**, *21*, 311–329. [\[CrossRef\]](#)
- Su, Y.; Xiang, H.Y.; Fang, C.; Wang, L.; Li, Y.L. Wind tunnel tests on flow fields of full-scale railway wind barriers. *Wind Struct.* **2017**, *24*, 171–184. [\[CrossRef\]](#)
- Wu, X.X.; Zou, X.Y.; Zhang, C.L.; Wang, R.D.; Zhao, J.Y.; Zhang, J.Q. The effect of wind barriers on airflow in a wind tunnel. *J. Arid Environ.* **2013**, *97*, 73–83. [\[CrossRef\]](#)
- Liu, H.S.; Lei, J.Q.; Zhu, L. Identification and Application of the Aerodynamic Admittance Functions of a Double-Deck Truss Girder. *Appl. Sci.* **2019**, *9*, 1818. [\[CrossRef\]](#)
- Lo, K.H.; Sriram, R.; Kontis, K. Wake Flow Characteristics over an Articulated Lorry Model with/without AC-DBD Plasma Actuation. *Appl. Sci.* **2019**, *9*, 2426. [\[CrossRef\]](#)
- Zhang, L.Q.; Chen, G.B.; Chen, W.L.; Gao, D.L. Separation Control on a Bridge Box Girder Using a Bypass Passive Jet Flow. *Appl. Sci.* **2017**, *7*, 501. [\[CrossRef\]](#)
- Alonso-Estebanez, A.; Del Coz Diaz, J.J.; Rabanal, F.P.A.; Pascual-Munoz, P. Performance analysis of wind fence models when used for truck protection under crosswind through numerical modeling. *J. Wind Eng. Ind. Aerodyn.* **2017**, *168*, 20–31. [\[CrossRef\]](#)

25. Kozmar, H.; Procino, L.; Borsani, A.; Bartoli, G. Optimizing height and porosity of roadway wind barriers for viaducts and bridges. *Eng. Struct.* **2014**, *81*, 49–61. [[CrossRef](#)]
26. Telenta, M.; Duhovnik, J.; Kosel, F.; Sajn, V. Numerical and experimental study of the flow through a geometrically accurate porous wind barrier model. *J. Wind Eng. Ind. Aerodyn.* **2014**, *124*, 99–108. [[CrossRef](#)]
27. Anderson, J.D. *Computational Fluid Dynamics: The Basics with Applications*; McGraw-Hill, Inc.: New York, NY, USA, 1995.
28. Zhong, W.; Tang, H.W.; Wang, T.G.; Zhu, C.Y. Accurate RANS Simulation of Wind Turbine Stall by Turbulence Coefficient Calibration. *App. Sci.* **2018**, *8*, 1444. [[CrossRef](#)]
29. Kim, S.J.; Yoo, C.H.; Kim, H.K. Vulnerability assessment for the hazards of crosswinds when vehicles cross a bridge deck. *J. Wind Eng. Ind. Aerodyn.* **2016**, *156*, 62–71. [[CrossRef](#)]
30. Wang, D.L. Study on Wind-induced Driving Safety of Vehicle on Bridges. In *Civil Engineering*; Tongji University: Shanghai, China, 2010.



© 2020 by the authors. Licensee MDPI, Basel, Switzerland. This article is an open access article distributed under the terms and conditions of the Creative Commons Attribution (CC BY) license (<http://creativecommons.org/licenses/by/4.0/>).