

Determination of aerodynamic characteristics of a vehicle immersed in the wake of bridge tower using computational fluid dynamics

Bin Wang^a, You-Lin Xu^a, Le-Dong Zhu^b, Yong-Le Li^c

^a*Department of Civil and Structural Engineering, The Hong Kong Polytechnic University, Hong Kong, China*

^b*Department of Bridge Engineering, Tongji University, Shanghai, China*

^c*Department of Bridge Engineering, Southwest Jiaotong University, Chengdu, China*

ABSTRACT: The running safety analysis of a vehicle subjected to crosswinds is one of the issues concerned in wind engineering. Particularly as a vehicle moves passing by a bridge tower, the shielding effect of the tower on wind forces of the vehicle occurs, which leads to a sharp change of aerodynamic forces on the vehicle in danger. To explore the aerodynamic forces acting on the vehicle passing by the bridge tower, the flows around the vehicle immersed in different positions to the bridge tower are simulated using the computational fluid dynamics (CFD) technique in this study. The aerodynamic forces measured in the wind tunnel are used to confirm the simulation results. Meanwhile, the aerodynamic characteristics of the vehicle immersed in the wake of the bridge tower and the flow field features are explored in detail.

KEYWORDS: vehicle, bridge tower, crosswinds, aerodynamics, computational fluid dynamics

1 INTRODUCTION

Vehicle behavior under crosswinds is one of the issues concerned in wind engineering. The running safety analysis of a vehicle subjected to crosswinds probably started from the Baker's work in 1986 (Baker¹). Since then, several studies on the safety of a vehicle subjected to crosswinds have been carried out. Particularly as a vehicle moves passing by a bridge tower, the shielding effect of the tower on wind forces of the vehicle occurs, which leads to a sharp change of aerodynamic forces on the vehicle in danger. The safety of a vehicle running through a tower region is therefore concerned. The aerodynamic forces of a vehicle in the wake of a bridge tower should be studied to make sure the running safety of the vehicle. Wind tunnel tests are a common method to obtain the aerodynamic forces on a vehicle. Charuvisit et al.² tested the transient aerodynamic side force and yawing moment on a vehicle when it passed by a bridge tower subjected to crosswinds. Argentini et al.³ carried out wind tunnel tests to measure the aerodynamic forces and the surface pressure distributions of the stationary vehicle model. With the rapid development of computer hardware, computational fluid dynamics (CFD) can be employed as an alternative tool, practically in the flow visualization which cannot be achieved easily by wind tunnel tests. In this study, the flows around a bridge tower and a vehicle in different positions behind the bridge tower are simulated using the CFD technique. The aerodynamic forces measured in a wind tunnel are used to confirm the simulation results. Then, the aerodynamic characteristics of the vehicle immersed in the wake of the bridge tower and the flow field features are explored in detail.

2 MODELS

A typical high-sided articulated lorry, which was investigated extensively by Baker and his colleagues in wind tunnels and later used in the dynamic analysis of wind-vehicle-bridge systems (Xu & Guo⁴, Cai & Chen⁵, Chen & Wu⁶), is selected as the reference vehicle. The geometric scales of the vehicle are shown in Fig.1. A long span highway bridge is also selected. It is a cable-stayed bridge of double towers with a main span of 688m (see Fig.3). The tower composes of a pair of side by side legs and a transverse beam to support and restrain the deck (see Fig.2). The tower legs are inclined from the height of deck to both the ends with a chamfered rectangle section. The transverse beam connected the two legs with a rectangle cross section. A flat box girder with side fairing is selected as the bridge deck, as seen in Fig.4. The cross section of the deck was 34.0m wide and 3.5m high carrying a dual two-lane highway on the upper surface. Two lines of handrail, four lines of protection rail and two lines of I-shape maintenance trace are mounted on the bridge deck. Both the vehicle and bridge models are scaled with a length ratio of 1:25. Under the crosswinds, the vehicle running on the upwind first lane is impacted by the bridge tower more seriously than the one running on the other lanes. Therefore the vehicle is arranged on the upwind first lane in this study. Because of the complex flow field around the tower, the vehicle in different locations with reference to the tower may suffer distinct wind flow. To reflect the influence of distance from the tower, three different positions of the vehicle in the wake of bridge tower are considered. They are (1) the vehicle immersed behind the tower entirely (position Xd1 in Fig.5), (2) half-length of the vehicle immersed behind the tower (position Xd3 in Fig.5), and (3) the vehicle just getting rid of the tower (position Xd5 in Fig.5). Aerodynamic forces of the same vehicle model were tested in the TJ-3 wind tunnel of the State Key Laboratory for Disaster Reduction in Civil Engineering at Tongji University in Mainland China. The models mounted in the wind tunnel are shown in Fig.6.

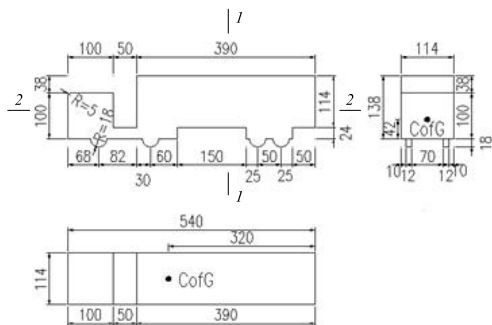


Figure 1. Dimensions of vehicle model (unit: mm)

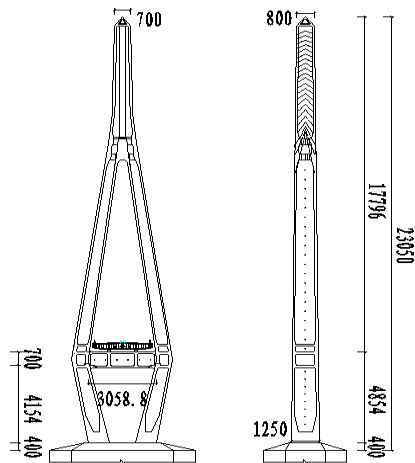


Figure 2. Dimensions of bridge tower (unit: cm)

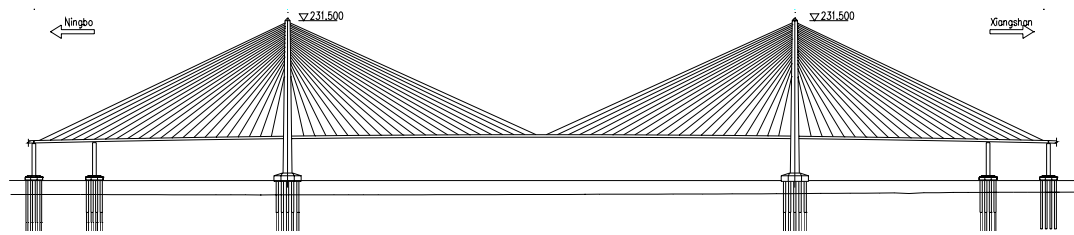


Figure 3. Elevation of the selected bridge (unit: cm)

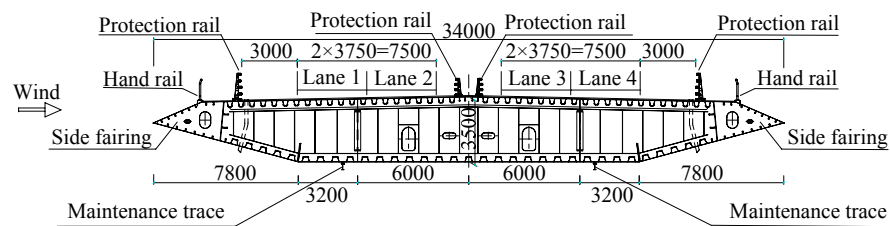


Figure 4. Cross section of bridge deck (unit: mm)

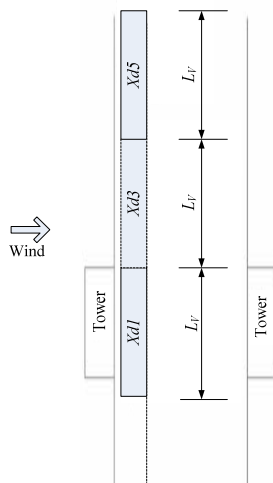


Figure 5. Vehicle locations



Figure 6. Wind tunnel tests of vehicle

3 NUMERICAL METHOD

3.1 Governing equations

In order to explore the flow characteristics around and the aerodynamic forces on the vehicle-bridge system, the large eddy simulation (LES) method is adopted for this study. In LES, turbulent flows are decomposed into large and small scales of turbulence by means of a high-pass filter in space. The large-scale turbulence flows can then be computed accurately by using the LES governing equations while the effect of the small-scale turbulence can be estimated through the subgrid-scale (SGS) model with the understanding that the small scales of turbulence tend to more homogeneous and universal, and less affected by the boundary conditions than the

large-scale ones. A box filter expressed by Eq. (1) is used for performing the spatial scale separation and a filtered variable is expressed by Eq.(2).

$$G(x, x') = \begin{cases} 1/V, & x' \in v \\ 0, & x' \notin v \end{cases} \quad (1)$$

$$\bar{\varphi}(x) = \int_v G(x, x') \varphi(x') dx' \quad (2)$$

where G is the filter function; φ is the variable before filtering; $\bar{\varphi}$ is the variable after filtering; v denotes the computational cell; and V is the computational volume. By applying the filtering operation to the momentum and continuity equations, the LES governing equations can be derived as

$$\rho \frac{\partial \bar{u}_i}{\partial t} + \rho \frac{\partial}{\partial x_j} (\bar{u}_i \bar{u}_j) = - \frac{\partial \bar{p}}{\partial x_i} + \mu \frac{\partial^2 \bar{u}_i}{\partial x_j \partial x_j} - \frac{\partial \tau_{ij}}{\partial x_j} \quad (3)$$

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \quad (4)$$

where $\bar{u}_i$ ($i=1, 2, 3$) is the filtered velocity component in the Cartesian coordinates; $\bar{p}$ is the filtered pressure; ρ is the fluid density; μ is the dynamic viscosity coefficient; and τ_{ij} is the subgrid-scale stress expressed by

$$\tau_{ij} = \rho \bar{u}_i \bar{u}_j - \rho \bar{u}_i \bar{u}_j \quad (5)$$

The subgrid-scale stress must be modeled, and the subgrid-scale turbulent model employed in this study is based on the Boussinesq hypothesis.

$$\tau_{ij} = -2\mu_t \bar{S}_{ij} + \frac{1}{3} \tau_{kk} \delta_{ij} \quad (6)$$

$$\bar{S}_{ij} = \frac{1}{2} \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \quad (7)$$

where μ_t is the subgrid-scale turbulence viscosity; τ_{kk} is the isotropic part of the subgrid-scale stress. The subgrid-scale turbulence viscosity can be determined by using the Smagorinsky-Lilly model. The Smagorinsky constant equal to 0.1.

3.2 Numerical models

In the simulation, only one wind direction (wind perpendicular to the bridge deck) is considered. Fig.7 shows the computational flow domain when the vehicle is located at Xd3. The entire computational domain is a box shape enclosed by six boundary faces. B, BV and BT represent the width of the deck, the length of the vehicle and the width of tower, respectively. The height of the computational domain is 7m which leads a blockage ratio at the level of the wind tunnel

tests. All those sizes determining the computational flow domain are optimized through parameter studies.

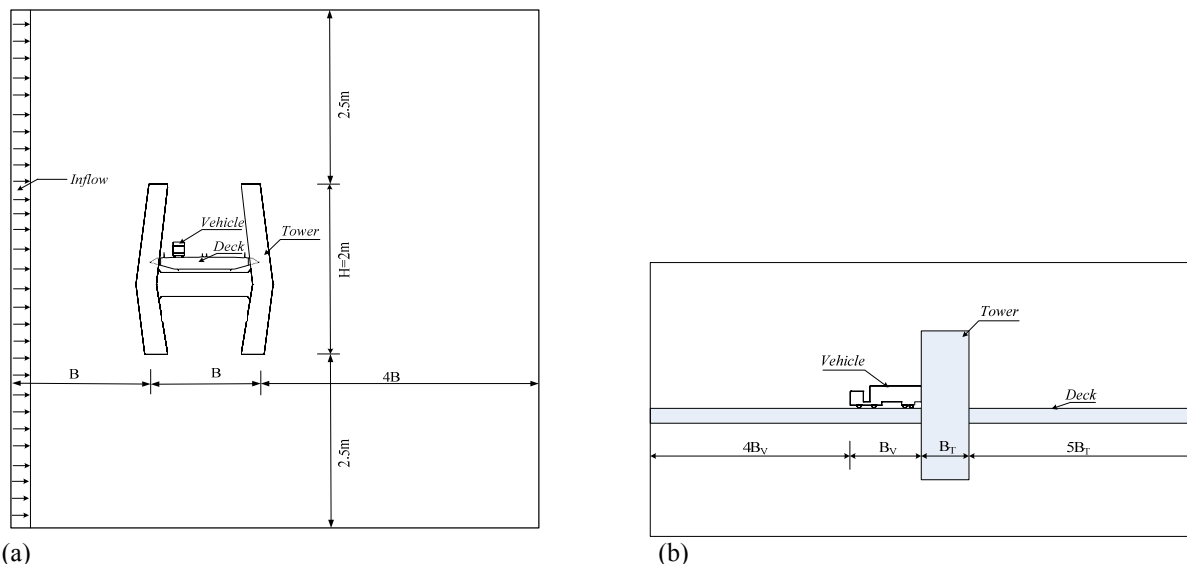


Figure 7. Computational domain sketch: vehicle-bridge system

3.3 Boundary conditions

The input boundary face is set as flow inlet with a uniform velocity of 10m/s, which leads to a Reynolds number of 1.13×10^5 in terms of the height of the vehicle. The output boundary face is specified as flow outlet with zero pressure. The other boundaries are all defined in such a way that the gradients of flow variables (including velocity and pressure) normal to those boundary faces are zero. The surfaces of the vehicle and the bridge are all modeled as nonslip wall boundaries.

3.4 Meshing

The characteristic sizes of components of the vehicle-deck-tower system vary from the size of rails to the size of tower greatly. The complex geometric shape and the large variation of characteristic size lead to a complicated flow field around the system in space. In order to obtain a relative adequate spatial discretization grid system corresponding to the ability of the current using PC, the grid optimization is achieved. Since the simulation with the LES method is time-consuming for the flow around three-dimensional complex geometric body, steady RANS equations enclosed with SST two-equation turbulence model are solved during the process of grid optimization.

3.5 Computational scheme

The LES governing equations are discretized using a three-dimensional finite volume method. The convective and diffusive terms are approximated by the bounded central differences of second-order accuracy and central difference respectively. The time integration is performed using the second-order implicit method. Pressure and velocity are solved with the SIMPLEC algorithm.

4 SIMULATION RESULTS

4.1 Aerodynamic forces and coefficients

The numerical simulation provides the pressure distribution over the surfaces of the vehicle. The aerodynamic forces acting on the vehicle can then be acquired by integrating the pressures over the surface. The aerodynamic forces and coefficients discussed below refer to the mean values. There are six aerodynamic force components on the vehicle in the Cartesian coordinate system: lift force F_L , drag force F_D , side force F_S , pitching moment M_P , yawing moment M_Y and rolling moment M_R , as shown in Fig.8. The non-dimensional aerodynamic coefficients are defined by

$$C_L = \frac{F_L}{qA}, \quad C_D = \frac{F_D}{qA}, \quad C_S = \frac{F_S}{qA} \quad (8)$$

$$C_P = \frac{M_P}{qAL}, \quad C_Y = \frac{M_Y}{qAL}, \quad C_R = \frac{M_R}{qAL} \quad (9)$$

$$q = 0.5\rho U^2 \quad (10)$$

where ρ is the air density; U represents the mean wind speed at the inlet boundary of the computational domain; A is the frontal project area of the vehicle without wheels and it refers to the project area in the X-Y plane in this study as shown in Fig.8; and L represents the maximum length of the vehicle in the Z-Y plane as shown in Fig.8. The aerodynamic force coefficients of the vehicle at different positions are listed in Table 1 for wind perpendicular to the length of the vehicle. From the comparative results of aerodynamic forces between the simulation and the wind tunnel tests, it can be seen that the simulated aerodynamic coefficients agree well with the experiment in general. The simulated coefficients demonstrate the same trend with the change of the vehicle location as the wind tunnel tests. The lift force coefficients are sensitive to the experimental conditions (Coleman & Baker⁷) and the calculated lift force coefficients are a slightly larger than the test values for all three locations. Compared with the test results, the calculated side force coefficients are a slightly larger while the calculated pitching moment coefficients are lower than the test results. When the vehicle moves out of the shielding area of the tower (from Xd1, Xd3 to Xd5), its side force, drag force, pitching and rolling moment all increase while its lift force decreases. The yawing moment decreases first and increases late.

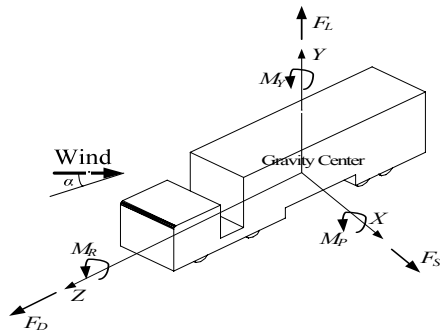


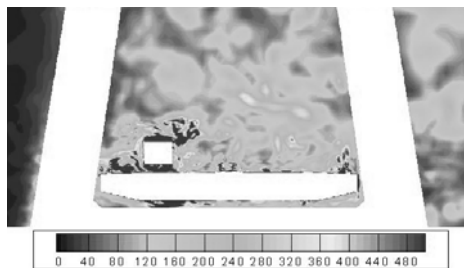
Figure 8. Definitions of aerodynamic forces

Table 1. Aerodynamic Coefficients of Vehicle

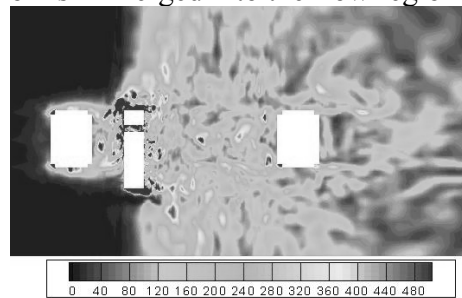
	Xd1		Xd3		Xd5	
	Experiment	LES	Experiment	LES	Experiment	LES
C_S	1.145	1.840	4.120	4.857	4.494	5.260
C_L	0.112	0.650	-0.939	-0.360	-0.837	-0.334
C_D	-0.136	-0.369	0.499	0.409	0.490	0.327
C_P	-0.062	0.039	-0.225	-0.217	-0.248	-0.262
C_Y	-0.670	-0.799	-0.371	-0.655	-0.473	-0.781
C_R	-0.129	-0.105	-0.251	-0.236	-0.236	-0.247

4.2 Flow fields

The visualization ability of CFD simulation is used here to analyze the flow fields around the vehicle. The instantaneous flow structures in terms of vorticity magnitude and the averaging flow structures in terms of projected streamlines and velocity contours are included. Sections shown as 1-1 and 2-2 in Fig.1 are selected to present the characteristic flow structures around the vehicle. The instantaneous contours of vorticity magnitude around the vehicle at different locations from the tower in the selected sections are shown in Fig.9, Fig.10 and Fig.11. Influenced by the tower, the flow environment becomes more and more complex from Xd5 to Xd1. At Xd5, the vortex street in the wake of vehicle-deck is disturbed slightly by the tower. At Xd3, the vortex street in the wake of vehicle-deck is interfered by the flow around the tower obviously. At Xd1, the selected vehicle section is immersed into the flow region of tower totally.

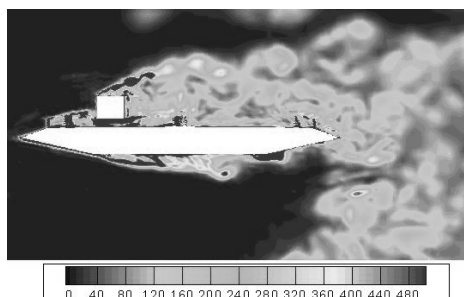


(a) section 1-1

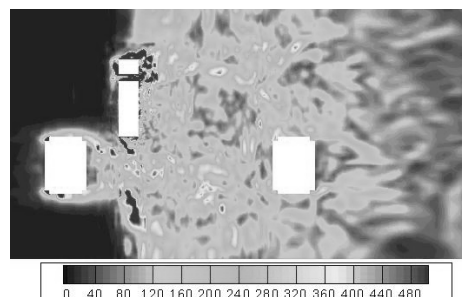


(b) section 2-2

Figure 9. Contours of instantaneous vorticity magnitude for vehicle at Xd1 (unit: s-1)

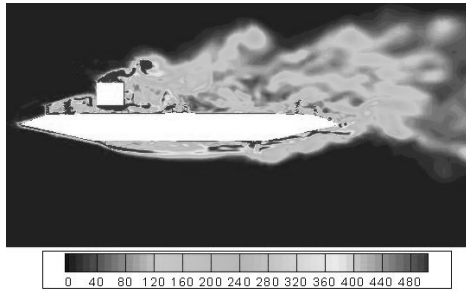


(a) section 1-1

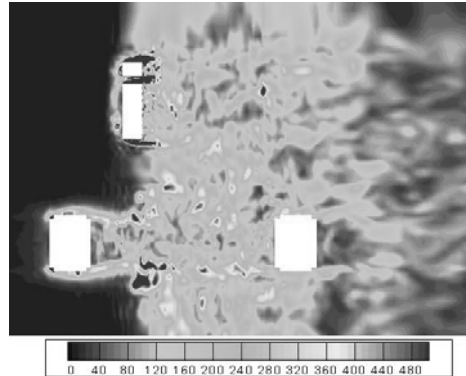


(b) section 2-2

Figure 10. Contours of instantaneous vorticity magnitude for vehicle at Xd3 (unit: s-1)



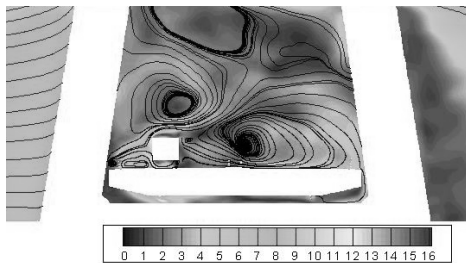
(a) section 1-1



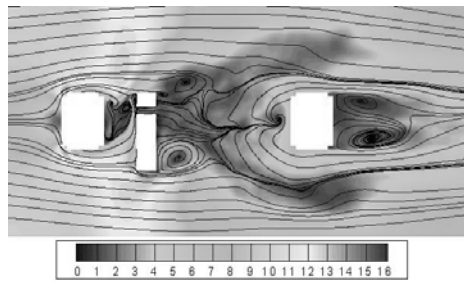
(b) section 2-2

Figure 11. Contours of instantaneous vorticity magnitude for vehicle at Xd5 (unit: s-1)

The averaging projected streamlines and velocity contours of the vehicle at different locations are displayed in Fig.13, Fig.14 and Fig.15. Different flow structures are formed. Flow separates at the upwind tower leg. At Xd1, the vehicle is almost totally immersed in the separating wake of the upwind tower leg while the separated flow from the tower pushes on the tail part of the vehicle. The symmetric flow feature around the tower is disturbed slightly by the vehicle. At Xd3, the accelerated flow around the upwind tower leg pushes to the vehicle. The separated flow around the vehicle in Section 1-1 reattaches to the bridge deck again. At Xd5, the vehicle nearly gets rid of the impact of the wake of the tower. The separated flow around the vehicle in Section 1-1 does not reattach to the bridge deck.

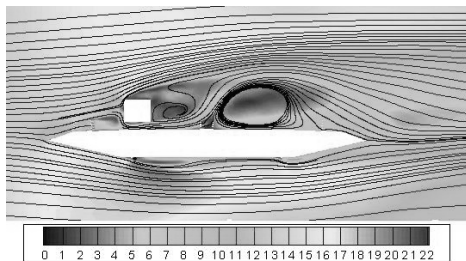


(a) section 1-1

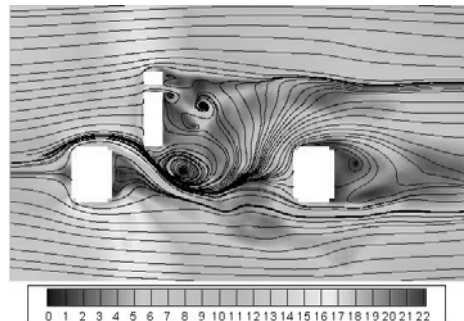


(b) section 2-2

Figure 12. Projected streamlines and velocity contours for vehicle at Xd1 (unit: m/s)



(a) section 1-1



(b) section 2-2

Figure 13. Projected streamlines and velocity contours for vehicle at Xd3 (unit: m/s)

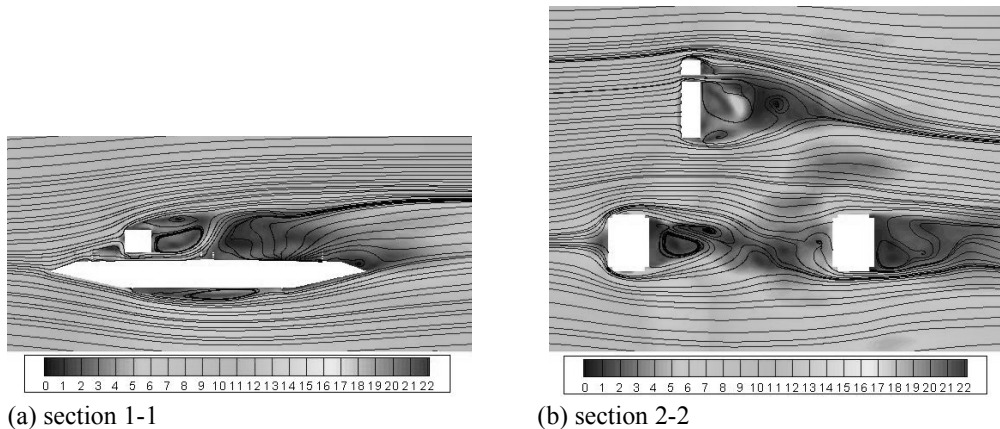


Figure 14. Projected streamlines and velocity contours for vehicle at Xd5 (unit: m/s)

4.3 Surface pressure distributions on vehicle

The mean pressure distributions over the surfaces of the vehicle under the three conditions are computed and displayed in Fig.16 for the case of wind perpendicular to the vehicle. The flow acts on the upwind side surface of the vehicle directly and some stagnation areas are formed. All the surfaces except the upwind one are in the flow separate region of the vehicle with negative pressure due to the flow separations around these surfaces. Compared with Xd5, the stagnation area for Xd3 moves to the head part while the area for Xd1 focuses on the tail part of the vehicle.



Figure 15. Mean pressure distribution on vehicle (unit: Pa)

5 CONCLUSIONS

By using large eddy simulation method, the aerodynamic characteristics of a vehicle across a bridge tower have been investigated. The simulated aerodynamic coefficients agree with the experimental results in general. The simulated coefficients demonstrate the same trend with the change of the vehicle location as the wind tunnel tests. When the vehicle moves out of the shielding area of the tower, its side force, drag force, pitching and rolling moment increase while its lift force decreases. The yawing moment decreases first and increases later. The instantaneous contours of vorticity magnitude show a complex flow environment around the vehicle due to the tower. The averaging projected streamlines and velocity contours show different flow separation phenomena around the vehicle at different locations. The flow acts on the upwind side surface of the vehicle directly and some stagnation areas are formed. All the surfaces except the upwind one are in the flow separate region of the vehicle with negative pressures.

6 ACKNOWLEDGEMENTS

The authors wish to acknowledge the financial supports from the Research Grants Council of the Hong Kong (PolyU 5311/07E), The Hong Kong Polytechnic University through a PhD studentship, and the State Key Laboratory for Disaster Reduction in Civil Engineering of China (2006-A-01).

7 REFERENCES

- 1 C. J. Baker, A simplified analysis of various types of wind-induced road vehicle accidents, *J. Wind Eng. Ind. Aerodyn*, 22(1986)69-85.
- 2 S. Charuvisit, K. Kimura, Y. Fujino, Experimental and semi-analytical studies on the aerodynamic forces acting on a vehicle passing through the wake of a bridge tower in cross wind, *J. Wind Eng. Ind. Aerodyn*, 92(2004) 749-780.
- 3 T. Argentini et al., Cross-wind effects on a vehicle crossing the wake of a bridge pylon, *J. Wind Eng. Ind. Aerodyn*, 99(2011) 734-740.
- 4 Y. L. Xu, W. H. Guo, Dynamic analysis of coupled road vehicle and cable-stayed bridge systems under turbulent wind, *Eng. Struct*, 25(2003)473-486.
- 5 C. S. Cai, S. R. Chen, Framework of vehicle-bridge-wind dynamic analysis, *J. Wind Eng. Ind. Aerodyn*, 92(2004)579-607.
- 6 S. R. Chen, J. Wu, Performance enhancement of bridge infrastructure systems: Long-span bridge, moving trucks and wind with tuned mass dampers, *Eng. Struct*, 30(2008)3316-3324.
- 7 S. Coleman, C. Baker, An experimental study of the aerodynamic behaviour of high sided lorries in cross winds, *J. Wind Eng. Ind. Aerodyn*, 53(1994) 401-429.