

# Research on feasibility of computational fluid dynamics (CFD) method for traffic signs board calculation

S Chao<sup>1</sup>, C W Jiao and S Liu

Research Institute of Highway Ministry of Transport, 8 Xitucheng Rd. Haidian Dist., Beijing, China

E-mail: z.chao@rioh.cn

**Abstract.** At this stage of the development of China's highway, the quantity and size of traffic signs are growing with the guiding information increasing. In this paper, a calculation method is provided for special sign board with reducing wind load measures to save construction materials and cost. The empirical model widely used in China is introduced for normal sign structure design. After that, this paper shows a computational fluid dynamics method, which can calculate both normal and special sign structures. These two methods are compared and analyzed with examples to ensure the applicability and feasibility of CFD method.

## 1. Background

In recent years, with the continuous development of China's highway, the investment of traffic signs is huge to ensure the safety of the highway [1]. The study on low wind load traffic sign board can reduce the sign board support structure and materials, and greatly reduce the production cost of signs. However, the wind load calculation model for signs in China national standard can only be used for typical sign board, which cannot meet the calculation requirements for special sign boards, which take measures (including punching, slotting etc.) to reduce the wind load. In this paper, CFD simulation model is provided to calculate the wind load on sign board. It will be provided as a basis for traffic signs structure design to reduce construction cost. This method will have wide application prospect.

## 2. Calculation method of wind load on traffic signs

Wind load is the main external load of traffic sign structure. The components of the structure, e.g. the sign board, the post and the crossbeam exposure to the sign board, are all under the wind load. Because of its large area, the largest wind load is on the board of traffic sign [2][3]. This study mainly refers to the wind load on the traffic sign board.

At present, there are two methods for calculating the structure of traffic signs normally [4]. One is application of structural mechanics and mechanics of materials, simplifying the structure into a thin walled bar system, and using limit state design method to check the design. The other one is approximate calculation method of finite element. The first method is widely used in real design because of rigorous and simple. There two methods considering wind load are both based on the static for the method that the wind load are distributed on board of traffic sign [5].

<sup>1</sup> Address for correspondence: Research Institute of Highway Ministry of Transport, 8 Xitucheng Rd. Haidian Dist., Beijing, China. Email: z.chao@rioh.cn.



The calculation formula of the wind load on traffic signs based on design method based on limit state is provided in National Standards GB5768 [6]:

$$F_{wb} = \frac{1}{2} \gamma_0 \gamma_Q \rho C V^2 A$$

$F_{wb}$  (unit N) is the wind load on the sign board.  $\gamma_0$  is structure importance coefficient, equals 1.0, because traffic signs structure security level refers to level 2.  $\gamma_Q$  is variable load factor, normally equals 1.4.  $\rho$  is air density, commonly equals  $1.2258 \text{ N} \cdot \text{s}^2 \cdot \text{m}^{-4}$ .  $C$  is coefficient of wind load, equals 1.2 for traffic sign board.  $V$  (unit m/s) is wind speed.  $A$  is the area toward the wind.

In recent years, with the development of CFD and its software, the relevant research on the wind load on traffic signs has been analyzed using CFD. In 2008, a research on the method of determining the wind load on signs and traffic signal structures was carried out in the University of Iowa. In this research report, the normal calculation method for signs wind load in relevant standards is static design method considering only static wind load, is only for isolated rectangular board. But this method is not suitable for the adjacent and complex structure sign boards. The CFD method is applied in this research to assess the total wind load on sign boards and its actual distribution on, in front and behind the board. The research focus on the aspect ratio and spacing of sign board, the thickness of variable information sign, back-to-back signs, add-on exit sign, the wind tunnel effect in the formation of trucks underneath the signs potentially by using CFD. These research findings have affirmed the necessity of applying CFD to study on the wind load on traffic sign board [7].

### 3. Computational Fluid Dynamics (CFD) overview

CFD is the product of the combination of modern fluid mechanics, numerical mathematics and computer science [8]. It can apply various discrete mathematical methods by using computers to do Numerical experiments, simulation, and analysis on various problems in fluid mechanics [9][10].

Basic features of CFD are numerical simulation [11] and computer experiment. It has greatly replaced the expensive experimental equipment for fluid dynamics based on the basic physics theorem. Therefore, it has a tremendous impact on scientific research and engineering technology. At present, CFD is widely used in many engineering fields, e.g. space design, automobile design, biomedical industry, chemical industry, turbine design, etc.. CFD method has the advantages of low cost and can be simulated more complex or more ideal process, etc.. After a certain assessment, CFD software can broaden the scope of the experimental study, and reduce the cost of expensive experimental work. Using a computer to simulate phenomenon under given parameters is equivalent to a numerical experiment. CFD software is normally able to achieve a variety of optimization of the physical model, e.g. steady and unsteady flow, laminar flow, turbulent flow, incompressible and compressible flow, heat transfer, chemical reaction, etc.. CFD software can be used to find a suitable numerical solution according to the flow characteristics of each physical problem. Furthermore, it can achieve the best in terms of computational speed, stability and precision [12].

### 4. Problems and computing software selection

The fluid involved in the wind load on the sign board problem is the air that can be simplified as steady state, steady, compressible viscous fluid. This type of fluid can be solved using most of the main fluid mechanics software. But the wind load on the sign board problem has three following characteristics large calculation area, special shape of computing object and huge amount of computational grids [13][14].

For the first characteristic, in practical application traffic signs are always in open space where very open around and without structure higher than signs. For this kind of practical application simulation, the calculation area must be set large enough to avoid similar situation of pipe flow, and to avoid the influence of calculation regional boundary on the fluid motion around traffic signs. In addition, the flow field variation must be fully grasped around traffic signs.

For the second one, traffic signs have a specific shape, whose prominent feature is the characteristics of thin board. In other words, the sign board area is large, whose length and height can

reach a few meters, but its thickness is quite thin. In China National Standard, the minimum thickness of the aluminum alloy board, thin steel board and synthetic resin board are 1.5mm, 1.0mm and 3.0mm. In practical use, the thickness of the sign board is generally no more than 5mm. Therefore, the area and the thickness of the signs are nearly 106 times.

For the last characteristic, the grids division of CFD simulation calculation for traffic signs can both reflect close to the sign board of the smaller size (mm level), but also can fill the whole simulation area (10m level). If all the grids are divided by the mm level, and the computational area is calculated according to the length of the 10m cube, the number of the mesh is 109. This order of magnitude grids number cannot be calculated by any existing commercial CFD software.

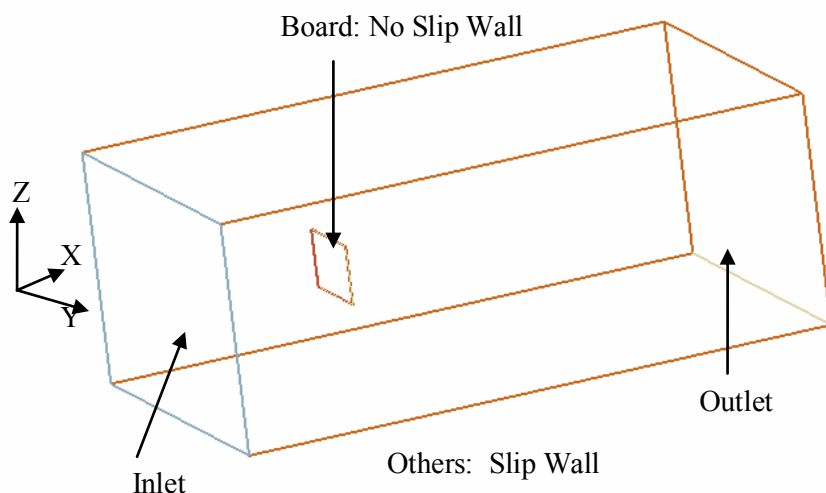
All above, CFD analysis of wind load on traffic sign board is faced with a huge amount of grids. There are two ways to solve this problem. One is to meet the computational area size, while to reduce the number of computing grids to the acceptable range. The other is to use large grids processing capacity of the CFD software. Accordingly, the CFX simulation software (from Ansys co.) was chosen in this study, and the icem-CFD in its software package was used as a grids partition tool.

Computational speed and stability are improved comparing the traditional method, because fully implicit multi grids coupled solution technique is used in CFX software. And because its convergence rate is fast, which is faster than other fluid software 1-2 magnitudes in the same conditions, it can calculate a million grids model.

Icem-CFD can generate a boundary layer mesh. According to the principle of fluid mechanics, dense mesh is generated in a certain region close to the simulation object to describe the details of the model and fluid. The gradually sparse mesh is generated in the calculation area far from the model. Thus, the number of the grids is reduced in the region.

## 5. Computational area and boundary conditions

According to the actual environmental characteristics of the sign setting, the computational area and boundary conditions of CFD simulation were set up. The size of the sign board is 6m length, 4m width and 0.003m thickness. For minimizing the number of grids, the fluid region is shown in Figure 1.



**Figure 1.** Computational area and boundary conditions from CFD software

In this condition,  $X = -12 \sim 30\text{m}$ , which is in front of and behind sign board 12m and 30m.  $Y = -12 \sim 12\text{m}$ , which is 12m on the left and right side of the median line of the sign board.  $Z = -8 \sim 8\text{m}$ , which is 5m and 7m from board lower and upper edge to bottom and top of computational region, and 4m high sign board. For fluid flow characteristics, considering the calculation of wind load on traffic signs the Reynolds number is  $Re = \rho VL/\mu$ . In the formula,  $\rho = 1.2258\text{N} \cdot \text{s}^2 \cdot \text{m}^{-4}$  stands for air density.  $V$

stands for the calculation wind speeds which are 20m/s, 25 m/s, 30 m/s, 35 m/s, 40 m/s. Feature length  $L$  is 6m the longest side of the sign board.  $\mu=17.9\times10^{-6}$  Pa·s is air dynamic viscosity. After that,  $Re$  is  $9.6\times10^6\sim1.92\times10^7$  in this condition. For the boundary conditions, the computational area import is speed imports, which are 20m/s, 25m/s, 30m/s, 35m/s, 40m/s. The export is free export, with 0Pa pressure. The four wall boundaries in the calculation area are all sliding wall boundaries. The effect of the air flow is neglected, because the calculation of CFD simulation is for cantilever or framed sign as an example whose height is 5m from board lower edge to bottom of computational region. In the test phase, non-slip and slip are compared for boundary conditions of fluid field bottom. The test results of wind load on sign board also show consistent in non-slip and slip conditions. Therefore, the bottom can be considered as a slip boundary.

## 6. Calculation for wind load on typical sign board

In the conditions above, CFD simulation for wind load on a typical sign board with 5 different wind speeds has been finished. The simulation results are consistent with the results of the empirical formula given by GB5768. Therefore, the method of CFD simulation calculation for wind load on typical sign board can be determined.

### 6.1 Computational model

#### 6.1.1 Model set up

According to the above calculation region and boundary conditions, wind load on traffic sign board simulation model has been set up, and grids has been divided using icem-CFD software in this study.

#### 6.1.2 Grid division

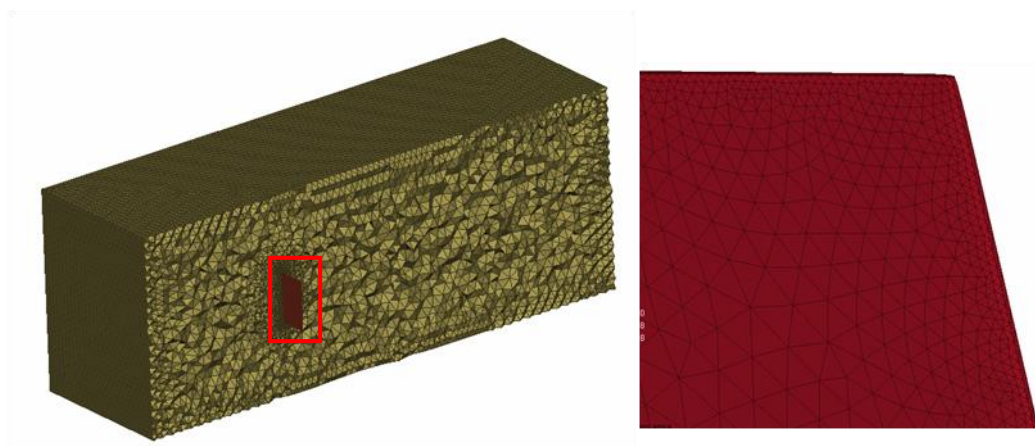
The essence of CFD is that the control equation is point or region dispersed in the required area to become defined algebraic equations at each grid point or in sub-region. Thus, the algebraic equations can be solved by using the method of linear algebra. The grid division is a geometric approximation of the physical component model. After grid division, the physical component model is divided into discrete aggregate of nodes and elements containing many simple geometric elements.

Grid division is one of the key steps in finite element calculation. When the object is very complex, the grid division technology often plays a vital role, which directly affects the accuracy of the subsequent numerical analysis results. The different grid divisions will cause the different solving method of numerical integration. Therefore, a reasonable unit must be used in this simulation solution.

The geometry of the traffic sign board is simple, but the thickness of the plate is only 3-5mm, which is far less than its height and width (generally more than 1m). Thus the grid division needs to reflect the details, but also to control the total number of grids. In addition, for the complex physical model structure (the different typical and special sign boards) the method of tetrahedral mesh and boundary layer was used in the computation phase to avoid the influence of hexahedral mesh block form the result.

Because of the large calculation area (length  $\times$  width  $\times$  height=42m $\times$ 24m $\times$ 16m) and the only 3mm thickness, the boundary layer mesh was set up to maximum reduce the number of grid. The grid density is increased in the area near the sign board, and the volume of a single grid is gradually increasing away from the signs to achieve the purpose of reducing the grids' number while meeting the computing requirement. The thickness of the first layer is 0.2mm, according to the 3mm thickness of the sign board.

Finally, the area was divided into 2,744,000 computational model grids, as shown in Figure 2.



**Figure 2.** Computational area and boundary conditions for typical sign board from CFD software.

### 6.2 Calculation results and analysis

After the above settings, the  $6\text{m} \times 4\text{m}$  sign board was simulated with the 5 different wind speeds, as the following results.

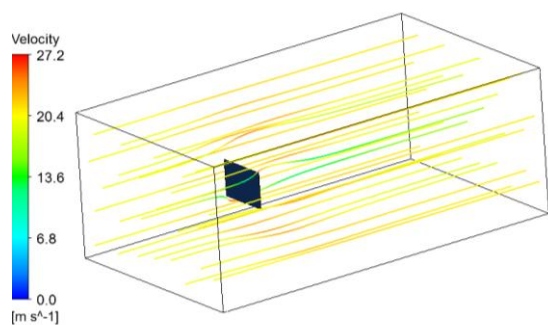


Figure 3. (a)  $v=20\text{m/s}$

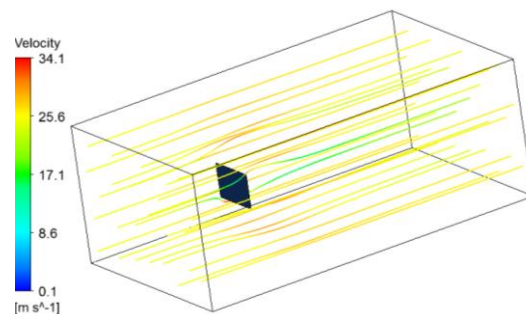


Figure 3. (b)  $v=25\text{m/s}$

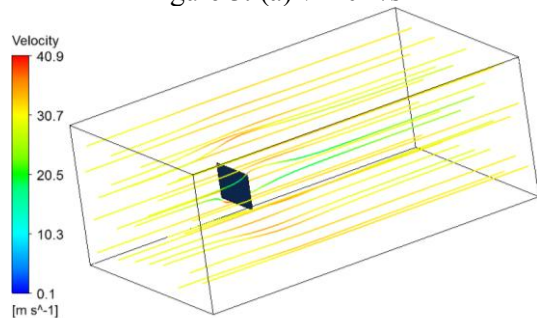


Figure 3. (c)  $v=30\text{m/s}$

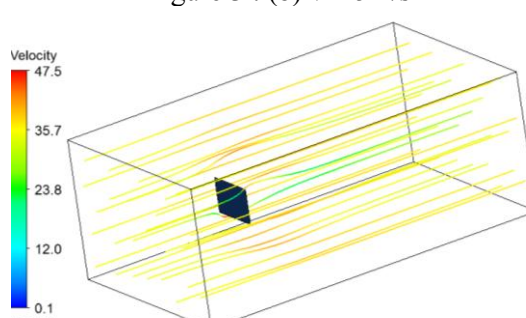
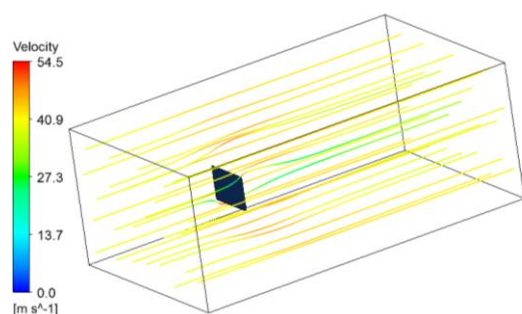
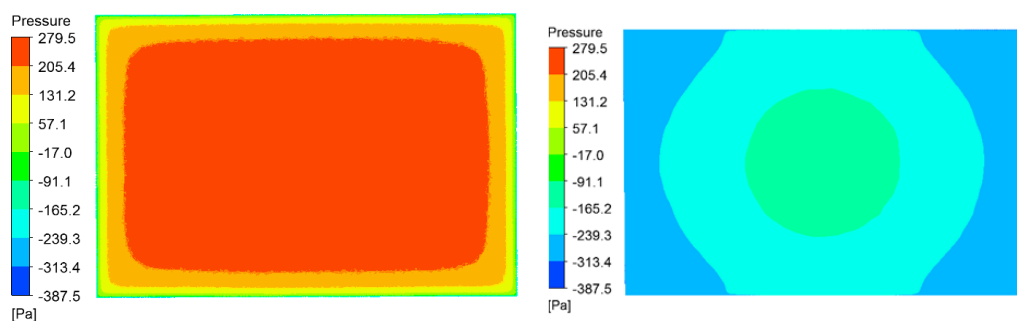
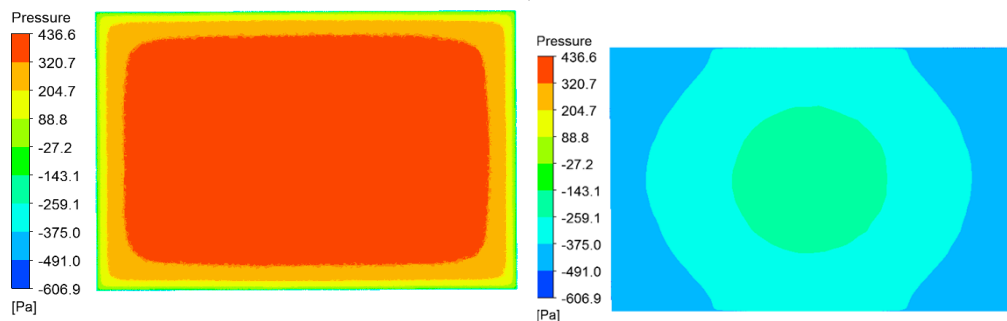
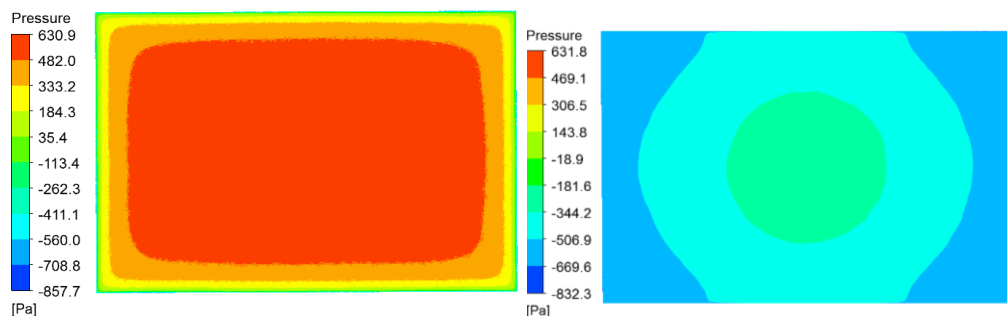
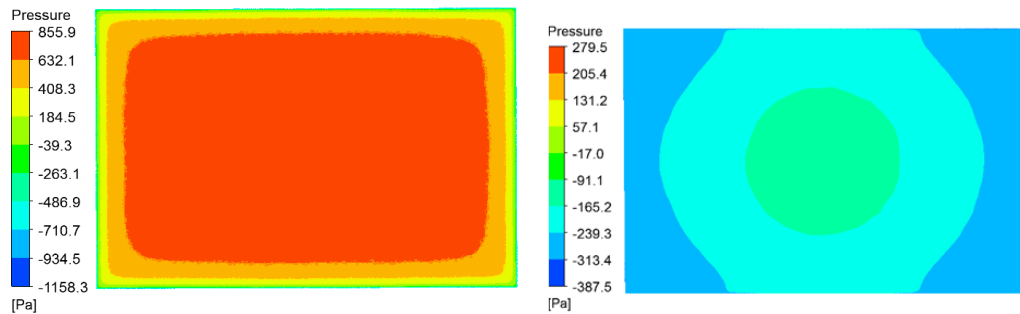
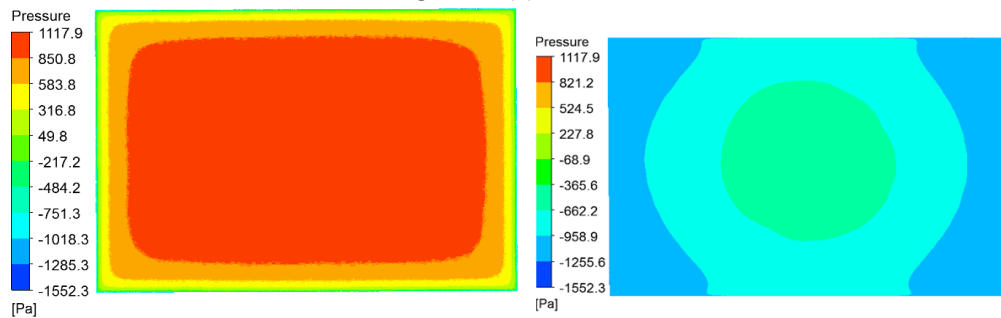
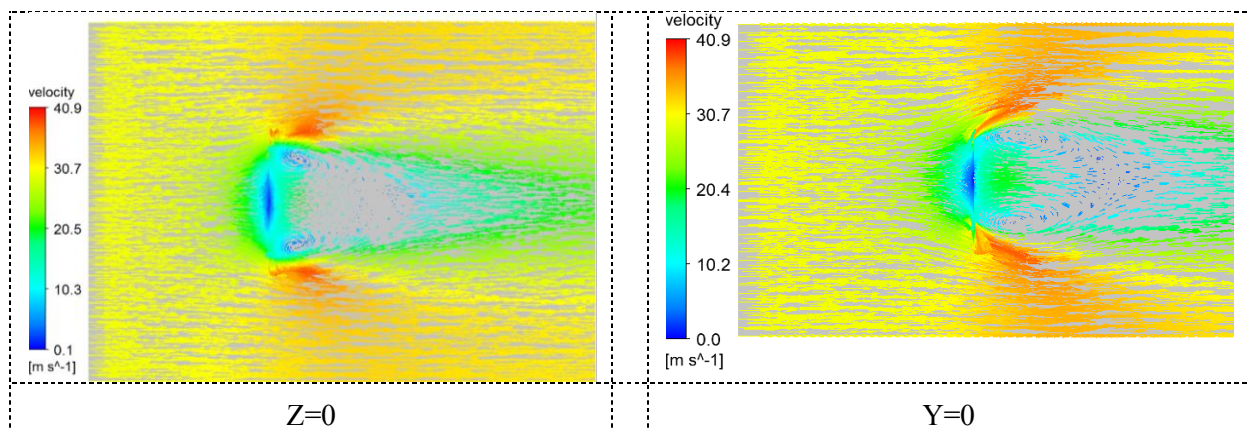


Figure 3. (d)  $v=35\text{m/s}$

Figure 3. (e)  $v=40\text{m/s}$ **Figure 3.** Calculation of watershed river flow diagram from CFD softwareFigure 4. (a)  $v=20\text{m/s}$ Figure 4. (b)  $v=25\text{m/s}$ Figure 4. (c)  $v=30\text{m/s}$

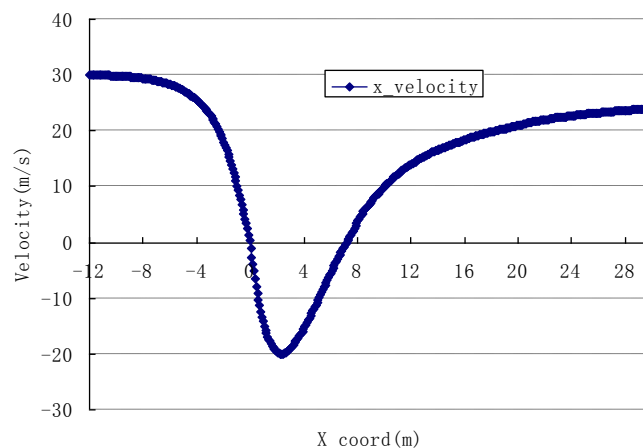
Figure 4. (d)  $v=35\text{m/s}$ Figure 4. (e)  $v=40\text{m/s}$ **Figure 4.** Calculate the watershed pressure distribution from CFD software.

In figure 4, the upwind and leeward sides pressure distribution of sign board decreases from inside to outside, but the decreasing trends are different. On upwind side, the pressure distribution is average with small gradient on central area. But the pressure gradient is large on the outer edge, where the speed increases significantly. On leeward side, pressure distribution is from the center to the outside similar to a ring.

**Figure 5.** Calculate watershed velocity vector diagram from CFD software.

According to figure 5, the main reason of the pressure distribution on leeward side is the formation of the return zone to absorb fluid. At the same time, it pushes the fluid around the field to form a new distribution of velocity and pressure.

Figure 6 shows the velocity distribution on the central line. The recirculation zone can be seen within 7.2m behind the sign board, obviously.



**Figure 6.** X direction speed on centre line: 30m/s from results.

Through integral on pressure on sign board surface, the pressure direction of wind direction was received. Then the wind load was compared with the empirical values. After comparison, the calculation results were very close, whose difference is about 1% shown in table 1. It can be considered that the grid is close to the different Re number. It shows that the simulation results are consistent with different wind speeds. Therefore, the computational model, grid division, computational area and boundary conditions are set reasonably.

Research Institute of Highway Ministry of Transport, 8 Xitucheng Rd. Haidian Dist.,  
Beijing, China

**Table 1.** Wind load calculation accuracy analysis from results.

Wind speed(m/s)	Empirical formula results(N)	CFD calculation result(N)	Relative error
20	9885	9979	0.95%
25	15445	15594	0.96%
30	22241	22468	1.02%
35	30272	30600	1.08%
40	39539	39988	1.13%

## 7. Summary

In this study, two methods for calculating wind load on traffic sign board have been provided and compared. The empirical method, calculate the wind load based on the area of the sign board, cannot solve the wind load on sign board with complex structure. The other CFD method can calculate the finite element the air flow field of complex structure. And it can well describe and solve the problem of fluid mechanics, which cannot be solved by the method of arithmetic. Furthermore, a typical sign board has been calculated by using CFD simulation software with 5 different wind speeds. The results are no significant difference with the results by using the empirical model. Therefore, it can be proved that the method of CFD is feasible to calculate the wind load of the sign board. In addition, it can develop the wind load reduction measures and the study on traffic signs in the complex situation.

However, the method is too complex for common traffic engineers. And it takes huge time about 3 days at least (normally a week) to build model and run program on large computer in CFD software for every different signs. Therefore, most traffic engineers prefer to use traditional method in Chinese standard and specification, which will take only several minutes to calculate wind load [10][11][12].

But this simple method cannot make calculation for special sign board (e.g. punched or slotted). For widely using in real traffic situation, some selected typical sign boards (or most common sign board) can be simulated and operated wind tunnel test. The results for those boards will be used in other projects. Furthermore, the approach has been successfully practiced in a demonstration project to calculate reducing wind load measures (including punching, slotting etc.). And the total steel consumption, construction and maintenance costs were reduced.

### Acknowledgments

This research is supported by “The National Key Technology R&D Program: Highway Safety Design Technology for Qinghai-Xizang Plateau Special Circumstances” (No.2014BAG05B02) and “special funding project of basic scientific research for central level public welfare research institute: Development and research of reducing wind load on traffic sign board”.

### References

- [1] JTG B01-2014, Ministry of Transport of the People's Republic of China Technical standard of highway engineering. 2014
- [2] British Columbia Ministry of Transport and Highways 2009 Manual of Standard Traffic Signs & Pavement Markings
- [3] JTG/T D81-2006, Ministry of Transport of the People's Republic of China Specification for Design of Highway Safety Facilities. 2006
- [4] Federal Highway Administration 2009 Manual on uniform traffic control devices 2009 edition
- [5] Constantinescu G, Bhatti A and Tokyay T 2008 Improved method for determining wind load on highway sign and traffic-signal structures
- [6] GB5768.2-2009 Ministry of Transport of the People's Republic of China 2009 *Road traffic signs and markings – Part 2: Road traffic signs*. GB5768.2-2009
- [7] Simiu E and Seanlan R H 1995 Wind Effects on Structures - An Introduction to Wind Engineering. The 3rd Edition. John Wiley & Sons, INC. **33-99** 135-88
- [8] Roache P J 1980 Computational Fluid Dynamics Hermosa Publishers
- [9] Krishna P 1995 Wind load on low rise buildings – a review *J. Wind Eng. hid. Aerodyn* **54/55** 383-96
- [10] Hangan H, Roberts D and Xu Z 2004 Downburst simulations experimental and numerical challenges *Proc. the 2004 Structures Congress – Building on the Past: Securing the Future* 1657-64
- [11] Murakami S and Mochida A 1987 Three dimensional numerical of airflow around a cubic mode by means of large eddy simulation *J. Wind Engr* **25** 291-305
- [12] Tamura T, Kawai H, Kawamoto S, Nozawa K, Sakamoto S, Ohkuma T, 1997 Numerical prediction of wind loadings and structures - Activities of AIJ Cooperative Project on CFD *J. Wind Eng. Ind. Aerodyn.* **67/68** 671-85
- [13] BS EN 1991-1-4:2005 Eurocode 1: General Actions – Wind Acitons
- [14] Kim J and Hangan H 2007 Numerical simulations of impinging jets with application to downbursts *J. Wind Engineering and Indus-trial Aerodynamics* **95** 279-98