Study on Wind Load Distribution on the Surface of Dome Structure Based on CFD Numerical Simulation

Peng Zhou¹, Baijian Tang¹,²*, Ping Liu¹, Gaoming Zheng¹

¹Jiangsu University of Science and Technology, Zhenjiang, China, 202003
²Suzhou University of Science and Technology, Suzhou, China, 215003

*Corresponding author e-mail:729404817@qq.com

Abstract: This study presents based on the constant Reynolds average Navier-Stokes equation, the surface wind pressure and surrounding wind field of the dome structure under different winds are numerically simulated by finite element software. The RNG turbulence model is used to calculate the flow coefficients around the building and the pressure coefficient distribution around the roof at 7 wind direction angles from 0° to 90°, and analyze the basic characteristics. In addition, the six aerodynamic components of each wind are calculated, and their variation with the wind direction angle is pointed out. Moreover, The maximum negative pressure at the top of different winds is predicted and compared with the wind tunnel test data of the structure. Those result agree well.

1. Introduction

Dome structures have been used in many buildings, especially in gymnasium. The space structure is a flexible structure that is sensitive to the effects of wind loads[1]. It is of great practical significance to carry out the study of wind loads on long-span dome structures. In structural design, wind loads often become the main control loads for such structures. Although the shape of the dome structure is mostly streamlined, due to the large span of the structure, the reasonable determination of wind load is a key issue in the calculation of wind resistance. This paper establishes a dome structure model for numerical simulation and compares it with the wind tunnel test of the National Grand Theater model by Dr. Yan Dazhao[2] from Peking University.

Wind tunnel test is the main method of structural wind engineering research, but it not only has high test cost and long period, but also has contradictions such as different scale ratios and complex turbulent structure, wind pulsation and difficulties in Simulation of High Reynolds number[3,4]. With the rapid development of computer hardware and the continuous advancement of computational fluid dynamics (CFD) technology, the application of computer and CFD technology to simulate the
construction wind field has become a new effective method to predict the wind effect of buildings\cite{5,6}. In this paper, the surface wind pressure and surrounding wind field of the dome structure are simulated and calculated by CFD numerical simulation method, and some results are compared with the wind tunnel test results.

2. Physical model
The dome structure model is a semi-ellipsoid dome with a titanium alloy plate and a partial glass curtain wall. Its governing equation is:

\[
\left(\frac{x}{108.125}\right)^2 + \left(\frac{y}{73.125}\right)^2 + \left(\frac{z}{46.125}\right)^2 = 1
\]  

(1)

The type of ground roughness is classified as Class III, the land type is C, and the ground roughness coefficient is 0.22. According to China's load code, the variation of the average wind speed along the height is expressed by an exponential function. The average wind speed at a height of 10 m is 9 m/s and the turbulence intensity is 0.31.

Figure 1. The geometry of model

The wind speed profile is fitted according to the atmospheric boundary layer wind speed profile simulated in the wind tunnel test:

\[
U_z = \begin{cases} 9 & Z \leq 10 \\ 9 \times \left(\frac{z}{10}\right)^{0.22} & Z > 10 \end{cases}
\]

(2)

There is no relevant specification for turbulence intensity in our country. The recommended value of wind load regulations in Japan is taken:

\[
I_z = \begin{cases} 0.31 & Z \leq 10 \\ 0.1 \times \left(\frac{z}{500}\right)^{-0.25} & Z > 10 \end{cases}
\]

(3)

The turbulent integral scale refers to the empirical formula suggested in the Japanese specification (AIJ-1996):

\[
L_z = 100 \times \left(\frac{z}{30}\right)^{0.5}
\]

(4)
Due to the randomness of the actual wind direction and the symmetry of the structure, a total of seven wind directions are numerically simulated. The wind direction angle is set to 0° in the north wind direction and from 15° to 90° every 15° as a wind direction angle.

3. Turbulence model and control equation

In this paper, the RNG $k-\varepsilon$ model is used for numerical simulation\[^7,8\], and the control differential equation as follows:

Continuity equation:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0$$  \hspace{2cm} (5)

Momentum equation:

$$\rho u_j \frac{\partial \bar{u}_i}{\partial x_i} = -\frac{\partial}{\partial x_i} \left( p + \frac{2}{3} \rho k \right) + \frac{\partial}{\partial x_j} \left[ \mu_e \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \right]$$  \hspace{2cm} (6)

Where, $u_i$ is the average velocity component; $p$ is the average pressure; $\rho$ is the density; $k$ is the turbulent flow energy; $\varepsilon$ is the turbulent energy dissipation rate; $\mu_e = \mu + \mu_t$, $\mu$ is the molecular viscosity coefficient; $\mu_t$ is the turbulent viscosity coefficient, and $\mu_t = C_{\mu} \rho \frac{k}{\varepsilon}$. $P$ is the enthalpy flow generation term, $P = \mu_t \frac{\partial \bar{u}_i}{\partial x_j} \left( \frac{\partial \bar{u}_i}{\partial x_i} + \frac{\partial \bar{u}_j}{\partial x_j} \right)$; $R = \frac{\eta \left( 1 - \frac{\eta}{\eta_0} \right)}{\left( 1 + \beta \eta^3 \right)} P$.

Which $\eta = \sqrt[3]{\frac{P}{\rho C_{\mu} \varepsilon}}$; Other constant taken value as follows: $C_{\mu} = 0.085$, $\sigma_k = 0.7179$, $\sigma_{\varepsilon} = 0.7179$, $C_1 = 1.42$, $C_2 = 1.68$, $\eta_0 = 4.38$, $\beta = 0.012$.

4. Boundary conditions and meshing

The calculation area is taken as a rectangular parallelepiped of 1000m × 700m × 300m, as shown in Figure 2.

![Figure 2. Calculation area space diagram](image)
Inflow boundary: The normal wind speed of the inflow surface is taken according to the Eq-1, the tangential speed defined as 0, and the value is as follows: $k = 1.5T^2U^2; \ v = k^{1.5} / L$.

Outflow boundary: The pressure on the outflow surface was $1.013 \times 10^4$ Pa. Calculate the top and sides of the area: the normal velocity of each face is 0. The ground and the surface of the building: the speed of each face is 0.

5. Calculation results and analysis

5.1 The velocity vector plot

Fig.3 shows the wind speed vector on the central axis of the dome when the angle is $0^\circ$ (wind direction is negative along the y axis). Since the dome of the Grand Theatre has a good streamlined shape, eddy currents are formed only in a small part of the downwind zone.

![Figure 3](image1.png)

**Figure 3.** The velocity vector plot on the central axis of the dome when the angle is $0^\circ$

![Figure 4](image2.png)

**Figure 4.** The plot of energy vs time of statics

Comparing Fig. 4(a) and 4(b), it can be found that in the lower horizontal section, a very obvious flow separation occurs in the downwind zone and a certain vacuum recirculation zone is formed, in which the wind speed vector diagram is symmetrically distributed. On the higher horizontal section, a significant wind speed vector range distribution is observed. The farther away from the dome, the smaller the wind speed. The maximum wind speed appears at the junction of the central axis (x-axis) and the plane.
5.2. Wind pressure coefficient distribution map

Figure 5 is a top view of the dome pressure distribution when the angle is 0°. Since the wind direction is consistent with the symmetrical central axis of the north-south direction of the dome, the pressure distribution is basically symmetrical. Compared with the reference static pressure of 1.013 ×10^4Pa, except for a small part of the windward and leeward sides, which are positive pressure, most of them are negative pressure. The maximum negative pressure appears at the top of the dome, and the maximum positive pressure appears on the windward side.

![Figure 5. Pressure distribution of the dome when the wind direction angle is 0°](image)

5.3. Maximum negative pressure and pressure coefficient at tops of different wind direction

Define the pressure coefficient of the building surface as:

\[
C_p = \frac{P - P_{ref}}{0.5 \rho U_{ref}^2}
\]  

(7)

Where \(P_{ref}\) is the reference static pressure and is taken as 1.013×10^4Pa; \(U_{ref}\) is the reference wind speed, and the wind speed at the height of 10m is taken; Air density \(\rho = 1.17\) kg/m^3.

As shown in Table 1, when the angle is 0°, the maximum negative pressure value at the top is the largest. As the wind direction angle increases, the maximum negative pressure value at the top gradually decreases. When the wind is 90°, the minimum is reached. The values of the maximum negative pressure coefficient of the 0° and 90° wind direction angles are -1.80 and -0.98, respectively, the numerical simulation are -1.85 and -0.92. The results are basically the same. As shown in figure 6.

| wind direction | 0°    | 15°   | 30°   | 45°   | 60°   | 75°   | 90°   |
|----------------|-------|-------|-------|-------|-------|-------|-------|
| wind pressure/Pa | 10042.2 | 10047.7 | 10051.6 | 10058.4 | 10070.4 | 10081.9 | 10086.3 |
| wind pressure coefficient/\(C_p\) | -1.85 | -1.73 | -1.66 | -1.51 | -1.26 | -1.06 | -0.92 |

Table 1. Maximum negative pressure and pressure coefficient at tops of different wind direction
5.4 Aerodynamic components in different wind directions

The expressions of the dimensionless coefficients $C_x$, $C_y$, and $C_z$ obtained by solving the integral of the pressure coefficient of the Eq-9 on the surface of the building are:

$$C_x = -\frac{1}{S}\int C_p dydz$$

$$C_y = -\frac{1}{S}\int C_p dxdz$$

$$C_z = -\frac{1}{S}\int C_p dxdy$$

(8)

Then, the jointless dimensionless coefficients obtained by Eq-10 are respectively obtained by $x$, $y$, $z$, and the dimensionless coefficients $M_x$, $M_y$, $M_z$ of the expression are:

$$M_x = -\frac{1}{Sa}\int C_p (ydy - z\text{sgn}(y)dxdz)$$

$$M_y = -\frac{1}{Sa}\int C_p (z\text{sgn}(x)dydz - xdydz)$$

$$M_z = -\frac{1}{Sa}\int C_p (x\text{sgn}(y)dxdz - y\text{sgn}(x)dydz)$$

(9)

Where $S$ is the surface area of the dome, $a$ is the length of the semi-major axis.

The trend of the dimensionless coefficient of the aerodynamic component of the dome surface changing with the wind direction angle is shown in Fig.6.

![Figure 6](image_url)

**Figure 6.** The maximum negative pressure coefficient at the top varies with the wind direction

**Figure 7.** The resultant force and resultant moment of the three directions of the dome structure with the wind direction
From the Fig.7, the normal force $C_z$ is larger than the combined forces $C_x$ and $C_y$ in the other two directions, and $C_x$ approaches 0 when the angle is $0^\circ$. When the wind direction angle is $90^\circ$, $C_y$ approaches zero because the dome structure is an axisymmetric structure about the $x$ and $y$ axes, and the wind pressure on the surface is roughly symmetric. Therefore, the resultant air volume is smaller than the $z$-axis. When the angle is $0^\circ$, the normal force reaches the max.

From the numerical simulation results, $M_y$ and $M_z$ are close to zero when the angle is $0^\circ$ and $90^\circ$. The combined moments $M_x$ and $M_y$ are proportional to the wind direction angle, while $M_z$ is symmetrically distributed with respect to the $45^\circ$ angle, which is more accord with wind tunnel test data, which verifies the reliability of the simulation of the number of times. The reliability of this numerical simulation is verified.

6. Conclusion
This study presents the surface mean wind pressure distribution of the dome structure under wind load is studied by numerical simulation method. The flow separation phenomenon of the dome structure is weaker than that of the bluff body, so that the force of the wind in the horizontal direction is smaller, and the vertical suction is the main force; the north wind (the wind angle is $0^\circ$) is the most unfavorable condition. The results are basically consistent with the wind tunnel test data.

Acknowledgments
This work was supported by the national Natural Science Foundation of China (grant numbers 51508238) and by the Jiangsu Postdoctoral Research Plan (grant numbers 1601014B).

References
[1] Wei De-min, Xu Mu, LI Di, Time-history analysis of wind-induced vibration response for a cable dome structure. J. Journal of Vibration and Shock. 2013, 32(17):68-73
[2] Yan Dachun, Sun Zhili, HUANG Sheng, Wind tunnel test of wind load in China National Grand Theater. J. Acta scientiarum Naturalium Universitatis Pekinensis, Vol.37, No.3 (May, 2001)
[3] Gu Ming WANG Xinrong, Research Progress of Reynolds Number Effect of Engineering Structures, D. Journal of Tongji University (Natural Science), 2013, 41(7):961-969.
[4] Liu Jianxin, Cui Xin, Li Jiawu, Reynolds number effect of surface pressure distribution and Strouhal number of a typical bridge deck section, J. Journal of Vibration and Shock, 2010, 29(4):146
[5] Li Chao, Study on CFD Simulation of Turbulence Wind Field Near Ground, D. Harbin Institute of Technology, 2010
[6] A. Sengupta, F. L. Haan, P. P. Sarkar, V. Balaramudu, Transient Loads on Buildings in Microburst and Tornado Winds, J. Journal of Wind Engineering and Industrial Aerodynamics. 2008, 96(10-11):2173-2187
[7] Rafat Al-wakcd, Musud Bchnia, CFD simulation of wet cooling towers, J. Applied Thermal Engineering. 2006, 26:382—359.
[8] Kim S E, Boysan F, Application of CFD to environmental flows, J. Journal of Wind Engineering and Industrial Aerodynamics, 1998, 81:145-158.