Research on aerodynamic and flow field characteristics of high speed elevator with different shaft blockage ratio

Hanwen Cui1, Qing Zhang1*, Ruijun Zhang1, Luzhong Zhang2

1Shandong Jianzhu University, School of Mechanical and Electronic Engineering, Jinan, China
2Shandong Electronic Information Products Inspection Institute, Jinan, China
*Corresponding author: zhangqing@sdu.edu.cn

Abstract—With the increase of building height, more and more high-speed elevators are applied to high-rise buildings in order to improve the transportation efficiency of people in buildings. Compared with the medium and low speed elevators, the airflow in the shaft is more severe when the high-speed elevator car is running in the narrow and long shaft, and the influence of aerodynamic force and aerodynamic noise is more obvious. In order to explore the aerodynamic characteristics of the high-speed movement of the elevator car and the airflow law in the shaft, and further optimize the aerodynamic parameters of the high-speed elevator shaft system, a full-scale three-dimensional model of the shaft with different blocking ratio is established and numerically simulated. Through the analysis of the calculation results of different blocking ratio of the shaft model, it is found that the greater the blocking ratio is, the greater the resistance of the car operation is, and the airflow disturbance behind the car movement is more severe than that in the front. The calculation results provide a reference for further exploring the airflow flow law in the high-speed elevator shaft.

1.Introduction
When the elevator car runs at high speed in the narrow and long closed shaft, the piston effect will squeeze the air in front of the car. The air will flow to the rear of the car through the gap between the car and the shaft. The speed of the air flow will increase when the air flow passes through the narrow gap, and the airflow disturbance in the shaft will be more severe. The unstable air pressure wave will produce on the car operation, the obvious aerodynamic influence will increase the resistance, noise and vibration of the car, and even affect the safe operation of the elevator [1]. Therefore, it is of great significance for the parameter optimization design of high-speed elevator to study the airflow flow law and aerodynamic characteristics of the high-speed elevator.

At present, a large number of scholars have studied the aerodynamic parametric design of high-speed elevator shaft and car. Based on Bernoulli's unsteady flow theory, Zhang et al. [2] carried out theoretical simulation of unsteady flow in shafts with no vent, single vent and double vent, and analyzed the influence of different vent parameters on airflow.

Bai et al. [3] tested the instantaneous speed of the elevator car at five different positions and the average pressure along the shaft before and after the car, carried out the test on four different shapes of the car, carried out numerical simulation on the parameters of the car and the shaft with different shapes, and analyzed the dimensionless pressure difference under the parameters of the car and the shaft with different shapes.
Yang et al. [4] established a four degree of freedom model of elevator lateral vibration based on Lagrange principle, and combined with computational fluid dynamics, analyzed the influence of various parameters corresponding to different working conditions on the lateral surface aerodynamic force of the lift car.

Ling et al. [5] used computational fluid dynamics to simulate the flow field of high-speed elevator with different deflector heights, and analyzed the resistance of the lift car.

Chen et al. [6] respectively established a three-dimensional frame car counterweight simulation model and a three-dimensional frameless car counterweight simulation model for simulation and comparison. By analyzing the aerodynamic characteristics of the elevator motion process in the two models, it is found that the frame model and the frameless model have little difference in aerodynamic characteristics.

Based on SST k-w turbulence model, Guo et al. [7] simulated the airflow in elevator shaft, calculated aerodynamic resistance, roll moment and pitch moment, and analyzed the change trend of experimental data.

Based on the above research results, in order to explore the influence of high-speed elevator shaft blockage ratio on the airflow characteristics in the shaft, this paper establishes a full-scale three-dimensional shaft and car model with different shaft blockage ratio. According to RNG k - ε turbulence model and using Fluent19.2 software, the dynamic grid numerical simulation method is used to simulate the motion of the car in the hoistway, and the flow characteristics and aerodynamic influence of the shaft flow field under different blockage ratios are explored.

2. Governing equations

When the high-speed elevator is running in the shaft, the air velocity in the shaft is slow, the velocity of air flow belongs to the low Mach number flow. It is not necessary to consider the compressibility of the air. The air flow in the shaft is regarded as the incompressible viscous fluid. Three dimensional incompressible air flow is described by continuity equation and N-S momentum equation [8].

Continuity equation:

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0$$  \hspace{1cm} (1)

Momentum equation:

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu(\nabla u_i + \nabla u_i^T) \right] + \frac{\partial \tau_{ij}}{\partial x_j}$$  \hspace{1cm} (2)

The RNG k-ε two equation turbulence model is adopted to consider the swirling air flow in the elevator car and the low Reynolds number flow near the wall, k equation:

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \alpha_k \mu_{eff} \frac{\partial k}{\partial x_j} \right) + G_k - \rho \varepsilon$$  \hspace{1cm} (3)

ε equation:

$$\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \alpha_\varepsilon \mu_{eff} \frac{\partial \varepsilon}{\partial x_j} \right) + C_{1\varepsilon} \frac{G_k \varepsilon}{k} - C_2 \rho \frac{\varepsilon^2}{k}$$  \hspace{1cm} (4)

Where, \(x_i, x_j\) is the coordinate component, \(u_i, u_j\) is the instantaneous component of velocity, \(i, j = 1, 2, 3\); \(\rho\) is the air density; \(p\) is air pressure; \(\alpha_k, \alpha_\varepsilon\) is the Prandtl number corresponding to turbulent kinetic energy \(k\) and dissipation rate \(\varepsilon\); \(C_k = \alpha_\varepsilon = 1.39; \quad C_{2\varepsilon} = 1.68; \quad G_k\) is the production term of turbulent kinetic energy \(k\); \(\mu_{eff}\) is the effective viscosity coefficient, the expression is
\[ \mu_{\text{eff}} = \mu + \mu_t \]  

(5)

Where \( \mu_t \) is expressed as \( \mu_t = \rho C_p \frac{k^2}{\varepsilon} \), \( C_p \) is a constant, \( C_p = 0.0845 \);

\[ C_{ie} = C_{te} - \frac{1}{1 + \beta \eta} (1 - \frac{\eta}{\eta_0}) \]  

(6)

In equation (6), \( C_{te} = 1.42, \eta_0 = 4.38, \beta = 0.012; \eta = (2E_y E_y)^{\frac{1}{2}} \frac{k}{\varepsilon} \), which

\[ E_y = \frac{1}{2} \left( \frac{\partial u_i}{\partial x} + \frac{\partial u_i}{\partial y} \right) \]  

(7)

3. Geometric Mode

When building the model of the shaft system, the shape of the car is simplified as a cuboid, and the guide rail, counterweight and other facilities in the shaft are ignored, and the three-dimensional model of the shaft and the car is established. As shown in Fig. 1, the lift car is 1.6m long in x direction, 1.8m wide in y direction and 2.8m high in z direction, the length of the shaft along the x direction is \( 2a + 1.6 \) m, the width along y direction is 2.24m, and the height along the z direction is 60m. By setting the distance between car wall and shaft wall in y direction as 1/6, 1/3, 1/2 and 2/3 times of car width, that is \( a \) is set to 0.3m, 0.6m, 0.9m and 1.2m respectively to reduce the blocking ratio of the shaft, and the car motion process under four blocking ratios is numerically simulated.

4. Meshing and boundary conditions

As shown in Fig. 2, the mesh of the car near the wall is encrypted to calculate the flow field around the car and the aerodynamic force of the car more accurately. The total number of grids is 870 thousand. In terms of boundary conditions, the bottom of the shaft is set as the pressure inlet, the top of the shaft is set as the pressure outlet [9], the surrounding of the shaft is set as the wall condition, and the dynamic mesh simulation method is adopted for the car motion [10]. The car starts to move from the bottom of the shaft with an acceleration of 1.5m/s\(^2\), and maintains a constant speed when the speed reaches 4m/s. The air flow in the shaft and the aerodynamic force of the car during 12s of acceleration and uniform motion of the car are monitored.
5. Simulation results

The grid file is imported into Fluent19.2 to calculate the flow field and aerodynamic force of the car within 12s from acceleration and uniform motion, and monitor the data changes. The aerodynamic drag coefficient curve and lift coefficient curve of the car motion process are drawn with the calculated data, and the real-time flow field cloud of the hoistway air flow is processed by CFD-Post in order to observe the change of flow field in the shaft more intuitively. As shown in Fig. 3, in the process of the car accelerating from static to \(4 \text{m/s}\) at the acceleration of \(1.5 \text{m/s}^2\), the resistance of the car under different shaft blockage ratio increases continuously; when the car speed reaches \(4 \text{m/s}\), the aerodynamic resistance of the car drops sharply due to the end of the acceleration process; after the acceleration process is completed, when the car moves at a constant speed of \(4 \text{m/s}\), the aerodynamic resistance of the car decreases gradually, and the larger the blockage rate of the shaft, the greater the resistance of the car.

As shown in Fig. 4, when the car speed is uniformly accelerated from 0 to \(4 \text{m/s}\), the aerodynamic lift coefficient of the car in the horizontal direction increases continuously, that is, the greater the blockage ratio of the hoistway, the greater the aerodynamic lift of the car; when the speed reaches \(4 \text{m/s}\), the acceleration process of the car ends. However, in the process of uniform motion, the lift force of the lift car changes dramatically due to the severe airflow disturbance caused by the acceleration process. The lift coefficient curve of the car in the horizontal direction tends to be smooth and decreases with the development of uniform motion.

In order to explore the air flow around the car during the acceleration of the elevator, the velocity streamline diagram of the flow field on the \(xoz\) plane when \(t = 2\)s is taken. As shown in Fig. 5, the
airflow disturbance around the car is obvious. Compared with the air in the front of the car, the air velocity at the rear of the car is higher and the airflow disturbance is more severe.

![Figure 4](image4.png)  
**Figure 4.** Lift coefficient curve of lift car in horizontal direction

![Figure 5](image5.png)  
**Figure 5.** Airflow streamline diagram around xoz plane car when $t = 2s$

6. Conclusions

In this paper, a simplified three-dimensional model of elevator shaft system is established, according to RNG k – ε turbulence model and using Fluent19.2 software, the dynamic mesh method is used to simulate the acceleration and uniform speed process of the car under different blocking ratios, and the results are analyzed:

(1) The drag and lift force of the car in the process of accelerating operation increase continuously, and the greater the blockage rate of the shaft, the greater the drag and lift of the car; after the end of the acceleration process, the drag and lift force of the car will change sharply, and finally gradually decrease.  

(2) When the elevator is running, the air disturbance around the car is more severe, and the air velocity behind the elevator is higher than that in front of the car.
REFERENCES
[1] X. Wang, Y. Yu, R. Zhang, S. Wang, and Y. Tian, “Summary of the Noise Research of High-speed Traction Elevators,” Noise and vibration control, vol. 34, no. 3, pp. 1-5, 2014.
[2] R. Zhang, S. Qiao, Q. Zhang, and L. Liu, “Universal modeling of unsteady airflow in different hoistway structures and analysis of the effect of ventilation hole parameters,” Journal of Wind Engineering and Industrial Aerodynamics, vol. 194, no. 3, 2019.
[3] H. Bai, G. Shen, and A. So, “Experimental-based study of the aerodynamics of super-high-speed elevators,” Building Services Engineering Research and Technology, vol. 26, no. 2, pp. 129-143, 2005.
[4] Z. Yang, Q. Zhang, R. Zhang, and L. Zhang, “Transverse Vibration Response of a Super High-Speed Elevator under Air Disturbance,” International Journal of Structural Stability and Dynamics, vol. 19, no. 9, pp. 164-179, 2019.
[5] Z. Ling, P. Tang, X. Wang, Z. Lin, and D. Tang, “Research on the drags of high speed elevator with different height diversion cover,” In: Proceedings of the 4th International Conference on Mechatronics, Materials, Chemistry and Computer Engineering, Xi An, pp. 2366-2370, 2015
[6] L. Chen, Y. Liu, K. Yu, C. Jiang, and Q. Zhang, “Research on the Influence of Frame Structure on Aerodynamic Characteristics of the High-speed Elevator During the Motion Process,” Chinese Hydraulics & Pneumatics, vol. 2, pp. 151-154, 2020.
[7] W. Qiu, T. Guo, Y. Xue, and J. Fu, “Hoistway Vent Optimization Design Based on SST k-ω Turbulence Model and Genetic Algorithm for High-speed Elevator,” Chinese Hydraulics & Pneumatics, vol. 5, pp. 51-57, 2019.
[8] H. Bao, C. Wang, F. Lu, and R. Zhu, “Windage losses analysis of GTF engine fan drive gearbox planetary gear based on CFD,” Journal of Central South University (Science and Technology), vol. 51, no. 4, pp. 971-978, 2020.
[9] D. Chen, “Aerodynamic characteristic of the super high speed elevator base on different hoistway models,” Mechanical & Electrical Engineering Magazine, vol. 33, no. 11, pp. 1359-1363, 2016.
[10] L. Shi, Y. Liu, S. Jin, and Z. Cao, “Numerical Simulation of Unsteady Turbulent Flow Induced by Two-Dimensional Elevator Car and Counter Weight System,” Journal of Hydrodynamics, Ser. B, vol. 19, no. 6, pp. 720-725, 2007