A study on characteristics of flow field around vehicle body

Ruochen Zhao¹, Zhen Liu²* Yinzhen Liu² Xiaobo Luo¹ and Yanan Sun¹

¹Engineering Management Department, Inner Mongolia Honder College of Arts and Sciences, Hohhot, Inner Mongolia, 010070, China
²College of Energy and Power Engineering, Inner Mongolia University of Technology, Hohhot, Inner Mongolia, 010051, China
*Corresponding author’s e-mail: 20191100157@imut.edu.cn

Abstract. A numerical simulation calculation is performed using large eddy simulation method to investigate the characteristics of flow field around vehicle body, analyze the change rules of the velocity and pressure of the three-dimensional flow field around the vehicle body, as well as the frequency domain characteristics of the aerodynamic coefficients in the vehicle body. The results reveal that with the gradually expanding distance from the rear of the vehicle body, the velocity in the flow field around the vehicle body gradually recovers and reaches the incoming flow velocity at 0.9m from the rear of the vehicle body. Within the range of 0~50HZ, the resistance coefficient and lift coefficient undergo a peak attenuation to about 20%, with a fluctuation of aerodynamic coefficient mainly within the range of 0~500HZ. The research results are of certain significance and value for analyzing and applying the characteristics of flow field around vehicle body.

1.Introduction
Ahmed bluff body, characterized by simple structure and easy control, is a primary choice when flowing around a vehicle body [1-3]. For example, in the aerodynamic research of bluff bodies, Chen Xin et al. [4] simulated the flow field of Ahmed vehicle model using DDES turbulence model, and obtained the influence law of different oblique back angles on the time averaged characteristics and transient performance of the wake flow field. In the meantime, the research on the transportation and evolution law of the main eddy's structure in the wake flow field is performed. Hu Yu et al. [5] calculated transient numerical values of the wake flow field on the Ahmed model with a caster angle of 25° using three different turbulence models of URANS, LES and SST kw-based DES, and analyzed the wake flow field after numerical simulation by three models. For vehicle aerodynamics and noise control, Cui Wenshi et al. [6] performed experimental investigation on the unsteady flow of three-dimensional vehicle wakes with different caster angles in combination with frequency spectrum and correlation analysis methods using a hot wire anemometer. Taking Ahmed bluff body model as the research object in this paper, transient numerical calculation of the flow field around the bluff body model is performed using large eddy simulation method and the characteristics of the flow field around the bluff body are analyzed.

2.Large eddy simulation method
In this paper, numerical simulation calculation is performed using large eddy simulation method, by which the instantaneous NS equation is used to directly simulate the eddy larger than the grid scale in the turbulence, rather than the simulation of the motion of the eddy on the full scale. Then, a
A turbulence model is established to indirectly simulate the influence of small eddies on large eddies, instead of directly simulating small-scale eddies. Filtering of Navier-Stokes equations of continuous and incompressible flow is performed using large eddy simulation method, and the control equation is obtained.

\[ \frac{\partial \overline{u_i}}{\partial x_j} = 0 \]  

\[ \frac{\partial (\overline{u_i})}{\partial t} + \frac{\partial \overline{u_i u_j}}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_j} + \nu \frac{\partial^2 \overline{u_i}}{\partial x_i \partial x_j} - \frac{\partial \tau_{ij}}{\partial x_j} \]  

\[ \tau_{ij} = \overline{u_i u_j} - \overline{u_i} \overline{u_j} \]

Where, \( \overline{u_i} \) and \( \overline{u_j} \) are the time-averaged velocity components after filtering; \( \overline{p} \) is the pressure after filtration; \( \rho \) is the density; \( \nu \) is Kinematic viscosity; \( \tau_{ij} \) is the sub-grid stress, which is the momentum transport between filtered small-scale pulsations and solvable-scale turbulent flows.

3. Calculation model and conditions

3.1. Calculation model and grids

A three-dimensional model of Ahmed bluff body is established in UG, as shown in Figure 1.

![Figure 1. Three-dimensional model of vehicle body.](image)

Ahmed blunt body is 1044mm in length, 389mm in width, and 288mm in height, respectively, with the head fillet radius of 100mm and the pillar radius of 50mm. The specific parameters are shown in Figure 2.

![Figure 2. Model parameters.](image)

In the front of bluff body model is set 5 times the length, 5 times the height, 5 times the width on the side, and 10 times the length in the rear. The calculation domain is shown in Figure 3.
Figure 3. Calculation domain.

The grid model established by polyhedral grid division is shown in Figure 4, with a grid quantity of 1620833.

Figure 4. Grid division.

3.2. Initial conditions and boundary conditions

The initial and boundary conditions are set as follows: Velocity inlet: $U=60 \text{ m/s}$, $V=0 \text{ m/s}$, $W=0 \text{ m/s}$, pressure outlet: relative pressure $P=0 \text{ Pa}$, the surface of the bluff body is a solid non-slipping wall surface, with solid wall sliding wall surface in other boundaries.

4. Analysis on the characteristics of flow field around the bluff body

4.1. Pressure field distribution

Figure 5 shows a diagram of pressure distribution on the symmetrical surface of the bluff body, where a small high-pressure area is formed at the incoming flow stagnation point, and a small low-pressure area is formed on both sides of the incident flow direction, which recovers gradually with the pressure in the flow direction. It can be seen from Figure 5 that a small-range negative pressure values will be generated in the places with dispersed local airflow at the upper and lower edges of the front and the caster angle.

Figure 5. Diagram of pressure on the symmetrical surface of the vehicle body.

Figure 6 is the diagram of the pressure distribution on the surface of the bluff body. Through the analysis in Figure 5, it is found that pressure at the nose cone of the bluff body reaches the maximum value of about 2200 Pa and gradually drops along the inflow direction. The pressure on the surface in the middle of the bluff body is uniformly distributed, and a small low-pressure area shows up on the rear slope, with a pressure value much higher than that in the head of the bluff body. The maximum value of the negative pressure on the surface of the bluff body reaches 4300pa.
4.2. **wake velocity analysis**

Figure 7 illustrates the velocity distribution cloud at the end of the blunt body. With the distance from the wake of the blunt body gradually increasing, the velocity gradually recovers and at 0.9m from the wake of the blunt body the velocity recovers to the incoming velocity. There is a 21% drop in velocity between 0 and 0.4 from the tail. According to the bottom velocity trend, there is a sharp drop in velocity between 0.3 and 0.5m from the tail of the blunt body, with a 56% drop.

Figure 7. Velocity distribution at the end of the vehicle body.

Figure 8 illustrates the velocity vector distribution of the obtuse body flow around the field, which can be seen to range from 0m/s to 89m/s. As the air flows through the blunt body head, it forms a standing point and the air flow separation results in a high velocity zone above and below the blunt body head, while the upper part of the tail also has a high velocity zone due to flow separation. Vortices of different scales appear in the triangle area, with clockwise vortices appearing above the area and counter-clockwise vortices below. This is because of the pressure gap in the triangular vortex region where the presence of these vortices causes the triangular vortex region to consume a large amount of energy, which results in a reduction in pressure of the end part.

Figure 8. Velocity vector diagram.

4.3. **Frequency domain characteristics of aerodynamic coefficients**

Pneumatic loads have all fluctuating nature and their variation in the frequency domain has a regularity, which is based on frequency. As shown in Figure 9 and Figure 10, the frequency domain variation of the blunt body drag coefficient and lift coefficient respectively. Generally, there are different peaks in the frequency domain for each of the aerodynamic coefficients of the obtuse body. Nevertheless, they are all gradually decreasing and stabilizing and the peak value of the aerodynamic coefficient decays to about 20% in the range of 0~50HZ. These fluctuations are mainly in the range of 0~500HZ, and the peak distribution of the drag and lift coefficients are also varied.
5. Conclusions
(1) The pressure is high at the nose cone of the obtuse body, with a maximum value of about 2200 pa, and it gradually decreases in the direction of incoming flow. The pressure distribution on the central surface of the blunt body is relatively uniform, while a small area of low pressure appears at the end slope, the pressure value of which is much higher than the pressure value at the head of the blunt body.
(2) With the distance from the end of the obtuse body gradually increasing, the velocity in the winding field gradually recovers and at a distance of 0.9m from the end of the obtuse body the velocity recovers to the incoming velocity.
(3) As the air flows through the blunt body head, it forms a standing point and the air flow separation results in a high velocity zone above and below the blunt body head, while the upper part of the tail also has a high velocity zone due to flow separation. Vortices of different scales appear in the triangle area, with clockwise vortices appearing above the area and counter-clockwise vortices below. This is because of the pressure gap in the triangular vortex region where the presence of these vortices causes the triangular vortex region to consume a large amount of energy, which results in a reduction in pressure at the end part.
(4) There are different peaks in the frequency domain for each of the aerodynamic coefficients of the obtuse body. Nevertheless, they are all gradually decreasing and stabilizing and the peak value of the aerodynamic coefficient decays to about 20% in the range of 0–50HZ.

Acknowledgments
The authors wish to acknowledge the support and motivation provided by the Key Projects in the Inner Mongolia natural science foundation: Research on the Aerodynamic Load of High-Speed Train Pantograph under Crosswind (2018LH01011).

Reference
[1] Conan, B., Anthoine, J., Planquart, P. (2011) Experimental aerodynamic study of a car-type bluff body. J. Experiments in fluids, 50: 1273-1284.
[2] Martinat, G., Bourgeut, R., et al. (2008) Numerical simulation of the flow in the wake of Ahmed body using Detached Eddy Simulation and URANS. In: 2nd Symposium on Hybrid RANS-LES Methods. Corfu, GREECE. 97: 125-131.
[3] Wang, Y., Wang, T., Li, S., et al. (2018) Aerodynamic Drag Reduction of Ahmed Model Based on Free form Deformation. J. Transactions of Beijing Institute of Technology, 38: 221-228.
[4] Chen, X., Li, M., Ruan, X., Wang, N., Wang, J. (2019) Vortex Structure of Ahmed Vehicle Wake Based on Submersion Element Method and Delayed Separation Vortex Model. J. Journal of Jilin University (Engineering and Technology Edition). 49: 1034-1042.
[5] Hu, Y., Yang, X. (2017) Flow Field Analysis of Ahmed Model Tail Based on Unsteady State. in: Chinese Mechanics Conference. Beijing, China. 1890-1896.

[6] Cui, W., Yang Z., Wang G., Zhou, H. (2017) Analysis of unsteady wake flow of three dimensional vehicle with different rear angles. J. Journal of Jilin University (Engineering and Technology Edition). 47: 717-724.