Flow field analysis of Ahmed model based on URANS

Huajie Wu\textsuperscript{1,3} and Shanwen Zhang\textsuperscript{2}
\textsuperscript{1} Yangzhou polytechnic institute, Jiangsu 225127, China
\textsuperscript{2} College of Mechanical Engineering, Yangzhou University, Jiangsu 225127, China
\textsuperscript{3} E-mail: whj_china@163.com

Abstract. In order to obtain the flow field characteristics of the Ahmed model, flow field analysis is carried out by using standard k-\(\varepsilon\) turbulent model and URANS method. Results show that the flow field of Ahmed model presents the transient characteristics of periodic changes, the base vortex forms on the Ahmed model while the fluid is detached from the lower surface, and transports periodically downstream; the corner vortex formed on the Ahmed model is induced by the inclined pats of the tail-dip.

1. Introduction
With the gradual popularization of automobiles in our life, many countries have invested heavily in the development of automobile industry. However, in the process of manufacturing automobiles, many scholars pay more attention to the aerodynamic characteristics of automobiles. The automobiles satisfied the aerodynamic characteristics not only conform to the aesthetic standards in appearance, but also can effectively reduce the automobile air-resistance and aerodynamic noise, and improve the overall performance of the automobiles. Therefore, the aerodynamics characteristic is the primary factor to consider in automobile production. The flow field in vehicle dynamics is a typical blunt body flow problem, and the flow field structure has the characteristics of unsteady and three-dimensional effects. Wind tunnel test is one of the important methods to study automotive aerodynamics. It can optimize automotive shape through repeated tests, and ultimately obtain better automotive shape parameters. However, the wind tunnel test has the disadvantages of long test cycle and large investment. To shorten the research cycle and save the cost, the flow field analysis of automobile under different working conditions by using CFD software and the corresponding parameters are adjusted. Reasonable post-processing of the simulation results is carried out and the relevant physical information at all times or at all locations can be obtained \cite{1-7}.

Yiping Wang, et al. have used the RANS/LES mixed mode to calculate the mean external flow field for the vehicle, and taken the k-\(\varepsilon\)-\(\nu^2\) turbulence model with the three equations as the RANS part of RANS/LES mixed mode. The results showed that the hybrid model can more accurately and efficiently simulate the air separation on the body surface and the tail flow field under the same calculation conditions \cite{8}. Liwei Dong, et al. have researched the influence of different mesh densities on the simulation of vehicle external flow field based on Mira international standard model, and which is compared with the wind tunnel test data. The results showed the hybrid schemes of the three prisms, tetrahedron and hexahedron can effectively improve the accuracy of simulation \cite{9}. Juan Qian, et al. have taken the passenger car model as the research object, and used Catia and Fluent software to analyze the three-dimensional external flow field and optimize vehicle models. The results reflected that the aerodynamic drag coefficient could decrease through numerical simulation and aerodynamic
characteristics analysis of the optimized model, which provided the basic reference for automobile shape optimization [10]. Nan Shao, et al. have had a research into the flow and vortex structures around vehicles during overtaking maneuver with lift force included. The simulation demonstrates how the flow and vortex structures generate and change, which influences the aerodynamical coefficients of the vehicles. In this study, a transient solver was used on the platform of ANSYS/Fluent, and the sliding mesh method was adopted to simulate the motion of the overtaking vehicle, which realized a more actual recovery of the road conditions [11]. Kuthada Timo, et al. have studied the effects of cooling air on the flow field around a vehicle, and drawn a conclusion that cooling air not only affects the internal flow of the vehicle but also changes the flow around it. They have changed in the flow field around the generic DrivAer model resulting from cooling air flow, especially in the wake behind the car and in the region around the front wheels [12].

However, establishing a reasonable mathematical model to capture the characteristics of external flow field is one of the main difficulties in vehicle external flow field simulation. Many methods can not accurately capture the model, for example, the RANS (Reynolds averaged simulation) method is difficult to capture the typical flow characteristics of the flow field [13]. Although LES (desert simulation) method or the combination of LES and RANS can improve the calculation accuracy and capture many flow characteristics, it takes a long time and requires a high level of computer hardware resources [14-15]. Therefore, we use the URANS (unsteady Reynolds time-averaged simulation) method to simulate the external flow field of the tail Ahmed model [16] with an inclination of 25°. Compare the predicted information of the time-averaged flow field with the experimental data, and analyze the unsteady flow field in a typical period, which provides a reasonable and reliable simulation method for the optimization design of automobile appearance.

2. Introduction of Ahmed model
Ahmed model is a geometric model similar to car body, which was specially designed by SR Ahmed in 1984 to study vehicle wake [16], whose test data are widely used to verify the reliability of the simulation program for the external flow field of automobiles. It has a simple geometric shape, its length is 1.044 meters, width is 0.389 meters and height is 0.288 meters. At the same time, its bottom is also designed with a 0.5-meter cylindrical foot, and the rear surface is a slope of 40°. In the validation model of flow through Ahmed body, the slope of Ahmed body is 25°, and it is placed in the space of 8.352 m × 2.088 m × 2.088 m, which is used to calculate the flow field of Ahmed body. The Ahmed model is shown in Figure 1, φ represents the tail inclination.

![Figure 1. The Ahmed model.](image1)

![Figure 2. The boundary conditions of the Ahmed model.](image2)

3. Boundary conditions and solution settings
According to the characteristics of automobile flow field, Fluent module is used to simulate the flow field. According to the existing test conditions [16], the inlet boundary is set as the velocity inlet, and the velocity is 40 m/s. Then the outlet boundary is set as a pressure outlet, and the pressure value equals
a standard atmospheric pressure. The wall and floor surface of Ahmed model are set as non-slip wall boundary conditions. The top and two sides of the computation domain are set to slip walls. It can discretize the governing equations by finite volume method. The pressure terms use second-order scheme. The velocity, turbulent kinetic energy and dissipation rate all use the second-order upwind scheme. The transient terms adopt second-order implicit scheme. The results of steady-state flow field calculation are taken as the initial field of unsteady flow field, and the convergence criterion of residual error of each variable is set to $10^{-4}$. To ensure the results are convergent at each time step, the maximum iteration step is set to 40. When the flow field varies periodically, the corresponding variables are sampled and counted. The boundary conditions are shown in Figure 2.

4. The analysis and discussion of the results

4.1. Analysis of time-averaged flow field in tail region

In order to compare with the experimental results, the four cross-sections of a, b, c and d in the computational domain are intercepted, as shown in Figure 3. Here, cross-section a coincides with the rear surface of Ahmed model, and its position is $x = 0$ mm; cross-section b is $x = 80$ mm, cross-section c is $x = 200$ mm; and cross-section d is $x = 500$ mm. In Figure 4(a) - 4(d) are the experimental measurements value of the velocity component $U$ on the corresponding cross-sections, and also the velocity vector distributions on the cross-section are shown in Figure 4(e) - 4(h). On cross-section a, the velocity component $U$ has no negative value, it means that there is no reflux of the fluid at this point. And the fluid vortices on two sides above the tail is captured successfully. On cross-section b, the value of velocity component $U$ appears negative, and there is reflux of fluid. And the velocity vector diagram shows that the vortex developed from upstream still exists here, and its influence area has a tendency to expand. The velocity distribution predicted by simulation basically coincides with the experimental results. At the cross-section c, the distribution of velocity component $U$ changes significantly. The negative value region begins to decrease, and the vortices on two sides develop towards the ground and there are abundant flow phenomena. The simulation results at this section coincide with the experimental results, and the turbulence model can capture the trend of flow preferably. At the cross-section c, this is far from the tail of the Ahmed model. And the reflux region disappears basically, the development range of the vortices on two sides is further extended to the floor region, and the $U$-value low-speed region at the center of the vortices on two sides presents concave distribution characteristics. And this result is consistent with the experimental results.

To further study the development of vortices on two sides of Ahmed model, it needs to process the time-averaged velocity field of simulation results for obtaining the distribution of vortices in the x-axis direction. The expression of this vortex component $\omega_x$ is:

$$\omega_x = \frac{\partial W}{\partial y} - \frac{\partial V}{\partial z}$$

where, W and V respectively represent the components of velocity in the z and y directions of the coordinate axis. Figure 3 shows the mean vorticity component x distribution of Ahmed model on different cross-sections. Figure 5(a) shows the vorticity components on two sides of the straight section for the Ahmed model are zero, it means that no vortices are generated here. Figure 5(b) shows the vorticity $\omega_x$ shows a positive and negative regional distribution on two sides of the tail tilt position for Ahmed model, and the physical meaning of positive and negative signs is the opposite direction of vortice rotation. Figure 5(c) shows the distribution of positive and negative vorticity components in the downstream of the tail tilt section for Ahmed model is further enlarged, while the intensity of the vortices is not weakened, which indicates that the vortices are developing continuously. Figure 5(d) shows the influence area of the vortices on two sides continues to expand on the cross-section of a distance from the tail of the Ahmed model, but the intensity of the vortices decreases significantly. Therefore, the tail inclined section of the Ahmed model is the key factor to generate vortex on two sides of Ahmed model. After the vortex is formed, it develops along the title inclined section. Although the vortex affected area leaved the Ahmed model is expanding, the intensity gradually decreases.
Figure 3. Tail section distribution of Ahmed model.

Figure 4. Time-averaged velocity distribution at tail section of Ahmed model.

Figure 5. The mean vorticity distribution of Ahmed model on different cross-sections.

Figure 6 shows the time-averaged pressure distribution on the symmetric surface of the Ahmed model. In the head area of the Ahmed model, the pressure decreases rapidly from the maximum to the minimum negative pressure; in the straight section of the model, the pressure on the lower surface is increasing monotonously. However, the pressure on the upper surface gradually decreases, and a negative pressure value appears again under the influence of the tail inclined part. So the upper and lower surface pressure curves do not coincide, and the area enclosed by the lower surface pressure curve and transverse axis is larger than that enclosed by the upper surface pressure curve and transverse axis. That's the reason why Ahmed gets lift under the action of aerodynamic force.

4.2. Analysis of development law of transient flow field

The condition for the lift coefficient $C_L$ of the Ahmed model is time-variation, as shown in Figure 7. The lift coefficient defined here is:

$$C_L = \frac{F_z}{0.5 \rho U_{in}^2 A_x}$$  \hspace{1cm} (2)$$

where, $F_z$ represents the lift of Ahmed model, $\rho$ is the air density, $U_{in}$ is the velocity of the inlet fluid, and $A_x$ is the windward area of Ahmed model in the x direction.
The steady-state calculation results are used as the initial flow field of transient calculation. When computing to a certain time step, $C_1$ represents periodic distribution. When carrying out the steady-state calculation, the mass residual can only converge to the order of $10^{-2}$, and the resistance and lift coefficients monitored fluctuate periodically with the iteration step. However, the resistance and lift coefficients of steady-state calculation are different from the time averaged statistical results obtained from transient calculation. Therefore, it can be seen that the working condition is essentially a flow field with transient characteristics, so the URANS method is more practical than the RANS method.

![Figure 6. The time-averaged pressure distribution on the symmetric surface of the Ahmed model.](image)

![Figure 7. The change of Lift coefficient with time and its distributions in a typical period](image)

The typical cycle from the time domain of the computation is selected, and the detailed analysis and discussion are carried out. The dates from 0.45s to 0.49s in Figure 8 are extracted and shown in Figure 8. Take the corresponding moment at the minimum value of lift coefficient as the starting point of a typical cycle, that is the A point in Figure 8. Then select respectively the 1/4 cycle, 1/2 cycle and 3/4 cycle period to analyze and these moments correspond to B, C and D points in Figure 9 respectively.

Figure 8 shows the streamline distribution of Ahmed symmetric plane at different times. At different times, the upper and lower surfaces of air flowing through demonstrate two-dimensional characteristics, but the flow in the near tail region presents three-dimensional characteristics. The flow structures are complex and changeable, and vortices are formed here and continue to transport downstream. There are two vortices in the tail region at A point; Up to B point, the area coverage of the upper vortex expands and the lower vortex falls off from the lower surface of the Ahmed model; The upper vortex still exists, and the lower vortex has been transported downstream at C point; At D time, The lower vortex transported a distance to the downstream., and the surface of the Ahmed model has begun to generate new vortices at this point.

Figure 9 shows the distribution of the vorticity component $\omega_y$ on the symmetric plane at different times, it is defined as:

$$\omega_y = \frac{\partial U}{\partial z} - \frac{\partial W}{\partial x}$$  \hspace{1cm} (3)

where, $U$ and $W$ respectively express the components of velocity in the x-axis and z-axis. There are two vortices component $\omega_y$ of positive and negative regions in the flow field of the tail. The positive value indicates that the vortex rotates clockwise, and the negative value indicates that the vortex rotates counterclockwise. At the four moments of A, B, C and D, the positive vorticity region above is relatively stable, while the negative vorticity region below is obviously changed. The analysis shows that the lower vortex is generated when the fluid is detached from the lower surface, and transport periodically downstream.
Figure 10 shows there are three typical vortices in the external flow field of Ahmed model, which is the top vortex, the corner vortex and the base vortex [16].

![Figure 8](image1.png)  
**Figure 8.** The streamline distribution of Ahmed symmetric plane at different times.

![Figure 9](image2.png)  
**Figure 9.** Distribution of symmetric surface vorticity at different times.

To identify the vortex structure in the flow field, the Q criterion is adopted to carry out the post-processing of the simulated data. The Q criterion is defined as:

$$ Q = \frac{1}{2}(\Omega^2 - S S) $$

where, $S = S = \frac{1}{2} \left( \frac{\partial u}{\partial x_i} + \frac{\partial u}{\partial x_i} \right)$ represents strain rate tensor; $\Omega = \frac{1}{2} \left( \frac{\partial u}{\partial x_j} - \frac{\partial u}{\partial x_j} \right)$ represents vorticity tensor. When the Q value is greater than 0, the rotation of the fluid dominates, and the corresponding region is the vortex structure.

![Figure 10](image3.png)  
**Figure 10.** Typical vortex structure of Ahmed external flow field.
5. Conclusions
The URANS (unsteady Reynolds time-averaged simulation) method is used to simulate the flow field of the Ahmed model with a 25° inclined angle at the tail, and the time-averaged flow field and transient flow field are analyzed. The conclusions are as follow:

1) The difference of time averaged pressure distribution on the symmetric surface of Ahmed model is the reason why the Ahmed model is lifted by aerodynamic force.

2) At different times, the upper and lower surfaces of air flowing demonstrate two-dimensional characteristics, but the flow in the near tail region presents three-dimensional characteristics. The flow structure is complex and changeable, and vortices are formed in the near tail region and continue to transport downstream.

3) At different moments, the top vortices adhere more stably to the intersecting line of the straight and inclined sections of the Ahmed model. The bottom vortex is induced by the bottom of the model. And the corner vortices on two sides are conical and expand from the smaller vortices on the top to the downstream.

References
[1] Tao Zhou and Zhong Zeng 2017 Aerodynamic optimization design of new front wing fins for FSAE racing car Journal of Chongqing University 40(10) 40-52
[2] Hui Zhu, Zhihang Yang, Peng Tan, et al. 2016 Comparative study on simulation ability of fluid simulation platform for vehicle external flow field Automotive Engineering 38(2) 163-167
[3] Feng Zhang, Jian Wang, Xia Huang, et al. 2016 Numerical simulation and optimization design of vehicle external flow field Machinery Design & Manufacture 223-227
[4] Zhitao Liao, Yiping Wang, Hao Li, et al. 2016 Mechanism of dust pollution on the back of a minibus based on CFD. Journal of Wuhan University of Technology (Transportation Science & Engineering) 40(3) 564-568
[5] Furen Zhang, Jinlong Zhang, Xian Qu, et al. 2015 Analysis of aerodynamic characteristics of external flow field under crosswind Science & Technology Review 33(15) 76-81
[6] Xingping Long, Qi Liu, Xiaofeng Ruan, et al. 2016 Numerical analysis of internal flow of abrasive entrained waterjet nozzle Journal of Drainage and Irrigation Machinery Engineering 34(8) 686-692
[7] Bo Wang and Manhua Zhang 2015 Contrast of One-dimension Thermal Management Model Calibration based on CFD and Vehicle Test Automobile Technology 15-18
[8] Yiping Wang, Zhengqi Gu, Yadong Deng, et al. 2012 Aerodynamic Simulations of Vehicle by Using the Hybrid Semi k-ε-v²/LES Model Journal of Mechanical Engineering 48(14) 97-103
[9] Liwei Dong, Zhengqi Gu, Shuichang Liu, et al. 2012 Effect of Meshing on Numerical Simulation of External Flow Field around Vehicle Automobile Science & Technology 12-15
[10] Juan Qian, Dongfang Wang, Xiaodong Miao, et al. 2016 Numerical simulation and model optimization for car body based on CFD Manufacturing Automation 38(4) 74-76+92
[11] Nan Shao, Guofeng Yao, Chang Zhang, et al. 2017 A research into the flow and vortex structures around vehicles during overtaking maneuver with lift force included Advances in Mechanical Engineering 9(9) 1-16
[12] Kuthada Timo, Wittmeier Felix, Bock Benjamin, et al. 2016 The Effects of Cooling Air on the Flow Field around a Vehicle SAE International Journal of Passenger Cars- Mechanical Systems 9(2) 723-732
[13] Xun Liu and Chenguang Lai 2013 Numerical Simulation Research of External Flow Field on Ahmed Model Journal of Chongqing University of Technology (Natural Science) 227(9) 122-127
[14] Eric Serre, Matthieu Minguez, Richard Pasquetti, et al. 2013 On simulating the turbulent flow around the Ahmed body: A French–German collaborative evaluation of LES and DES Computers & Fluids 78 10-23
[15] E Guilmineau, G B Deng, P Queutey, et al. 2017 Assessment of Hybrid LES Formulations for
Flow Simulation Around the Ahmed Body *Turbulence and Interactions* (135) 171-176

[16] H Lienhart and S Becker 2003 Flow and Turbulence Structure in the Wake of a Simplified Car Model *Communication Law & Policy* 8(1) 25-110