# Simulation for Vehicle Rear Eddy Based on Models of Different Rear Angle

Zhang Jin-long
School of Mechanical &Automotive Engineering
Chongqing Jiaotong University
China

Qu Xian
School of Mechanical &Automotive Engineering
Chongqing Jiaotong University
China

***Abstract*****—The aerodynamic around automobiles become complicated because of the vehicle rear. Study on aerodynamic influenced by different vehicle rears is more and more cared. It is one important part in vehicle aerodynamic study. And, it is great significant to optimize the vehicle shape and design vehicles with low drag coefficient. On this paper, The vehicle models with different rear angles are created by a three-dimensional modeling software—CATIA. Then the models are analyzed for aerodynamic around automobiles by Star-ccm+ which is a computational fluid dynamic (CFD) software. The drag coefficient and lift coefficient of air are simulated for models with different rear angle. The separation of air at the vehicle rear is also simulated. The effect is analyzed that the different rear angle influence on the air separation and eddy becoming at the vehicle rear. With the analysis, the vehicle rear of -10° can put off the air separation and the location of rear eddy. Compared with the other models, it has better aerodynamic charter.**

***Keywords—vehicle rear angle ; numerical simulation ; air separation ; rear eddy***



## I. INTRODUCTION

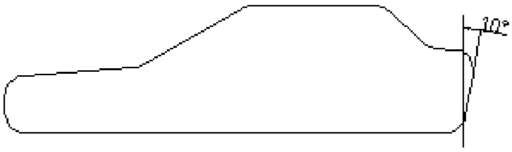
With the improvement of modern highway, automotive engine, transmission technology the speed is greatly improved when a car is on the highway traveling. Then, the problem of air resistance, lift are followed. This has caused major car manufacturers concerned. When a car is at high speed, the economy, power, safety, comfort and some other performances will be influenced by the air resistance and lift directly. In the car's air resistance, a large part is caused by the pressure drag which is due to the airflow separation and formation of the rear eddy in the car rear.

Lots of researches have been done by scholars at home and abroad and fruitful results have been achieved. Professor FU Limin , Jilin University, who made a large number of experiments. The formation mechanism of vehicle rear eddy is summarized and the vehicle rear eddy is divided into several different models. In addition,it also pointed out the impact of the the rear window angle to the rear eddy[1-2]. TAN qun made some generic description on the process and results of CFD calculation to the external flow field around vehicles. The flow separation of the external flow field at the automobile rear is simulated[3]. XU Jian-min made a three-dimensional numerical simulation of steady flow around the automobile to provide a basis for optimizing the car aerodynamic characteristics design by software FLUENT[4]. Professor DU Zi-xue[5], Chongqing Jiaotong University, researched the influence of different windshield angles, rear window angles, approach angles, departure angles to the the car air drag coefficient,lift coefficient. In the simulation, the windscreen airflow at the top and the airflow's in and out at the bottom of the chassis is researched. The impact from the rear angle when simulated the separation of airflow at the vehicle's rear is not studied.Using three-dimensional modeling software CATIA, the simplified models of different car rear angle is constructed. And then the simulation calculations is done by the CFD software Star-ccm +, the impact of different rear angles to the car rear airflow separation is analyzed. Because this is a quantitative exploratory experiment so that the rear angle is set only to show the three different direction of the rear[6-8].

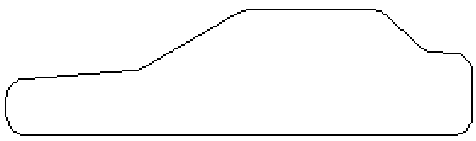
## II. BODY MODELS ARE ESTABLISHED

### A. *Simplified body model*

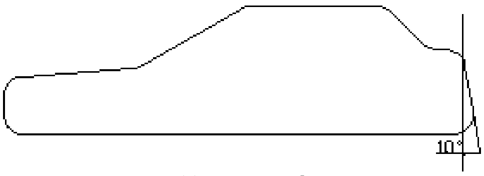
A car is covered with a variety of surfaces. These surface have different curvature and are very complexity. So,the rear angle is only studied and the separation at the car's rear is cared. Amount of simplifications are done when build the body models. Referring to the shape of a vehicle, surfaces out of the car are substituted for plane. Some rugosity such as doors, door handles and windows are overlooked. Even the wheels are omitted so that the role of the rear can be stressed. In order to ensure comparability, dimensions of the model's front window, side windows, roof, length, overall width, overall higher remain unchanged, only three directional changes are set by the model rear angle (angle between the rear with vertical). The angles are -10 ° , 0 ° , 10 ° . As shown in Figure 1.




(*a*) -10°rear angle

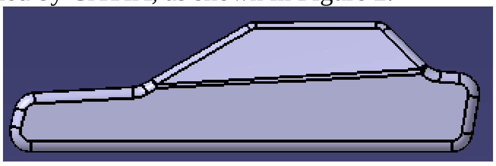


(*b*) 0°rear angle

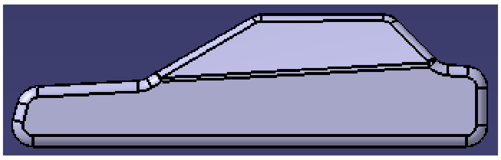



($c$)10°rear angle
Fig.1 Rear inclination parameter

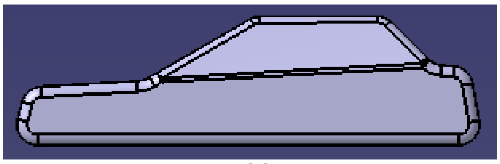
Simplified three-dimensional models of vehicle are established by CATIA, as shown in Figure 2.



model *a*



model *b*



model *c*
Fig.2 Body model

*B. Mesh body model*

As the body is symmetrical along the longitudinal profile, only the half body are took to mesh for saving memory and reducing computing time. The bodies'maximum grid is set to 12mm, and minimum 1mm; the maximum surface grid of computational domain is 300mm, minimum 100mm. the mesh is set to 5 floors in the non-slip field. And the total thickness of the boundary layer is 1mm. As shown in Figure 3, 2.7 million hexahedral core Trim grids are generated.

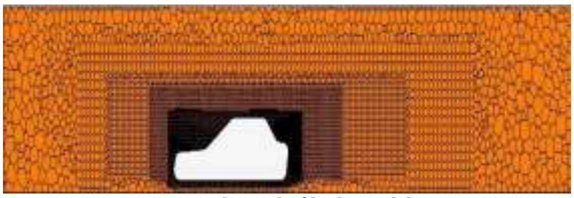


Fig.3 The grid of body model

## III. CFD SIMULATION

*A. Equations and turbulence model*

The airflow around car is followed the law of conservation of mass, conservation of momentum, conservation of energy. The N-S equations of automotive exterior flow field can be drawn from these laws. Those are the continuity equations, momentum equations, energy equations. The mach number of air current outside the car is less than 0.3 so that the air can be regarded as incompressible fluid. That is to say the equations of Reynolds averaged Mass and momentum exist.

$$\partial(\rho u_j)/\partial x_j = 0 \quad (1)$$

$$\partial(\rho u_j u_i - \tau_{ij})/\partial x_j = -\partial p/\partial x_{ij} + s_i \quad (2)$$

Where, $S_i$ is the source term and $\tau_{ij}$ is the stress tensor. To Newtonian fluids, a formula is below.

$$\tau_{ij} = 2\mu(s_{ij} - \frac{1}{3}\partial u_k/\partial x_k\, \delta_{ij}) - \overline{\rho u_i' u_j'} \quad (3)$$

Where,μis coefficient of the air molecular dynamic viscosity ; δij is Kroneker number; $\overline{\rho u_i' u_j'}$ is Reynolds stress tensor; Sij is fluid deformation rate tensor and given by the following formula:

$$s_{ij} = \frac{1}{2}(\partial u_i/\partial x_j + \partial u_j/\partial x_i) \quad (4)$$

In order to close the airflow control equation, turbulence model of SST Ⱪ-O was used in this experiment. The control equations are followed

Ⱪ Equation for the turbulent kinetic energy:

$$\partial(\rho\kappa)/\partial t + \partial(\rho u_i \kappa)/\partial x_i = \partial[(\mu + \mu_t/\sigma_\kappa)\partial\kappa/\partial x_j]/\partial x_j + G_\kappa - Y_\kappa \quad (5)$$

$o$, the mean square value of Turbulent vorticity, is expressed as:

$$\partial(\rho\omega)/\partial t + \partial(\rho u_i \omega)/\partial x_i = \partial[(\mu + \mu_t/\sigma_\omega)\partial\omega/\partial x_j]/\partial x_j + G_\omega - Y_\omega \quad (6)$$

The wall function is applied to simulate the phenomena in the boundary layer accurately and capture air separation outside the wall layer. This is applicable to the analysis of flow field in the rear car.

*C. Boundary conditions*

To save calculation time,the simulation domain refer to the literature [5]: the distance between entrance and the front of the car is 2 times the vehicle length, and exit from the rear end of the car is four times longer. The whole width of the computational domain is 2.5 times the vehicle width. The whole height is set to five times the vehicle height. Automotive exterior flow field boundary conditions is set ,as shown in table 1:

TABLE I BOUNDARY CONDITIONS

| | |
|---|---|
| Velocity inlet | 100Km/h |
| Pressure outlet | 0 gradient |
| Top and sides of the computational domain | Slip |
| Other surfaces | No-slip |
| Relaxation factor | Velocity: 0.7, Pressure: 0.1 |

## IV. NUMERICAL SIMULATION ANALYSIS

*A. Comparison of air drag coefficient and lift coefficient*

From the simulation, the air drag coefficient (*Cd*) and lift coefficient (*Cl*) of each car model can be obtained. They are shown in table 2:

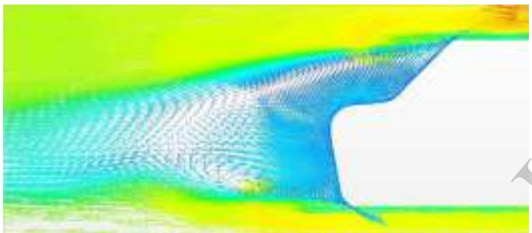
Tableп Air Drag Coefficient And Lift Coefficient

| | Model a(Rear tilt-10°) | Model b(Rear tilt0°) | Model c(Rear tilt10°) |
|---|---|---|---|
| Air drag coefficient (Cd) | 0.216 | 0.261 | 0.227 |
| Air lift coefficient (Cl) | -0.277 | -0.356 | -0.256 |

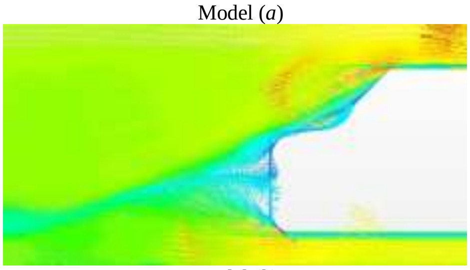
As can be seen from the table, the air drag of model a approximates that of model c,but the aerodynamic characteristics of model a is slightly better. Air drag coefficient of model b is significantly larger than the other models. And lift coefficient is superior to the others.

*B. Analysis of the impact of the rear car to air separation and vortex*

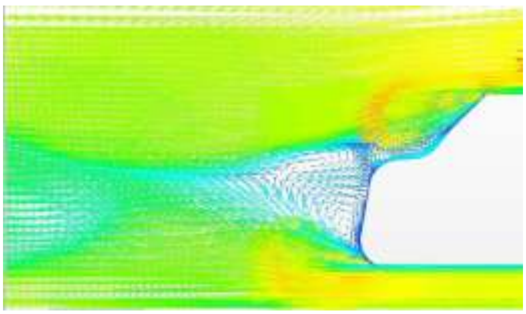
The quality of the three vehicle models aerodynamic characteristics are reflected by the air resistance coefficient and lift coefficient. In order to analyze the impact of a car rear angle to the rear vortex, and find how it works to air drag coefficient and lift coefficient, the car's velocity vector at the rear is focused in the post-processing. Then, the separation of the airflow at the rear can be observed. The velocity vector is shown in Figure 4.



Model (*a*)



Model (*b*)



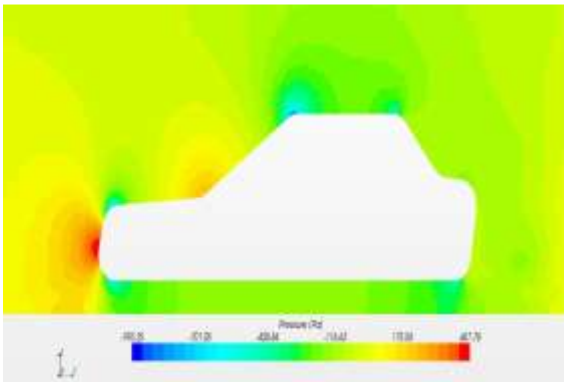
Model (*c*)

Fig. 4 Rear airflow speed vector

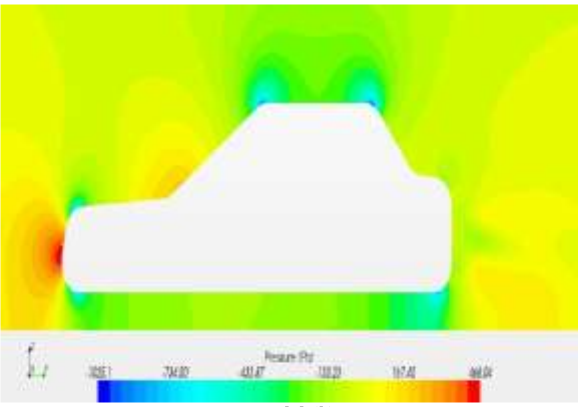
As can be seen from the figure, the top airflow began to separate at the top of the rear window. It is confluent with the airflow separating from the vehicle bottom at the rear. Two vortexes are formed at the car rear. One is up and the other is down. However, some differences about these rear vortexes for the changes of car models' rear angle.For model a ,the velocity gradient of air separation from the top and bottom is smaller than the other two because the affect of a negative beveled angle. Then, air separation is reduced. Because of the negative beveled angle, the whole separated is downward. And vortexes is formed far away from the rear. The occurrence of wake vortex is delayed. That is why the aerodynamic drag of model a is better the the other two. To model c, The velocity gradient of air separation from the top and bottom are both slightly larger than model a. Meanwhile, the separated airflow has a upward trend due to the angle of rear so that the location of the rear vortex is close to the rear. It causes the upper airflow obstruction and another vortex are formed between the rear window and deck lid. All of these make the aerodynamic characteristics of the model c is somewhat worse than the model a. Affected by 0° rear angle, the velocity gradient of air separation from the top and bottom is very large. The airflow separation in the rear of the vehicle is severe. The separated airflow can't be guided to up or down since no rear slope. So the trailing vortex forms at the rear of model and a large negative pressure zone then forms. Due to the longer rear, another vortex forms between the rear window and deck lid. All these lead to model c the worst aerodynamic drag of the three.

Figure 5 is pressure contour of the vehicle longitudinal symmetrical plane. As we can see from the figure, the pressure of the front vehicle, roof and bottom of the three model are the same. At the rear of the model, the negative pressure of model a at the top of rear window is smallest of all and model b is largest. At rear bottom of the model, the negative pressure of model a is also smallest and the other much larger. This is consistent with the airflow separation of bottom and rear window top at the model rear. The pressure between front and rear make the drag of model a smaller the other two. From top to bottom, the top negative pressure at the model c rear is larger than that of model a. And there is a big negative center at the top rear of model b. But ,at the bottom rear, model c negative pressure is larger. Integrating the negative pressure up and down, the lift characteristics of model a is slightly better with the model c. Model b has a large negative pressure at bottom rear and small between the rear window and deck lid. That is why the lift characteristics of model b is the best. This case coincides with the strength of airflow separation at the models rear.

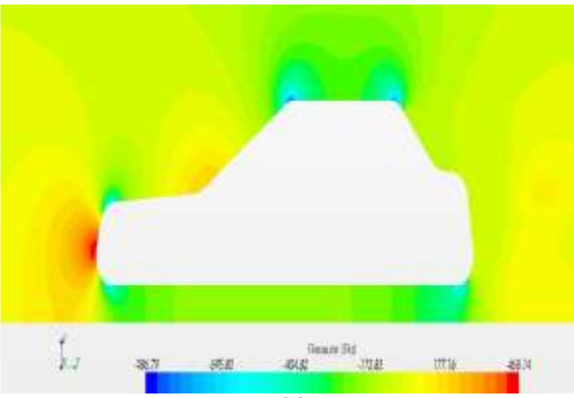




Model 5*a*



Model (*b*)



Model (*c*)

Fig.5 The pressure nephogram of vehicle longitudinal plane

## V CONCLUSIONS

When the rear angle is downward (-10°), the airflow separation is weak at the car rear. A leading role is brought by the negative angle of the rear which can make the trailing vortex away from a car. This lead to negative pressure at the bottom rear smaller,then the resistance characteristics of a car can be improved.

- When the rear angle is upward (10°),the airflow separation from underbody is strong at the model rear. This is why there is a large negative pressure zone at the rear end of a car. Because of the inclination at the rear, the separated airflow rises up and the automobile upper vacuum increases. This make the lift lift characteristics of a car dropped .

- When the rear angle is not inclined (0°),there is a large negative pressure region for the airflow separation at the rear of a car. Because no rear inclination exists, the flow separations become wake vortex at the rear of a car, which decline the resistance characteristics significantly. While the lift characteristics of the car be improved.

By changing the angle of the rear in this paper, the influence which a car rear inclination in 3 different directions act on the airflow separation and rear eddies at a car rear is analyzed to explain the how the rear angle affects the influence of the inclination of the direction of the rear three cars tail flow air drag coefficient and lift coefficient of a car. A direction of optimizing the design of a car rear have been find. And, a basis has been made for further research.



## REFERENCES

[1] Fu Limin, Zhang Qinfeng, Jin Chunning."The Mechanism and the Control of the Road Vehicle Wake Vortices Formation"[J].Automotive Engineering,2000(Vol22),pp.1: 13-16 .

[2] Fu Limin. Automobile Design andAerodynamics[M]. Beijing: China Machine Press, 2006

[3] Qin Qun, Li Shi-zeng, Wu Chun-ling, Huang Sen-ren. Numerical Simulation of the Vehicle External Flow with STAR-CCM+[J]. Manufacturing Automation, 2012-04,34(4) pp.151-153.

[4] Xu Jianmin, Yi Jiming, Zhao Jun, Ding Tao. Numerical Simulation of Flow Field around Streamline Car[J]. Journal of Shaanxi University of Science & Technology, 2011,29(5) pp.61-63.

[5] Du Zi-xue, Zhang Jie. Simulation for External Flow Field Around Automobiles based on Models of different Parametric Shape[J]. Journal of Chongqing Jiaotong University(natural science), 2011,30(4) pp.848-851

[6] Wagner B, Schmidt W. Computation of Automobile Aerody–namics by Use of Numerical Methods Developed in Aerodynamics Industry. SAE 8707 1 6.

[7] Wang Fujun. Analysis of Computational Fluid Mechanics[M]. Beijing: Tsinghua University press, 2004

[8] A. P. Gaylard, A. J. Baxendale etc, The Use of CFD to Predict the Aerodynamic Charateristics of Simple Automotive Shape, SAE 980036