Experimental and Numerical Investigations on the Influence of Vehicle Rear Diffuser Angle on Aerodynamic Drag and Wake Structure

Chenguang Lai 1), 2), Yasuaki Kohama 3), Shigeru Obayashi 4) and Shinkyu Jeong 5)

1) · 3) – 5) Tohoku University
2) Key Laboratory of Manufacture and Test Techniques for Automobile Parts (Chongqing University of Technology), Ministry of Education
No.69, Hongguang Rd., Baan, Chongqing 400054, P.R. China

Received on June, 28, 2010

ABSTRACT: Experimental and numerical studies are performed to evaluate and analyze the influence of the notchback rear diffuser angle on aerodynamic drag and wake structure. The relationship between aerodynamic drag and rear diffuser angle is summarized, and the flow mechanism are analyzed and discussed. A speculation regarding lower trailing vortices is proposed, and is verified in the present research model. Rear diffuser angle is an important factor influencing the wake structure, and optimizing the vehicle rear diffuser at a favorable angle can contribute to reduce the drag force and improve the wake structure.

KEY WORDS: (Standardized) Heat·Fluid, Aerodynamics, (Free) Rear Diffuser Angle, Aerodynamic Drag, Wake Structure, Lower Trailing Vortices [A1]

1. Introduction
The wakes of bluff bodies play important roles in their aerodynamic characteristics. As a passenger vehicle is a typical bluff body, the wakes behind these types of vehicle markedly influence their aerodynamic characteristics. Over the past several decades, many investigations of passenger vehicle wake structures have been carried out on the upper-rear region. Ahmed et al. 1) performed an experimental study of the effects of base slant angle of a hatchback shape on the time-averaged properties of the wake. They proposed a wake vortex system for hatchbacks. Subsequently, Nouzawa et al. 2) investigated the wake structure of the notchback type vehicle. They structured a flow pattern for notchbacks with no rear diffuser slant. Their results indicated that the aerodynamic drag could be increased by generation of the trailing vortex at the trunk-deck and the so-called arch vortex behind the backlight. These studies improved our understanding of the influence of the upper-rear shape on wake structure.

On the other hand, recent computational and experimental results indicated that underbody flow is closely related to the wake structure and aerodynamic force 3) – 5). Yoshihiro et al. 6) performed a numerical investigation of vehicle underbody aerodynamics and reported that the wake of a notchback vehicle could be modified by optimizing the positions of underbody obstacles. Most of these investigations are carried out on the influence of large rough underbody shapes, such as wheels and wheel housings, on aerodynamic drag and the wake, but few focuses on the rear diffuser. In most instances, rear diffuser is studied for racing cars to generate down force 7). Cogotti 8) performed a large-scale parametric study on a simplified car model to investigate the effects on aerodynamic coefficients produced by important geometric changes that affect the flows under the car. In addition to the wheels, wheel housings, overhangs, and ground clearances, the author also measured the drag and lift with 3 different rear diffuser angles. Although not sufficient to evaluate the tendencies in variation of aerodynamic coefficients related to rear diffuser angle, the test results indicated that the influence of rear diffuser angle on aerodynamic drag cannot be neglected. In fact, the rear diffuser is an important factor influencing the underflow and the wake. In addition, due to its ease and economy of modification compared with other underbody components, the rear diffuser is suitable for more elaborate study.

The purpose of this study is to evaluate and analyze the influence of the rear diffuser angle on aerodynamic drag as well as the wake structure. A series of experimental investigations regarding aerodynamic performance on different rear diffuser angles were conducted. To gain further understanding of the flow mechanism, numerical simulations were also performed to investigate the detailed flow fields and pressure distributions.

2. Research Approach
2.1. Models
The model for this research was determined taking into account the following considerations:

First, the model had to have a car-like shape and be both easy to build and easy to remodel in the future in case the same model is used for later research. (This is one of a series of studies on the combined underflow and engine-cooling flow.)

On the other hand, the model should include some important underbody components, such as wheels and wheel housings, as the rear diffuser is closely related to the underflow.
Taking into account the above discussion, a 1/8 simplified notchback model was constructed for this research. The model scale was determined by the blockage ratio of the wind tunnel experiment. The model is made up of several components, each of which can be installed and disassembled conveniently. Figure 1 shows the main shape parameters of the model. In the present study, the ground clearance $D$ can be adjusted to 20, 25, and 30 mm, and the rear diffuser angle $a$ is varied at 0°, 4°, 8°, 12°, 16°, and 20°.

It should be noted that the front edge of the model is sharp. The sharp edge is used to generate a turbulent boundary layer, because the flows around a passenger vehicle are turbulent. For bluff bodies, such as rectangular cylinders, flow separation occurs at the front sharp edge. Therefore, the boundary layer on the surface of a rectangular cylinder is turbulent. As the model in this study is a typical bluff body, the shape is complex and three-dimensional. Therefore, the sharp front edge can ensure that the boundary layer on the surface of the model is turbulent in the present study.

![Fig. 1 Main shape parameters of the test model](image1)

2.2. Wind tunnel

The wind tunnel utilized in this study was the low-turbulence wind tunnel at the Institute of Fluid Science, Tohoku University, Japan. A fixed plate was used to simulate the ground. To eliminate the influence of the boundary layer on the nozzle wall, the plate was installed downstream of the nozzle exit and 0.3 m higher than the lowest side of the nozzle. To reduce the thickness of the boundary layer on the plate, the leading edge of the fixed plate was processed into an ellipse. The Cartesian coordinate system was used, where $x$ denotes the freestream direction, $y$ denotes the spanwise direction, and $z$ denotes the height direction. The origin was fixed at the position of the rear end symmetry line of the models on the ground plate. A 3-component load cell was used to measure the drag, and a Particle Image Velocimetry (PIV) system was used to observe the velocity distribution. Oil visualization was also performed to observe the flow pattern on the rear diffuser plane. The experiments were carried out at a nominal freestream velocity of 30 m/s, which corresponds to a Reynolds number based on model length of $1.03 \times 10^6$. To reduce the influence on underflow, the model support strut was set at the top of the model (see Fig. 2).
Fig. 3 Aerodynamic drag coefficients vs. rear diffuser angle at 3 different ground clearances

Fig. 4 Time-averaged velocity vectors in the longitudinal central section of the wakes

3.2 Velocity vector

To confirm the structure of the dominant fluid motion behind the models, a PIV system was used to measure the velocity distribution in longitudinal sections of the models at $D = 25$ mm. A pair of reversed vortices could be seen in the recirculation region of the wake, which Ahmed calls “separation bubbles” (1). With increasing rear diffuser angle, the junction position of the upper and lower flow tends to move upward and near to the trunk-deck, which may reduce the recirculation region and reduce the drag. The time-averaged velocity vectors in the longitudinal central section at 6 different rear diffuser angles are shown in Fig. 4.

To determine the size of the recirculation region, the freestream velocity component $u$ at the longitudinal central section is analyzed. The distribution of the flow position where the value of $u$ changes from negative to positive is shown in Fig. 5. From the figure, it can be seen that the recirculation region tends to move to the upper side and the length of the recirculation region decreases with increasing rear diffuser angle. The upper edge of the recirculation region is affected little by rear diffuser angle, while the lower edge of the recirculation region moves obviously upward with increasing rear diffuser angle. For these reasons, the underflow mass will be increased, and the decelerated velocity region in the wake is decreased. In this context, velocity recovery may be accelerated with increasing rear diffuser angle, which will contribute to a reduction in the aerodynamic drag.

Fig. 5 Recirculation region in the longitudinal central section of the wakes

3.3 Oil visualization

To observe the flow on the rear diffuser plane, oil film and drop visualization experiments were performed for the model with $D = 25$ mm. The photos for oil film and oil drop visualization are shown in Figs. 6 and 7, respectively.

The results of oil film visualization indicated that the flow on the rear diffuser plane can be divided into the middle field and lateral fields. Due to the influence of the rear wheels, the flow in the lateral fields is complex. However, it also tends to arch the flow from the front edge corners of the rear diffuser plane to the middle field with increasing rear diffuser angle. The middle field flow shows a clear trend when rear diffuser angle is smaller than 16°, i.e., with increasing rear diffuser angle, the flow passes through the rear diffuser plane by converging toward the rear center of the plane. This trend can also be clearly observed in oil
drop visualization as shown in Fig. 7. These results indicate that there is a lateral-to-central shear flow on the rear diffuser plane, and the velocity of the shear flow increases with increasing rear diffuser angle. The spanwise shear flow can contribute to the inrush of lateral flow into the underbody, and increases the mass of underflow.

In addition, due to the continual increase in spanwise velocity with increasing rear diffuser angle, it is speculated that, if this velocity is large enough, it may form a pair of trailing vortices in the wake region behind the separation of underflow, and the directions of these trailing vortices should be counterclockwise on the left and clockwise on the right (in the view from +x to –x direction).

When \( \alpha = 16^\circ \) and \( 20^\circ \), the oil films in the middle field become blurred. From Fig. 7, it can be seen that the front edge oil drops in circle 1 are no longer moved downstream, and the middle oil drop in circle 2 is moved both upstream and downstream. These observations indicate that the underflow in the middle field may separate at the front edge of the rear diffuser plane, and then reattach at the rear part of the plane. When \( \alpha = 20^\circ \), most of the front edge oil drops are no longer moved downstream, and most of the middle oil drops are moved upstream. These observations indicate that the flow separation is greater at this rear diffuser angle, and the area of reattachment is moved further downstream or remains separated from the rear diffuser plane. This is also the reason why the oil film becomes blurred at \( \alpha = 16^\circ \) and \( 20^\circ \). The flow separation will markedly increase the drag, which is one of the main reasons for the increase in drag with increasing rear diffuser angle above \( 12^\circ \).

The above visualization results support the results of the PIV observation: by increasing the rear diffuser angle, the mass of the underbody flow is increased, which can accelerate the velocity recovery in the wake region, and decrease drag force. However, this positive effect may be eliminated by the separated flow on the rear diffuser plane, and by a pair of lower trailing vortices generated after the underflow enters the wake region at large rear diffuser angle.

4. CFD Results and Discussion

4.1. Comparison between CFD and experimental results

Before discussing the CFD results of the flow details and pressure distribution, it is worth comparing the aerodynamic drag coefficients between CFD and experimental measurements. Figure 8 shows the comparison of drag vs. rear diffuser angle between CFD and experimental results.

As can be seen in Fig. 8, the CFD and experimental results are very close, with a maximum drag coefficient difference of 0.039 at \( \alpha = 8^\circ \). The tendency of drag related to the rear diffuser angle shows quite good agreement with the experimental results. With increasing rear diffuser angle, the drag first decreases and then increases, and both the minimal drag coefficients are obtained at \( \alpha = 12^\circ \). In general, the CFD results of drag force are slightly lower than the experimental results. This may be because the CFD simulation has no flow interference from the wind tunnel.
model support strut. (the experimental results was obtained by subtracting the forces of the separate strut. But it seems that there were also some induced forces generated due to the flow interference of the support strut.)

The comparisons implied that the numerical technique employed in the present simulation is adequate for observation of the flow details around the model and the pressure distribution on the model surface.

4.2 Wake details

Nouzawa et al. (2) proposed the vortex system for a notchback model as shown in Fig. 9. This pattern shows that the flow field behind this notchback is dominated by counter-rotating longitudinal vortices generated at the trunk-deck and an arch vortex behind the backlight. Notice that the rear diffuser angle of his model is 0°.

![Fig. 9 Proposed vortex system for a notchback model by Nouzawa et al. (2)](image)

The wake details of the simulations indicated that when \( \alpha = 0^\circ \) the wake structure for the present notchback model is in good agreement with Fig. 9. In this case, the main underflow has a center-to-lateral spanwise velocity. When it enters the wake region, it can contribute to development of the counterclockwise trailing vortex generated at the trunk-deck. However, at large rear diffuser angle, the underflow of this notchback model markedly influences the wake structure. With increasing rear diffuser angle, the spanwise velocity of the main underflow tends to shift from lateral to center, which weakens the trailing vortex generated at the trunk-deck after the underflow enters the wake region. When \( \alpha \) is larger than 12°, due to the high lateral-to-center spanwise velocity, a clockwise vortex is generated behind the trunk-deck, and develops into a clockwise trailing vortex in the following wake region. This trailing vortex is defined here as the “lower trailing vortex”. In these cases, instead of the trailing vortex generated at the trunk-deck, the wake is dominated by the lower trailing vortex.

Fig. 10 shows the velocity distribution in some specified cross-sections of the wake at \( \alpha = 0^\circ \) (left) and \( \alpha = 20^\circ \) (right).

4.3 Pressure distribution

The experimental investigation showed that flow separation may occur at the front edge of the rear diffuser plane at large \( \alpha \). Therefore, the pressure distributions along the y direction (Fig. 11) and y pressure gradient \( \partial p/\partial y \) (Fig. 12) at the front edge of the rear diffuser plant were extracted.

For the middle part on the edge (the middle part is defined as \( y \leq 69 \) mm, and the lateral part is defined as \( 69 \) mm \( \leq y \leq 109 \) mm on the basis of the widths of the model and wheel housing), Fig. 11 shows that the pressure plummets from lateral to center at \( \alpha = 16^\circ \) and 20°, which means the flow separation has occurred in the middle part of the front edge of the rear diffuser plane. Due to the drop in pressure, a large pressure gradient is generated as shown in Fig. 12. This large pressure gradient brings a high spanwise shear flow on the rear diffuser plane, and drives the underflow moving from lateral to center by shear stress. With decreasing rear diffuser angle, the pressure in the middle part gradually increases and the pressure gradient is decreased. When \( \alpha = 0^\circ \), the pressure gradient in the middle part is negative, which means the airflow on the rear diffuser plane has a force to flow toward lateral. This is why the middle part airflow flows laterally at \( \alpha = 0^\circ \).
0°, while it is converged toward the center at larger rear diffuser angle.

For the lateral part, due to the influence of the rear wheel and wheel housing, the changes in pressure and pressure gradient are more complicated. Overall, the pressures tend to decrease along the y direction. However, there are no obvious differences among these rear diffuser angles.

![Fig. 11 Pressure distributions along the y direction at the front edge of the rear diffuser plant](image1)

![Fig. 12 Spanwise pressure gradients at the front edge of the rear diffuser plant](image2)

From the above discussion, it can be inferred that at large rear diffuser angle, due to the low pressure on the separation area, the spanwise pressure gradient increases sharply in the middle part of the rear diffuser front edge. This large pressure gradient brings a spanwise shear flow on the rear diffuser plane, and drives the underflow moving from lateral to center. The left and right underflows converge into the center part of the model rear end and form a vertical flow. When the underflow enters the wake region, the spanwise and vertical flows roll up into counterclockwise and clockwise lower trailing vortices on the left and right sides, respectively.

5. Conclusions

The following conclusions were made based on the experimental and CFD investigations presented here:

1. The aerodynamic drag of a vehicle can be influenced by rear diffuser angle. With increasing rear diffuser angle, the drag first decreases and then increases.

2. By increasing the rear diffuser angle, the mass of the underbody flow is increased, which can accelerate the velocity recovery in the wake region, and contribute to decrease drag force. However, this positive effect will be eliminated by the separated flow on the rear diffuser plane, and by a pair of lower trailing vortices in the wake region at large rear diffuser angle.

3. Rear diffuser angle is an important factor influencing the wake structure. At large rear diffuser angle, due to the flow separation, the high spanwise pressure gradient on the rear diffuser surface may generate a pair of lower trailing vortices in the wake downstream of the underflow.

Therefore, optimizing the vehicle rear diffuser at a favorable angle can contribute to reduction of the drag force and improve wake structure. It is proposed to increase the rear diffuser angle without flow separation in the diffuser.

Acknowledgments

This research is supported by the Key Laboratory of Manufacture and Test Techniques for Automobile Parts (Chongqing University of Technology), Ministry of Education. The project No. is 2010KLMT09. Part of the work was carried out under the Collaborative Research Project of the Institute of Fluid Science, Tohoku University.

References

1. S. R. Ahmed, G. Ramm, and G. Faltin: Some Salient Features of the Time-averaged Ground Vehicle Wake, SAE Paper 840300, 1984.
2. T. Nouzawa, S. Haruna, K. Hiasa, T. Nakamura, and H. Sato: Analysis of Wake Pattern for Reducing Aerodynamic Drag of Notchback Model, SAE Paper 900318, 1990.
3. H. B. Kwon, D. H. Lee, and J. H. Baek: An Experimental Study of Aerodynamic Drag on High-speed Train, KSME International Journal, 2000, 14(11), p.1267-1275.
4. J. P. Howell, and K. W. Everitt: The Underbody Flow of an EDS-type Advanced Ground Transport Vehicle, Journal of Wind Engineering and Industrial Aerodynamics, 1981, 8(3), p.275-294.
5. P. Mullenbach, and K. R. Deutenbchle: Influence of boundary-layer control-systems on the flow field around passenger cars, Journal of Wind Engineering and Industrial Aerodynamics, 1991, 38(1), p.29-45.
6. Y. Kato , N. Horinouchi, and S. Shinano: Numerical Investigation of Aerodynamics of Vehicle Underbody with Overset Grid System, Technical Journal R&D Review of Toyota CRDL, 1997, 32(2), p.23-33.
(7) K. R. Cooper, and J. Syme: Selecting Automotive Diffusers to Maximize Underbody Downforce, SAE Technical Paper 2000-01-0354, 2000.
(8) A. Cogotti: A Parametric Study on the Ground Effect of a Simplified Car Model, SAE Technical Paper 980031, 1998.
(9) S. Kuroishi, and H. Nkatani: Analysis of Airflow Around Rear Underside of Road Vehicle, Society of Automotive Engineers of Japan, 1989, 22(1), p. 94-98.
(10) A. Laneville: An Explanation of Some Effects of Turbulence on Bluff Bodies, 4th WEBS, 1975, p.333-341.
(11) B. Lokhande, S. Sovani, and B. Khalighi: Transient Simulation of the Flow Field Around a Generic Pickup Truck, SAE Technical Paper 2003-01-1313, reprinted from Vehicle Aerodynamics 2003 (SP - 1786), 2003.