

The Influence of Wheel Spoiler Height on Vehicle Drag Reduction

Q. Jia^{1,2}, M. M. Rashidi^{1,3,*}, SB. Ju^{1,2}, ZG. Yang^{1,2}

¹Shanghai Key Lab of Vehicle Aerodynamics and Vehicle Thermal Management Systems, Tongji University, Shanghai, China

²School of Automotive Studies, Shanghai, China

³ENN-Tongji Clean Energy Institute of Advanced Studies, Shanghai, China

Email address:

mm_rashidi@tongji.edu.cn, mm_rashidi@yahoo.com (M. M. Rashidi)

To cite this article:

Q. Jia, M. M. Rashidi, SB. Ju, ZG. Yang. The Influence of Wheel Spoiler Height on Vehicle Drag Reduction. *Fluid Mechanics*. Vol. 1, No. 2, 2015, pp. 11-17. doi: 10.11648/j.fm.20150102.12

Abstract: Simplified vehicle has been applied in this paper to study the influence of wheel spoiler height on vehicle drag reduction. Two different conditions including stationary and rotating wheels were employed in numerical simulation with steady RANS (Reynolds Averaged Navier-Stokes) equations. Based on detailed analysis of flow around the wheels and integral curve of vehicle drag, it can be concluded that for both the stationary and rotating conditions, the drag decreased first and then increased as the spoiler height gets bigger. For front wheel spoiler, the optimal wheel spoiler height for stationary conditions lower than rotating one due to different drag reduction mechanism, so in wind tunnel test rotating condition was performed.

Keywords: Automobile, Wheel Spoiler, Aerodynamic Drag, Numerical Simulation

1. Introduction

Due to the demand of car bone mission reduction around the world, drag reduction and emission reduction become the top priority in automobile research. Previous studies show, a modern car, the basic exterior shape accounting for 45% of total drag for a modern car, chassis structure and other detail sac counting for 25%, while the wheels cover for 30%, Pfadenhauer et al (1997), so recently the drag reduction of wheels has attracted a lot of attention in automobile drag reduction researches. At present, relevant studies mostly focused on wheels, such as the wheel plate and groove (Shi cun et al 2006, 01, 05, 12; Weipin et al 2011) width of wheels, Jia et al 2014, and wheel arch on the relationship with the relative position of the wheel, other than adding attachment on the chassis to optimize the flow field near the wheel, Qiao et al 2000.

This article will be based on Xing-jun 2012, and others research, simplified saloon cars, two different conditions including stationary and rotating were employed in the research on the influence of wheel spoiler height on vehicle aerodynamic drag, reveal the wheels drag reduction mechanism of the spoiler. For the design of modern wheel spoiler and wind tunnel test we must provide more valuable reference data and programs.

2. Computational Simulation

2.1. The Vehicle Geometry Model and the Computational Domain

In this paper, with the model shown in figure 1 as reference, 1:1 full-size model is established, body length is 4.32 m, width is 1.66 m, and height is 1.33 m. The wheel and brake discs are established according to the actual, simplify the structure of rough surface and small cracks, the wheel is the 5 spokes structure.



Fig. 1. Full scale vehicle model.

To compare the front spoiler height influence on the vehicle aerodynamic drag, the paper on the basis of predecessors work, combined with the actual geometric model, use the front spoiler model shown in figure 2, spoiler width 176 mm, thickness 5 mm.



Fig. 2. Full scale vehicle model installed with front wheel spoiler.

2.2. Meshing and Parameter Settings

This article uses the hybrid grid method effective encryption key parts of the grid. In the neighborhood of the wheel, using surface adapt ability good high-density tetrahedral mesh, in the rest of the region for decreasing the CPU time, we have used hexahedral grid, such to ensure the quality of grid, and can improve the computing speed. The number of grid is selected around 13 million. On the basis of $y^+ = 30 \sim 100$ control the height of center of the first layer grid to the wall.

For the working condition of rotating case, using realizable turbulence $k-\epsilon$ model with two equation model, boundary conditions are set as follows: computational domain entrance is entrance to 30 m/s speed, outlet is pressure outlet. The ground for the mobile boundary, with speed of 30 m/s; symmetry plane is set to the symmetry; the rest domain is no slip wall boundary. Wheel rotation speed is 96.463 rad/s. Static condition, the wheels are set to no slip wall with the ground, the rest of the set is consistent with rotating conditions.

3. Results Analysis

3.1. Rotating Condition

3.1.1. Drag Coefficient Contrast

We have considered constant width and changed height as follow 20, 40, 60, 80, 100 mm. From table 1 can be found, under the rotating condition the vehicle aerodynamic drag coefficient with the front spoiler height a trend of increase with the decrease of the first, get the minimum value in the 60 mm, drag coefficient is about 18 count lower than the standard

model. To facilitate comparative analysis below, tag the standard model and front spoiler height 40 mm, 60 mm, 100 mm model, respectively as a, b, c, and d.

Table 1. Vehicle aerodynamic drag coefficient change with front wheel spoiler height.

The front spoiler height /mm	0	20	40	60	80	100
The vehicle drag coefficient	0.313	0.302	0.297	0.295	0.298	0.300

Partial view, as shown in figure 3, with an increase of height of front spoiler, the front and the rear wheels of the drag coefficients shows the tendency of decline, and the front wheel is faster than the rear wheels, the front spoiler itself shows the tendency of increasing, but the body showed a trend of decrease after the first increase. With increasing the front spoiler height, body drag coefficient decreases and reaches to a minimum value, after that point, body drag will increase.

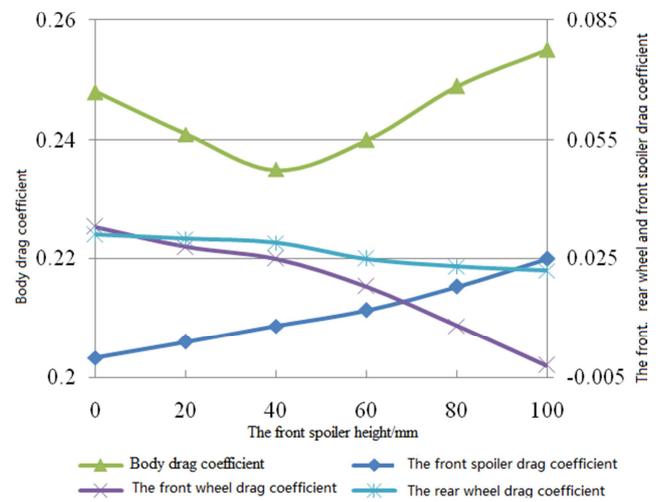


Fig. 3. Curve of different parts' aerodynamic drag coefficient change with front wheel spoiler height.

3.1.2. Front and the Rear Wheel Pressure Contour Contrast

We divided the front and rear wheel as two parts along the X direction, that means a vertical line separated every wheel into two sections. We have integrated the drag forces on every part; table 2 shows the total drag on each part. The table 2 shows that the front wheel drag reducing mainly reflects in front the first half, then half of the resistance has little change. Similarly, the decrease of the rear wheel drag is mainly reflected in the first half. As car flow field around the approximate symmetry, this article will take the left front wheel the first part of the model a and c interpreted into cases.

Table 2. Integrals of different parts of front and rear wheels.

Drag/N	Category	The front wheel first half	The front wheel later half	The front wheel combined
Model a		-13.49	46.40	32.91
Model c		-28.46	45.82	17.36
Drag/N	Category	The rear wheel first half	The rear wheel later half	The rear wheel combined
Model a		-7.86	38.84	30.98
Model c		-12.88	38.30	25.42

Figure 4 shows front spoiler makes flow to separate earlier, and reduce the velocity of air in front of the wheel. From Fig.5, one can conclude that velocity reduces in front of the wheel, and velocity under car body reduces respect to without spoiler case. Another reflect of spoiler is in the back pressure of car, with using spoiler back pressure of car increases. Therefore with using spoiler, total pressure drag will decrease.

Positive pressure area gets down obviously, part of the positive pressure zone transfer to the front spoiler. Similarly, because of the rear wheel primarily in influence of front zone, the impact of the model c air velocity is below the same model a, thus model c positive pressure below the same model of the rear wheel of a.

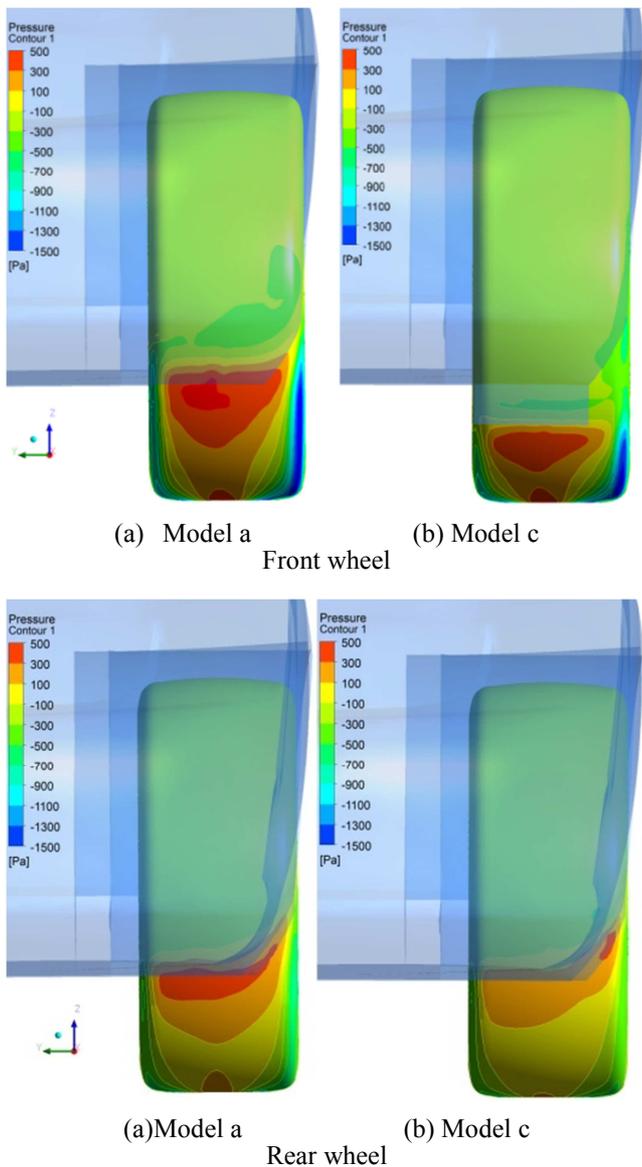


Fig. 4. The pressure contour of front and rear wheel surface in front view.

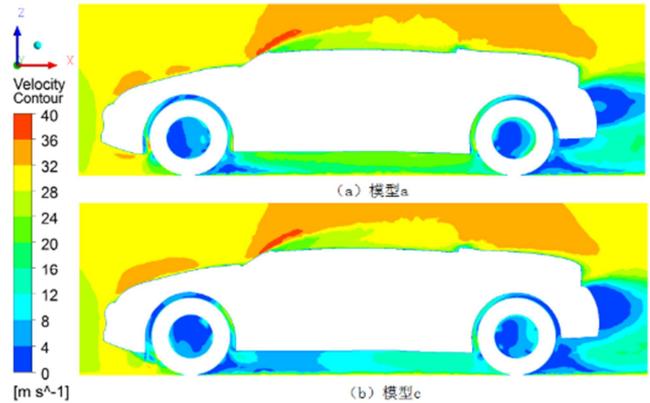


Fig. 5. Velocity contour on the wheel longitudinal symmetry plane.

3.1.3. Body Flow Field

For spoiler height equal to 40mm, we have minimum body drag, for better comparison in this paper we will use the models a, b, d.

Figure 6 is for body drag integral along the length direction (X direction), take a sample point every 0.1 m. It can be found that the influence of the front spoiler to body drag is mainly embodied in the 0~0.518 m (front wheel cavity before), 0.518~1.236 m (front wheel cavity), 3.11~3.82 m (rear wheel cavity), and 4.2~4.35 m (body tail), and other areas.

In 0~0.518 m, with increasing the height of front spoiler the congestion at the bottom of the air is more and more obvious. The front of the front wheel cavity flow speed slow down, pressure build-up, thus the body resistance difference increases with the increase of the height of the front spoiler.

In order to observe the front spoiler height on the influence of front wheel cavity, in the center of the wheel plane of symmetry, build a XOZ plane, in the plane of the figure 7 for the total pressure contours and the velocity y vector diagram. It can be found, after adding front spoiler, the structure of vortex, location and intensity of all great changes have taken place.

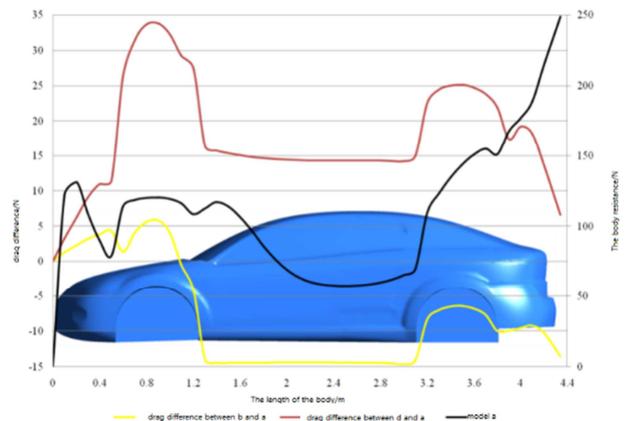


Fig. 6. Integrals of body aerodynamic drag along the length direction.

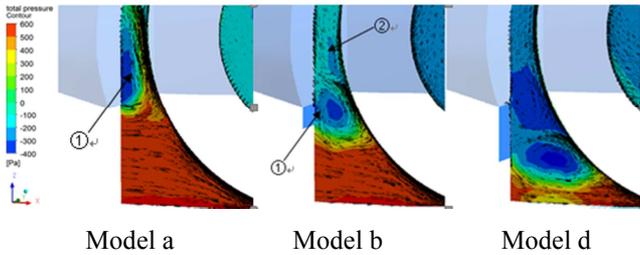


Fig. 7. Total pressure contour and velocity vector diagram of wheel center symmetry plane.

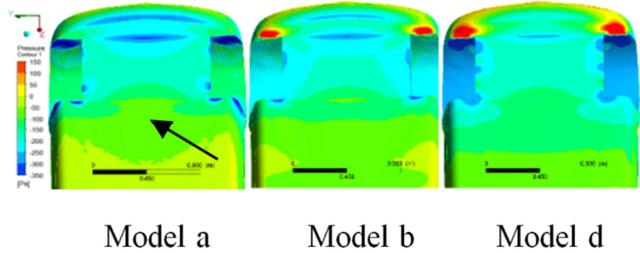


Fig. 8. Pressure contour of front wheel cavity.

After adding front spoiler, separation point position get down, the vortex core position of shear vortex ① also significantly get down. In addition, the shear vortex ① hit the wall on the form reattachment, part of the air flow upward along the wheel chamber for gas viscosity role with the airflow downwash of wheels rotation interaction, form the shear vortex ②. Compared with the model a, b the shear vortex ① separation position is even more, and its reattachment location happened on the front wheel cavity, equivalent to the front wheel cavity added energy, thus the surface pressure relative to the model a have a pick up. Model d relative to b, front spoiler height is higher, larger surface area, more boundary layer vortex into the tail concentrated vortex, cause vortex increase in strength and scale. Vortex is one of the main methods of energy consumption, is also produce drag factor, thus total pressure is lower than the model b. In addition, because the front spoiler height increased, shear vortex reattachment position moved to the front spoiler, it further explains why the model d over the round 1 cavity the surface pressure is lower than the model b.

Because the rear wheels mainly in the wake zone of front wheels, with the increase of height of front spoiler, round into the cavity of the air flow speed gradually decreases, and the kinetic energy, so can't add extra energy to wheel cavity dead zone, further reduce the surface pressure, in model b, d, and the difference of aerodynamic drag with model a in the rear wheel cavity showed a trend of decrease after the first increase.

Figure 9 shows the streamline diagram in the distance from the ground 0.14 m, after adding front spoiler, flow separates on the spoiler. On the one hand, the rear wheel in the wake zone of the spoiler, where flow speed is slow and pressure is higher. On the other hand, because of the flow separation behind of the rear wheel, the pressure behind the rear wheel is low. Sounder the pressure differences more air will flow into the back region of the rear wheel. Thus more air will enter into

the back tail region of the car (as shown in table 3). Thus provides additional kinetic energy to the back tail region, which increases the pressure in the back region of the car. In addition, with flow velocity increasing, the boundary layer become thinner, flow separation delayed, which cause the narrow tail area, so the energy dissipation is decreased here. So the pressure drag of the car is decreased.

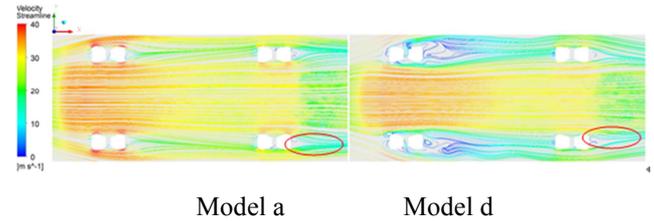


Fig. 9. Near ground surface streamline chart.

Table 3. Mass flow statistics.

Air in flow(kg/s)	Location (m)	Model a	Model b	Model d
x=4.14		7.006	7.276	7.621

3.2. Static Condition

3.2.1. Drag Coefficient Contrast

By table 4 can be found, static conditions the vehicle aerodynamic drag coefficient varies with the front spoiler height a trend of increase with the decrease of the first, the minimum is in the 30 mm. However, compared with the rotating condition, the optimal height decreases. In addition, the static vehicle aerodynamic drag coefficient decreased under the condition of 9 (count), compared with the working condition of rotating (18count), is also reduced.

Table 4. Vehicle aerodynamic drag coefficient change with front wheel spoiler height.

The front spoiler height /mm	0	10	20	30	40	50
The vehicle drag coefficient	0.319	0.316	0.313	0.309	0.314	0.317

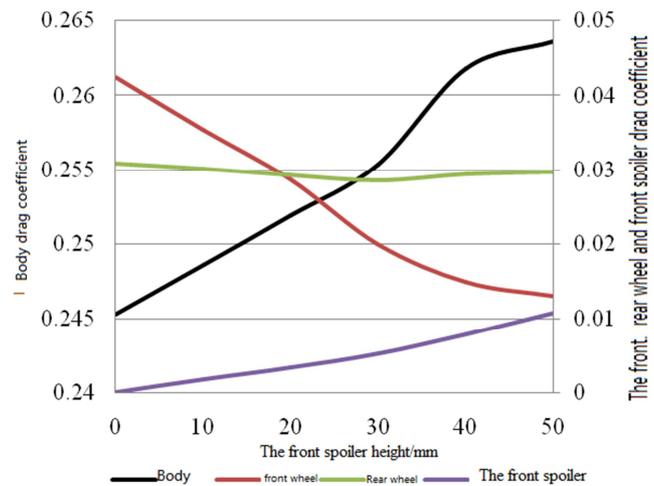


Fig. 10. Curve of different parts' aerodynamic drag coefficient change with front wheel spoiler height.

Partial view, as shown in figure10, with the increase of the

height of the front spoiler, front wheel drag coefficient shows the tendency of decline, the rear aerodynamic drag coefficient changed little, the front spoiler presents the increasing trend, and unlike rotation condition, body showed a trend of increasing the aerodynamic drag coefficient. For the convenience of comparative analysis below, tag the standard model and front spoiler height 30 mm, 50 mm model, respectively as a, b, c. Wheel and the body will be used as two ways to interpret the vehicle aerodynamic resistance changing with the front spoiler.

3.2.2. The Front Wheel Drag Contrast

The front and rear wheel along the X direction symmetry plane cut into two parts, for the two parts respectively drag integral, the drag of the parts as shown in table 2. And different with rotation condition, the front wheel drag reduction is not only reflected in the front part, and also has a certain decline in second half of the drag. The first half drag decline is almost the same cause.

Table 5. Aerodynamic drag of front wheels' front and the back half part.

Drag/N Category	The first half	The second part	The front wheel combined
Model a	-3.904	46.390	42.486
Model c	-24.296	37.373	13.077

Figure 11 is X=1.2 m section on vorticity chart and streamline chart. Trailing vortex can be found both in model a and model c rear wheels. This is because on the touching area of the wheel and ground, the positive pressure area is formed, which make the flow here slide up. Together with the flow rushed from the top of the wheel the trailing vortex is formed.

As a result of the existence of wheel spoiler, model c trailing vortex separation position get down obviously, significantly reducing trail area, the trail was filled with more irregular motion of the vortex, strong movement of the vortices conversion kinetic energy into heat energy, so wake area is low pressure, resulting in a larger pressure drag, This is one of the reasons for a greater drag in model a.

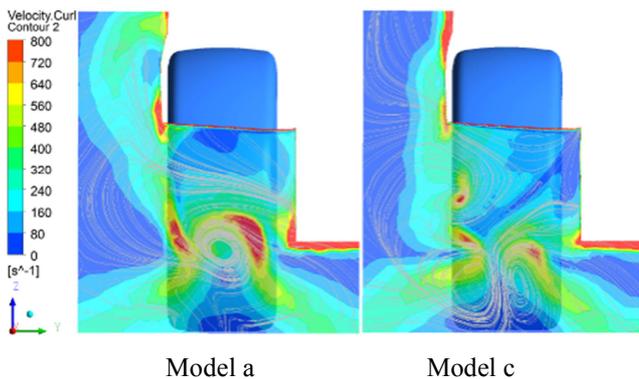


Fig. 11. Vorticity magnitude contour and streamline on plane near the rear end of wheel.

While rotating wheels at the back of the trailing vortex is formed due to the rotating flow in the tail vortex area, Yang *et al* 2014, and the eddy current to flow with lateral also have

close relationship. Due to the rotation of the wheels, flows through the front and on either side velocity increases, make the wheels on both sides appear the low negative pressure, at the same time, it takes more air flow through the wheel side gathered in the dead zone at the end of the wheel, led the pressure at the end more than the static condition, this explains why the spoiler influence static conditions flow pressure big, at the end of the wheel, but small to the rotating conditions.

3.2.3. Body Drag Contrast

It can be found from the figure 12, different front spoiler heights almost have the same effect on the body drag changes of the car. So for convenient, in the following g analysis, only model a and c were chosen as an example to do a comparison. At position of 0~4.1 m, the difference is mainly manifested in the back pressure on the rear body work.

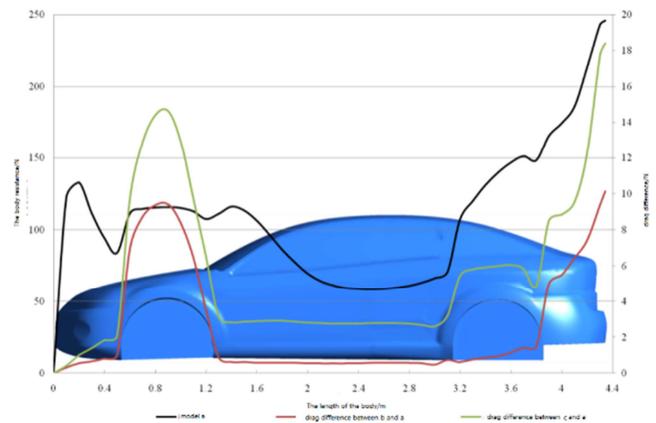


Fig. 12. Integrals of body aerodynamic drag along the length direction.

In the X=4.7 m(0.36 m from the car tail), take a 2 m wide, 1.1 m high plane, figure 13 is the surface vorticity chart and streamline chart. In figure 13 we can found a pair of symmetrical trailing vortex, this trailing vortex is from the bottom of the coil, the air flow from auto top the air flow towards the rear of the car and flow from car side towards the rear of the car flow interaction. Can be found, the car tail the location and structure of trailing vortex is no big changes, This is because the eddy current is only effected by side rear window and trunk angle Faltin *et al.* 1984.

Because of the spoiler's obstruction, part flow which flow pass through the wheel plate into the under body of the car now directly flow to the back region of the car beside of the wheel. More flow entered into the trailing vortex movement, which caused the increasing of the turbulence intensity here (as shown in figure 14). Due to the effect of gas viscosity, have more kinetic energy is converted into heat energy, make the larger negative pressure area that generates the tail, then create a greater drag.

It is worth noting that the lateral air velocity under the condition of rotating also increases with increasing of the height of the front spoiler. However, due to the effect of suction of the rotating wheel front-end makes lateral air flow velocity increases, the separation get less, in addition, the rotation of the wheels will take part of the energy to the

outside air, so the under the working condition of rotating the lateral flow separate later, the size of the eddy current is smaller than static condition, thus the increase of lateral flow did not cause more air separation from the wheel rim, which does not lead to turbulence kinetic energy and the vortex intensity strengthen in the rear body work, Zhi gang *et al.* 2014. On the contrary, the lateral speeding up flow can provides additional kinetic energy to the rear dead zone, same location boundary layer thinning, trailing vortex is delayed, flow separation later, wake in the rear body work area reduction. This explains why the static condition body tail back pressure decreases with increasing the height of the front spoiler, but rotation condition the body tail back pressure increases with the increase of the height of the front spoiler.

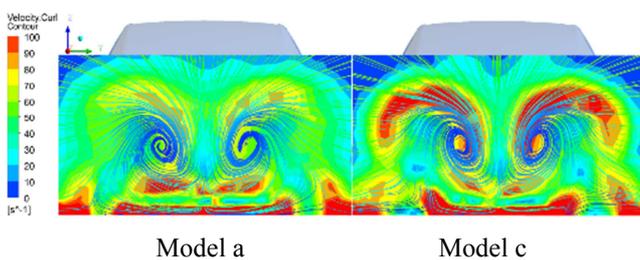


Fig. 13. Vorticity magnitude contour and streamline on plane near the rear end of body.

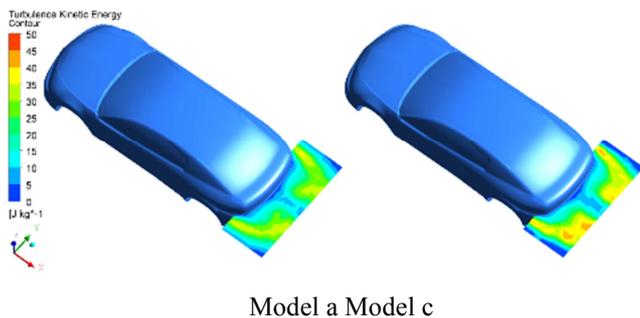


Fig. 14. Turbulent kinetic energy of vehicle rear area.

4. Conclusion

In this article we have used the Realizable $k-\varepsilon$ turbulence model for studying the influence of front spoiler height on vehicle aerodynamic drag under the rotation and stationary conditions of wheels. Conclusions areas follow:

(1) For static conditions, with the increase of the height of the front spoiler, wheel surface positive pressure zone getting down and decreases the separation of the tailed wheel front and rear part drag lower. Lateral air flow velocity increased, leading to more air separation from the rear wheel rim, leading to the tail turbulent kinetic energy increase, the increase of vorticity, produces more energy dissipation, back pressure to reduce gradually, the body drag to rise.

(2) For rotating conditions, after adding front spoiler, flow separates on the spoiler. On the one hand, the rear wheel in the wake zone of the spoiler, where flow speed is slow and pressure is higher. On the other hand, because of the flow

separation behind of the rear wheel, the pressure behind the rear wheel is low. Sounder the pressure differences more air will flow into the back region of the rear wheel. Thus more air will enter into the back tail region of the car (as shown in table 3). Thus provides additional kinetic energy to the back tail region, which increases the pressure in the back region of the car. In addition, with flow velocity y increasing, the boundary layer become thinner, flow separation delayed, which cause the narrow tail area, so the energy dissipation is decreased here. So the pressure drag of the car is decreased.

(3) For rotating and stationary condition, the vehicle aerodynamic drag with front spoiler height increase both decrease of the first then increase, the optimal height is respectively 60 and 30 mm, 18 and 9 count were lower than the original model. The optimum wheel height on stationary is lower than on rotation condition, so test on the wind tunnel should be rotation.

References

- [1] A. Humnic, A. Chiru, On CFD investigations of vehicle aerodynamics with rotating wheels' simulation, SAE, 2006: doi: 10.4271/2006-01-0804.
- [2] A. Cogotti, Aerodynamic characteristics of car wheels, Technological Advances in Vehicle Design, 1983, 13(3): 173.
- [3] I. Dimitriou, S. Klusmann, Aerodynamic forces of exposed and enclosed rotating wheels as an example of the synergy in the development of racing and passenger cars, SAE, 2006: doi:10.4271/2006-01-0805.
- [4] F. Li min, H. Xing-Jun, Z. Shi cun, Numerical simulation of influence of holes in wheel spokes on automotive external flow-field, Journal of agricultural mechanics, 2006, 01:8-11.
- [5] F. Li min, H. Xing jun, Z. Shi cun, Research on numerical simulation for the flow field around automotive wheels with different geometric parameters, Automotive Engineering, 2006,05:451-454-459.
- [6] G. Zheng qi, L. Xiao-hui, L. Weipin, Effect of the shape of wheel spokes on vehicle aerodynamic performance, Science & Technology Review, 2011, 06: 57-61.
- [7] H. Xing jun, F. Li min, Z. Shi cun, Research on aerodynamic characteristics of Automotive Wheels with different wheel spokes, Journal of Tongji University (Natural Science), 2006, 12:1684-1688.
- [8] H. Xing-Jun, L. Fei, G. Peng, Influence of wheel spoiler on aerodynamic drag of passenger car, Jilin Daxue Xuebao, 2012, p32-36.
- [9] S. Krajnovic, S. Sarmast, B. Basara, Numerical investigation of the flow around a simplified wheel in a wheel house, Fluids Engineering, 2011, 133(11):126.
- [10] A. P. Mears, R. G. Dominy, D. B. Sims-Williams, The air flow about an exposed racing wheel SAE, 2002: doi:10.4271/2002-01-3290.
- [11] J. Qing, T. Li, Z. Yang, Influence of rotating wheels on isolated wheels aerodynamics[J].Journal of Tongji University(Natural Science), 2014, 02:287-291.

- [12] J. Qing, T. Li, Z. Yang, Influence of rotating wheels on isolated wheels aerodynamics[J].*Journal of Tongji University(Natural Science)*, 2014,06:949–955.
- [13] S. R. Ahmed, R. Ramm, G. Faltin, Some salient features of the time-averaged ground vehicle wake. *SAE technical Paper Series* 840300, 1984: 481.
- [14] A. F. Skea, P. R. Bullen, J. Qiao, CFD Simulations and experimental measurements of the flow over a rotating wheel in a wheel arch, *SAE*, 2000:2000-01-0487.
- [15] G. Wickern, K. Zwicker, M. Pfadenhauer, Rotating wheels-Their impact on wind tunnel test techniques and on vehicle drag result, *SAE*, 1997: 97-01-33.
- [16] Z. Yang, X. Sha, J. Qing, Influence of wheel width on vehicle aerodynamic drag, *Journal of Tongji University (Natural Science)*, 2014, 11:1682-1686+1732.