Chapter 2
Numerical Simulation of the Influence of Additional Aerodynamic Devices on the Aerodynamic Drag of Van-Body Truck

Dong Wang, Yansong Wang, Yu Han, Yan Dang, Dengyun Fan and Liguang Li

Abstract With CFD software, the 3D flow field of a van-body truck and its aerodynamic drag coefficient are obtained based on numerical simulation. The Cd optimization of the truck is carried out by installing additional aerodynamic devices on the research above. Simulation results show that taper part installed on the tail of van postpones the flow separation, weakens tail vortex, and reduces the Cd. The wind deflector installed on the driver's cab minimizes the incoming flow that impacts the top of the van, especially the part higher than the driver's cab, and the airflow into the gap between driver's cab and van decreases, with lower pressure. As a result, the pressure drag of the driver's cab increases, while that of the van it decreases. And pressure drag of the whole vehicle is reduced with the Cd.

Keywords Van-body truck · CFD · Aerodynamic drag · Additional devices

2.1 Introduction

With the development of highway, and the growing transport demand, the van-body truck has become an important pillar of road transportation. Although the cost of a van-body truck is high, the ownership and operation speed continues to escalate because of its high transport efficiency and convenient handling; at the same time, the fuel economy of van-body truck has received more attention. Under the background of high efficiency and environmental protection in the world, energy conservation and emission reduction for truck have become over increasing stronger [1].

Conventional energy saving and emission reduction techniques include gains in fuel efficiency, optimization of exhaust system, lightweight technique, and so on [2].

D. Wang (✉) · Y. Wang · Y. Han · Y. Dang · D. Fan · L. Li
School of Automotive Studies, Tongji University, 4800 Cao'an Road, Shanghai 201804, China
e-mail: wangdong@tongji.edu.cn

Society of Automotive Engineers of China (SAE-China) (ed.),
Proceedings of SAE-China Congress 2014: Selected Papers,
Lecture Notes in Electrical Engineering 328, DOI 10.1007/978-3-662-45043-7_2

As for private cars, hybrid and new energy technologies are extensively used, and alternative fuel cars have widely deployed. The demand for transporting goods determines the truck with very good power performance, and thus, it starts from optimizing shape and using new materials and continues to reduce the truck's emissions. One important aspect of energy saving and emission reduction is vehicle aerodynamics, and it is one of development directions of truck to save fuel with lowering air drag. The influence from additional aerodynamic devices on van-body truck's aerodynamic drag and the reason for the influence are investigated with numerical simulation in this paper.

2.2 Numerical Simulation

2.2.1 Geometric Model

The object for the study is one domestic van-body truck; the overall dimensions are 7,260 mm × 2,600 mm × 4,000 mm, as shown in Fig. 2.1. The model consists of engine compartment, radiator, gearbox, cab, container and chassis, as well as the features of rearview mirror, air filter, suspension, fuel tank, battery, wheels and wheel cover, and so on, and thus, the model is close to real structure.

2.2.2 Computation Domain and Mesh Generation

To simulate the flow field around the truck, a rectangular boundary model is built. It is twice vehicle length in front of the truck and 7 times vehicle length behind the truck along length in order to make the trailing vortex fully developed, while it is twice vehicle width in both sides and 4 times vehicle height. The blockage ratio of the numerical model is 4 %.

In order to achieve high quality of shell element, the triangular element is used to mesh the complicated parts, and cracks and bolt holes are ignored because of their

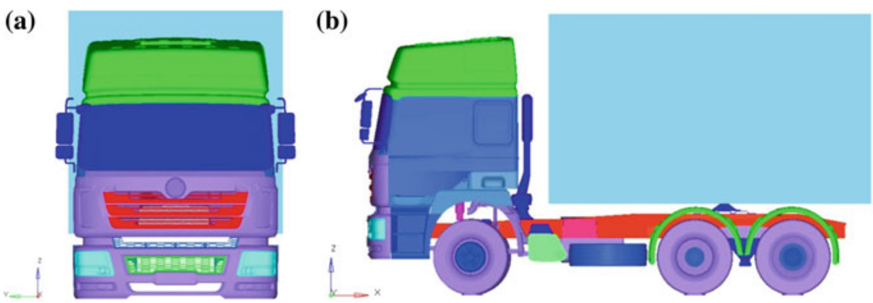

Fig. 2.1 Geometric model. **a** Front view. **b** Side view

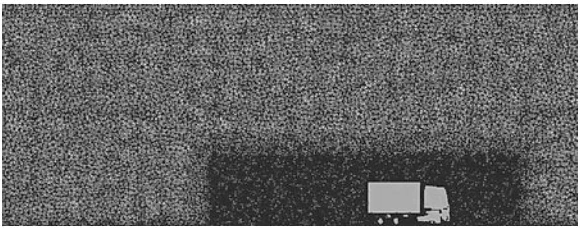

Fig. 2.2 Grid on the section $y = 0$

very weak effect to the truck's aerodynamic characteristics. Tetrahedral meshes are used to simulate the flow field, and the overall elements are 12 million. The flow field around the moving truck which is similar to blunt body with high Reynolds number is high turbulent. In order to get more accurate flow field, the density of the computing domain around the truck is increased as shown in Fig. 2.2.

2.2.3 Boundary Conditions

There are three kinds of flow boundary conditions used in the simulation: external boundary condition, porous media approach, and MRF fan. The moving velocity of the truck is 25 m/s, and a standard atmospheric pressure and normal atmospheric temperature are assumed. The boundary conditions of a numerical wind tunnel are given in Table 2.1.

The condenser, intercooler, and radiator located in the engine compartment cannot be fully penetrated. And it is assumed that their cores are simulated with porous media approach, which the condenser intercooler and radiator are treated as ventilation boundary with resistance coefficient. The pressure drop is simulated in the thickness direction with the porous media model and replaces the complicate structure with stable pressure change. For saving time and resource of the calculation, it is highly recommended to use the MRF method in front-end flow

Table 2.1 External boundary conditions

Surface	Boundary condition
Inlet	Velocity inlet, $u = 25$ m/s, $v = w = 0$
Outlet	Pressure outlet, standard atmospheric pressure
Ground	No slip ground ($u = v = w = 0$)
Slide and top	Stationary wall
Body surface	Stationary wall
Condenser, intercooler, and radiator	Porous media approach
Fan	MRF

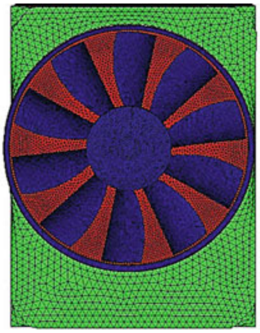

Fig. 2.3 Grid around the fan

simulation [3]. When the MRF model is built, it is divided into motive region and static region as shown in Fig. 2.3. The fan rotation area is the uniform motion region, and the computation domain outside the fan rotation area is the static region.

2.2.4 Solution Parameters

The study object is a van-body truck with complicated shape. Air flows in and on the truck on which there are many components whose curvature changes sharply can cause flow separation. As a result, the turbulence model is employed with RNG k-Ɛ model and non-equilibrium wall function for better adaptability. The convection term of the control equation iterates with second-order upwind scheme after a certain number of iterations of the first-order upwind scheme. The velocity–pressure couple method is the SIMPLE algorithm [4].

2.3 Results and Analysis

2.3.1 Resistance Characteristics Analysis

The coefficient of drag for the truck is 0.5629 where the aerodynamic drag of the container contributes the most and then the cab, the wheel, and other components, as shown in Table 2.2.

2.3.2 Analysis of External Flow Field

Figure 2.4 shows the streamline around the truck on the section $y = 0$, in which the incoming flow arrives at the cab and then is separated into 3 parts. Part one of the flow slips the windshield and flows over the cabin toward the rear part of the vehicle; part two of the flow goes through the grill toward the engine compartment

Table 2.2 Aerodynamics drag of heavy truck

Component	Drag coefficient	Proportion
Cab	0.1522	27.04 %
Container	0.2722	48.36 %
Rearview mirror	0.0177	3.15 %
Engine	0.0126	2.23 %
Driving system	0.0941	16.72 %
Underbody and other attachments	0.0141	2.5 %
Total	0.5629	/

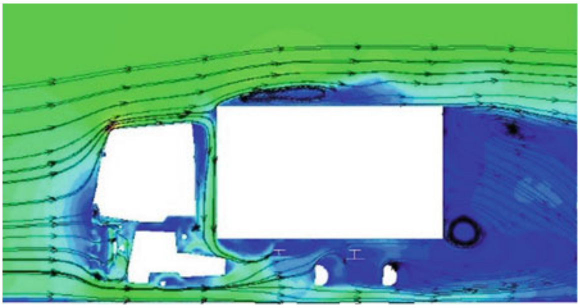

Fig. 2.4 Streamlines in section $y = 0$

which provides the cooling airflow for parts in the engine compartment; part three of the flow goes from the lower of the cab to the rear of the vehicle.

The airflow separates above the leading upper corner of the container since there is a sharp turning. Some of the airstream separates, and vortexes occur in the front of the container and above the container. The airflow above the vortex passes by the vortex toward the end of the container, and airstream twists down on the cross section of the container so that a downwash vortex is generated. Some airstream flows down to the gap and is separated at the corner of the tail edge of the cab, some of which forms vortex, and the rest bypasses the vortex and flows down through the chassis and finally goes out of the vehicle. On the cross section of the tail of the container, the airstream flows up and then forms an upwash vortex. At the bottom of the truck with complicated structure, many small vortexes form which are dragged to the rear of the truck when advancing. Those vortexes merge into large-scaled tailing vortexes x with the downwash vortex and the upwash vortex.

The pressure distribution of the flow field around the truck depends on the airstream's movement when it flows around the vehicle body. When the truck is moving, the incoming airflow impacts the front fascia to form a stagnation point as shown in Fig. 2.5. Therefore, the highest positive pressure is located at the front fascia of the vehicle. On the front side of the container, especially in the region higher than the cab, the airstream is hampered with higher pressure. The airstream flows along the front cab, and the pressure decreases gradually. The region near the grill in the front of the fascia is of the positive pressure. Around the corners of the

Fig. 2.5 The pressure contour of section $y = 0$

windshield and the cab, the velocity of the airstream is higher compared with the narrow region of negative pressure. At the front of the cab's upper wall, the airstream separation causes a negative pressure region.

2.3.3 Analysis of Internal Flow Field of Engine Compartment

As shown in Fig. 2.6, the incoming flow flows into the engine compartment through its bottom and the grill. Most of the airstream goes backward to provide the cooling air for heat-dissipating parts. The airstream partially flows over the cooling module since the air is blocked and directly goes toward the rear of the engine compartment and then flows into the chassis. There are also some airstreams that are directly pumped by the cooling fan into its rotating region from the bottom of the engine compartment, without going through the radiator, which does not give full cooling effect to lower the efficiency of the cooling module and to increase the driving resistance of the truck.

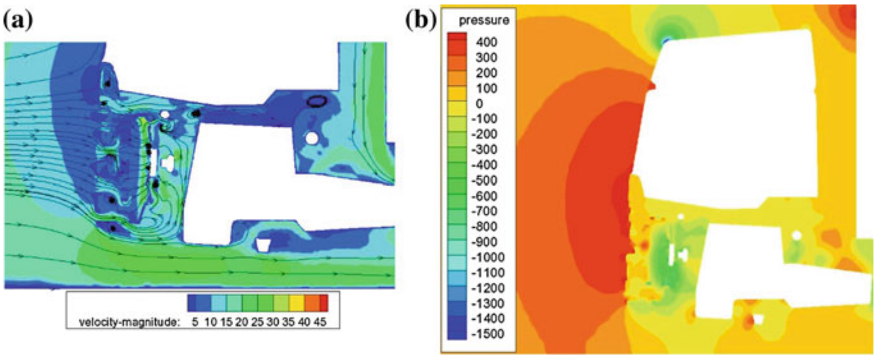

Fig. 2.6 Internal flow field of engine compartment. **a** Stream line on section $y = 0$ in engine compartment. **b** The pressure contour of section $y = 0$ near the cab

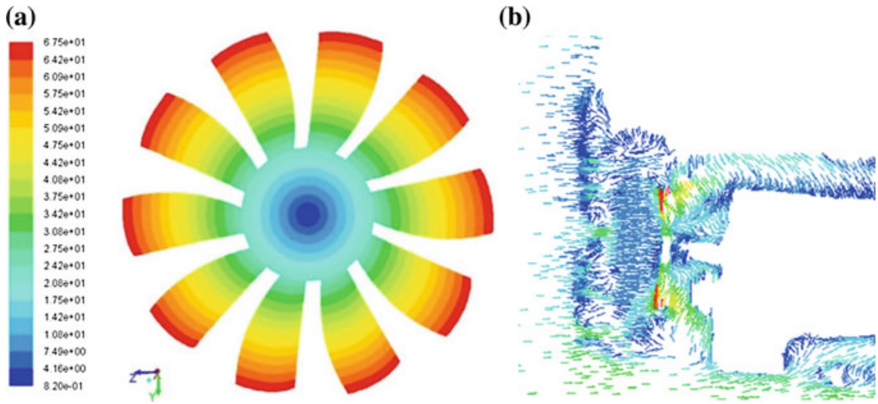

Fig. 2.7 Internal flow field of engine compartment. **a** Velocity contour of fan surface. **b** Velocity vector on section $y = 0$ of the front end of engine

The air flows out of the condenser and goes through the intercooler and radiator in turn. Since the resistance caused by the porous media exists, the velocity of the airstream drops, and then, the airflow through the cooling fan results in the increase in the pressure and velocity. Then, the airstream flows toward the engine and separates over the surface of the engine, and a part of which flows up over the top of the engine to cool the upper surface. However, there are also some dead zones over the engine where vortexes are generated to block the flow of hot air. It does not only have an effect on the cooling efficiency of the radiator, but also increases the aerodynamic drag of the engine. The velocity of airstream is high in the rotation region of the fan, especially on blade edges, as shown in Fig. 2.7. The fan rotates to drive the air to rotate around it. Because of the pumping action of the fan, the kinetic energy of airstream increases, which improves the cooling of the heat exchangers.

The arrangement of components in the engine compartment is compact along with the rotation and pumping action of the fan; the flow field nearby is very complicated, and there are a lot of separation vortexes. Finally, these airstreams flow from engine to the rear of the truck where they meet the flow field near the chassis.

2.4 Optimizing Analysis of Additional Devices

2.4.1 Geometry Model

The objects in this study are as follows: ① base model (Fig. 2.1); ② the base model with tail cone, whose side faces have an angle of 10°, while the top and bottom faces have an angle of 15° (Fig. 2.8a); ③ the base model equipped with fairing (Fig. 2.8b); and ④ the base model with both tail cone and fairing (Fig. 2.8c).

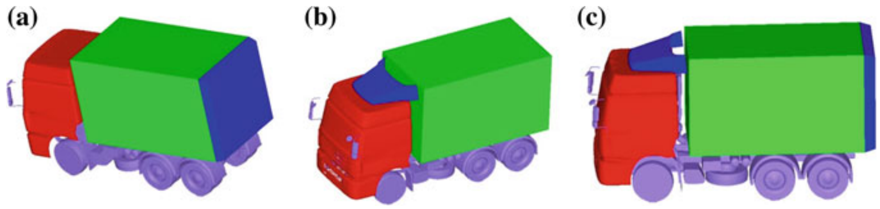

Fig. 2.8 Geometric model. **a** Truck with tail cone. **b** Truck with fairing. **c** Truck with both tail cone and fairing

Table 2.3 The aerodynamic drag coefficient of the truck with different additional devices

Model No.	Additional devices	Drag coefficient	Drop rate
①	None	0.5629	/
②	Tail cone	0.5365	4.69 %
③	Fairing	0.4940	12.24 %
④	Tail cone and fairing	0.4736	15.86 %

These numerical models are accomplished with the same method as the above, and the simulations are carried out with the same parameters, and the results are shown in Table 2.3.

Table 2.3 shows that the installation of a tail cone lowers the aerodynamic drag coefficient by 4.69 %, while that decreases 12.24 % when installing a deflector. Compared with the tail cone, the deflector's drag reduction effect is better. When the truck is equipped with both tail cone and deflector, the drag coefficient would reduce considerably for better fuel economy.

2.4.2 Comparison of Velocity Distributions

The streamlines in section $y = 0$ of truck with different additional devices are shown in Fig. 2.9. The distribution of streamlines around the truck changes significantly.

Comparing Fig. 2.9a with b, it is found that the airstream distributes similarly in front of the truck no matter whether a tail cone is installed or not, but the model with a tail cone results in a smaller tailing vortex, because of the tail cone's guiding performance so that the airstream flows smoothly through the rear of the truck, and then, the airstream breaks away from the tail cone and generates flow separation. The tail cone efficiently puts off the flow separation, and it makes the separation vortex smaller and more far away from the truck. According to hydrodynamics, the pressure drag is intimately linked with the trailing vortex. The smaller the trailing vortex, the less the energy loss by the turbulence, which shows up as bigger static pressure of the tail and smaller differential pressure value between the front and back of the truck.

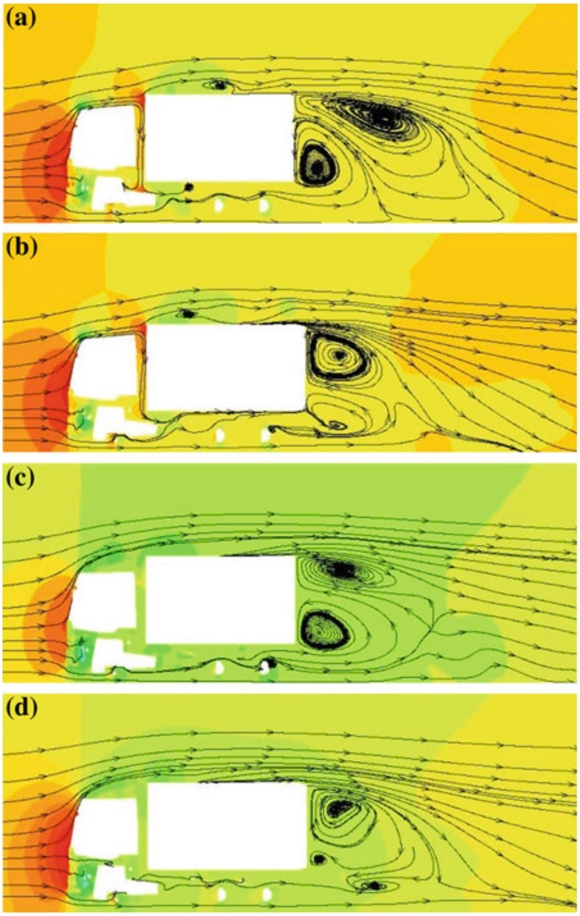

Fig. 2.9 Streamlines on section $y = 0$. **a** Truck without any additional devices. **b** Truck with tail cone. **c** Truck with fairing. **d** Truck with both tail cone and fairing

Comparing Fig. 2.9a with c, it is found that the installation of a deflector on the top of the cab makes airstream flow smoothly through the cab to the container, which considerably decreases the pressure between the cab and the container, so that the positive pressure generated in the front of the container is reduced, and then, the differential pressure between the front and rear of container decreases, and the container's aerodynamic drag coefficient reduces. At the same time, the differential pressure between the front and back of the cab increases as the pressure behind the cab reduced, but its effect is much smaller than that of the container, so the overall pressure drag of cab and container drops considerably to lower the aerodynamic drag coefficient of the truck.

Compared with other models, the streamline distribution in Fig. 2.9d is the best because of smoother streamlines in front of the truck and the smaller pressure difference in the gap between cab and container. At the same time, because of the

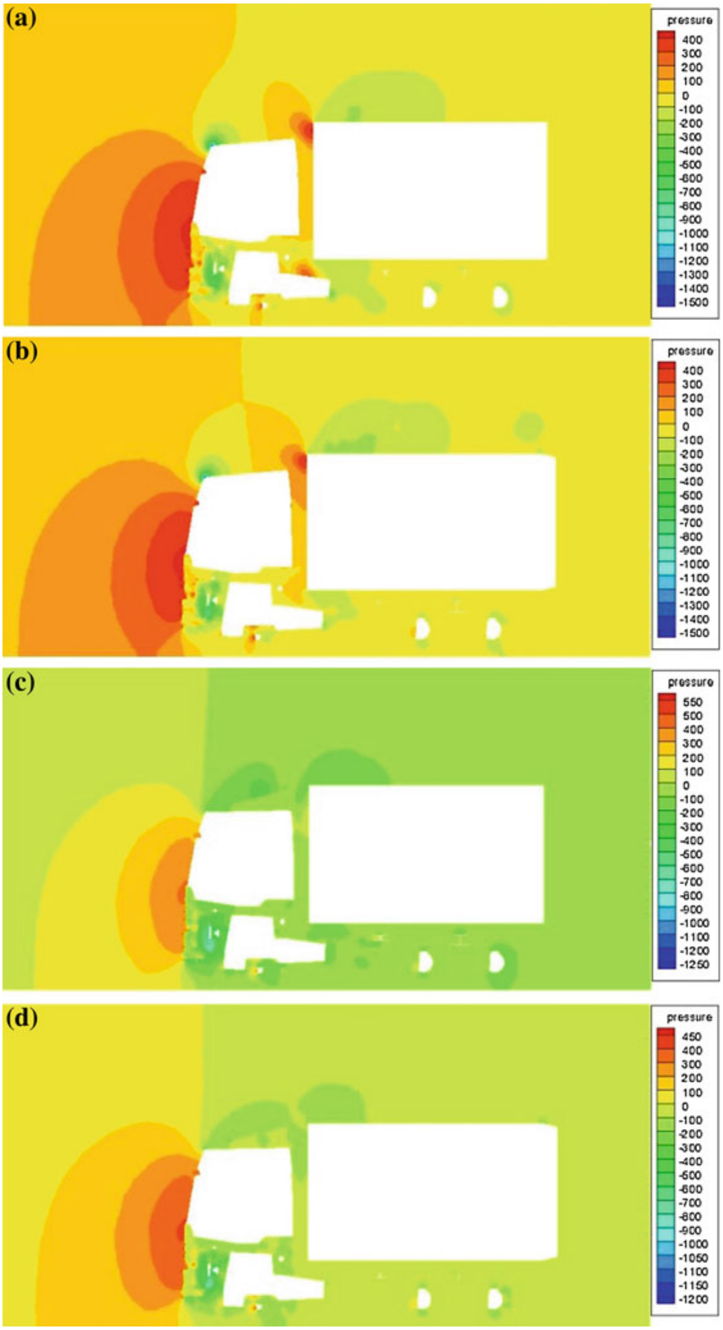

Fig. 2.10 Pressure contour of models with different additional devices on section $y = 0$. **a** Base model. **b** Base model with tail cone. **c** Base model with fairing. **d** Base model with both tail cone and fairing

guiding action of the fairing, airstream separation is put off and tailing vortex region becomes smaller. The energy loss decreases. Therefore, the aerodynamic coefficient of this model is the smallest.

2.4.3 Comparison of Pressure Distribution

Figure 2.10 shows the pressure contour of the model with different additional devices in section y = 0. As seen from Fig. 2.10b, d, there is a small negative pressure region on the top against the tail cone of the model equipped with one because of the airstream separation at the corner of the tail cone. From Fig. 2.10c, d, the fairing equipped on the top of the cab efficiently improves the pressure distribution in the front of the truck. Thanks to the guiding action of fairing, airstreams do not positively impact the part of container taller than the cab and the positive pressure of this part considerably reduces. Besides, since the fairing plays a sealing role in the gap between cab and container, the airstream going into the gap reduces sharply. All factors efficiently reduce the pressure resistance generated on the container.

Since the fairing is hollow, the airstreams behind the fairing flow backward because of the adverse pressure gradient to form vortex in the fairing. It results in loss of energy. As shown in Fig. 2.10c, d, the pressure of the flow field in the fairing is low, and lower than the positive pressure in the front end of the fairing. As a result, there is a pressure resistance applied on the fairing, and it would dissipate some energy. However, this can be ignored compared with the improvement of flow field caused by the fairing. In general, the fairing makes important positive influence on reduction of aerodynamic resistance of the truck.

2.5 Conclusions

With investigation of the aerodynamic additional devices which affect the aerodynamic characteristic of the truck, the aerodynamic coefficients under different circumstances are analyzed, for example, how the aerodynamic coefficient reduces by comparing with base model. And the conclusions come as follows:

1. If a tail cone is equipped in the rear of the truck, the flow separation can be put off and make the tailing vortex smaller for less energy loss. Besides, the pressure resistance applied on the container greatly drops and the aerodynamic coefficient also reduces.
2. Because of the fairing installed on the top of the truck, the airstreams do not impact the part of container taller than the cab positively. Besides, the air which goes into the gap decreases and the pressure lowers. The pressure resistance applied on the cab reduces to a certain extent, while the positive pressure in the

front end of the container becomes much lower. So the overall pressure resistance of the truck reduces as well as the aerodynamic coefficient.

3. If the truck is equipped with both tail cone and fairing, the flow field distribution would be the best. Under this circumstance, the truck's aerodynamic coefficient reduces significantly.

References

1. Zhang H, Zhong S (2009) Energy saving and emission reduction technology of vehicle and it's development tendency. Forum on energy-saving emission reduction and new energy technology, China
2. Ni Y (2010) Lighten the burden on European and American type semi-trailer. Special Purpose Vehicle 10(8), China
3. He W, Ma J, Wang D, Yang Z (2007) Comparing MRF method with sliding mesh method for automotive front end airflow simulation. Comput Aided Eng 16(3), China
4. Patankar SV (1984) Numerical heat transfer and fluid flow. Science Press, Beijing

http://www.springer.com/978-3-662-45042-0

Proceedings of SAE-China Congress 2014: Selected Papers
(Ed.)
2015, X, 512 p. 434 illus., Hardcover
ISBN: 978-3-662-45042-0