A CFD STUDY OF AIR ENTRANCE'S INFLUENCE ON THE AIR RESISTANCE OF AHMED REFERENCE BODY

Yuan Ji, Jiangang Cheng, Jiaqi Xing, Yuqi Huang, Jinxing Chen, Yue Yang*
*Author for correspondence
Power Machinery & Vehicular Engineering Institute,
Zhejiang University,
Hangzhou 310027,
China
E-mail: huangyuqi@zju.edu.cn

ABSTRACT

In traditional design and optimization of vehicle's aerodynamic performance, there are few considering about engine compartment's influence on vehicle's air resistance. In order to find out the engine compartment's influence on the air resistance of the whole vehicle, we conducted some simulations. In this study, Ahmed model was applied to find out how the position and the aspect ratio of the air entrance in the front of the vehicle influence the air resistance of vehicles. And the study was performed only in the premise that the open area and the volume of the engine compartment were constant. The simulations were calculated using the RNG k-ε model with Standard Wall Functions which has already been proved to fit Ahmed model very well by a large number of numerical results. Through simulation, the following conclusions have come to eyes:1, When β which means the ratio of the width and height is in the range of 2 to 4, the drag coefficient continuously decreases. That is, the flatter the entrance is, the smaller the drag coefficient would be. 2, the drag coefficient also decreases with the average diameter of porous media in the case that the cab was filled with porous media. However, what can't be ignored is that small average diameter may cause the shortage of cooling capacity.

NOMENCLATURE

I	[-]	*turbulent intensity*
ε	[-]	*porosity*
Vt	[m^3]	*the total volume of porous media*
Vs	[m^3]	*porous skeleton volume*
D	[mm]	*the average particle size D of the porous media*
α^{-1}	[mm-2]	*viscous resistance coefficient*
C_2	[mm-1]	*inertial resistance coefficient*
V	[m/s]	*velocity of free flow*
β	[-]	*which mean the ratio of width and height*
x	[m]	*Cartesian axis direction*
y	[m]	*Cartesian axis direction*
z	[m]	*Cartesian axis direction*

INTRODUCTION

Many studies have indicated that the vehicle's air resistance is directly proportional to its speed squared once it reaches above 60 km/h and then rises steeply. When heavy trucks are moving at the speed of 80 km/h, there would be a reduction of 12%-13% in the fuel consumption if the air resistance can decrease by 30%. Therefore, reducing air resistance has always been a major goal in vehicle design, and its significance keeps growing for energy saving and emission reduction is becoming the theme of energy policies worldwide. Reducing the fuel consumption by lower the aerodynamic drag is a direct and effective way to conserve energy, which is more significant for high fuel consumption heavy trucks.

In the case of China, its heavy-duty truck sales reached 88,000 in Feb, 2017 with a net growth of 53,000, ring growth of 6% and a year-on-year growth of 152%. This indicates a vast expansion of the market. Basically, there are two kinds of solutions has been put forward by the past studies to reduce the air drag of trucks: passive boundary layer control and active boundary layer control. For passive boundary layer control, it is proposed to use a flexible link with the cab-roof fairing. And it can also install a fence at the gap between the cab and the container. Adding a side plate at the gap between the container and the floor is another effective way. Active boundary layer control is mainly increasing the momentum to delay the separation of the boundary layer, such as the use of motor-drive roller.

The geometry of the Ahmed body is a car-like body conducted by SR Ahmed when he studied the tail vortex in 1984. The external flow field of the model can simulate the basic characteristics of the real vehicle's flow condition except for the rotating wheels, the engine compartment and protrusions on the bottom and surface of the vehicle body. It is a bluff body with separated boundary layers, recirculating flows and complex three-dimensional wake structures [1].

Over the past twenty years there have been many simulations or experimental studies on the aerodynamic performance of the Ahmed body. For example, applying a tail which is coned in top view can effectively reduce the drag coefficient, but the reduction in the drag coefficient is affected by the flow separation thus has a maximum value, for the Ahmed model the drag coefficient can be reduced by 36.85%, and for the actual vehicle model with a rear cone angle of 5 degrees, it can only be reduced by 11% [2]. Using active fluidic oscillators to control the wake stream is also an effective way to restrict flow separation, which can lower the Cd value of Ahmed body by 70 counts and is hardly influenced by actual road conditions [3].

The drag coefficient can also be reduced by using synthetic jet array which can optimize tail air flow. In terms of this synthetic jet array method, with the 25° Ahmed model, the reduction in the drag coefficient is proportional to the momentum coefficient and the drive frequency. However, with an angle of 35°, the reduction effect does not follow the same law. This difference between the two cases was considered due to the generation of the oblique vortex caused by the flow separation at the 30° rear angle, and the influence of the jet momentum is more obvious compared with the direction of the jet and the driving frequency. The more the jet air flow approaches free flow, the more the drag coefficient decreases [4]. Some researchers use underbody diffuser, which means cutting off a part of the model obliquely on the rear bottom on the Ahmed body, when the cutting angle is smaller than 7° there is a predictable drop in air resistance; when the cutting angle is larger than 7° the flow separation come out and affects the tail airflow [5]. In the year of 2006, RMIT University designed a wind tunnel experiments, they used four different ways to cover the air intake grille of a Ford Falcon AU. They blocked the intake grille to change the value of its area and studied how this would influence the air resistance. They studied the effect of air entrance grille on the whole vehicle's air resistance and obtained the conclusion that horizontal cover is a more effective way to reduce to air resistance [6]. Based on this, it was believed that the aerodynamic performance of air entrance could also be discussed on the shape which is approximately rectangular and symmetrical. However, what was not mentioned is that the influence on internal cooling performance through this covering method. In the same year, Ford company compared the difference of air flow between opening and sealing the intake through CFD simulation. They studied what caused the internal flow resistance and made detailed research on the influence of front intake opening parameters including the parameters of the two grilles, the parameters of the air flow tube, etc., and summed up the standard to reduce the air pressure loss on front grille [7]. In 2011, Chrysler Corporation, of the United States studied the effect of variable air grating opening and closing on engine heat management performance. The advantages of variable pneumatic grids are demonstrated by experiments.

However, in previous designs, the design of the front cabin always following the idea of putting the requirements of the cooling system on first concern and the air flow is not well optimized. As is mentioned above, the air drag reduction design of the external flow field in the automobile is mature today, the engine compartment flow resistance optimization to reduce the vehicle air resistance has become an important way to reduce fuel consumption. Therefore, in order to find out the influence of the flow around the engine compartment, a series of simulations were carried out. In this study, Ahmed model was applied to study how the position and the β of the air entrance in front of the vehicle influent the air resistance of vehicles. And the study was performed only in the premise that the open area and the volume of the engine compartment was constant. The simulations were calculated using the RNG k-ε model with Standard Wall Functions which was already proved to fit Ahmed model very well by a large number of simulation results. The relative optimum arrangement of the air entrance position is obtained through the analysis.

NUMERICAL METHOD

The RNG k-ε model was applied to this numerical simulation, because the application of it is more mature, high computational efficiency. It is a method which has been proved to be accurately accord with the actual flow of Ahmed reference model very well by a large number of simulations and experiments. Standard wall function with second order up-wind scheme was also attached to this simulation.

validation of numerical models

The validation of this study was conducted based on this reference model. In order to improve the computational efficiency, the simulations were carried out using scaled down Ahmed models with the same various rear angles of 0°，12.5°，25°，27°，27.5°.

The Ahmed model was scaled to 300mm in length, 110mm in width, 80mm in height and with a front round radius of 30mm. A model with 30° rear angle is generated and shown as below.

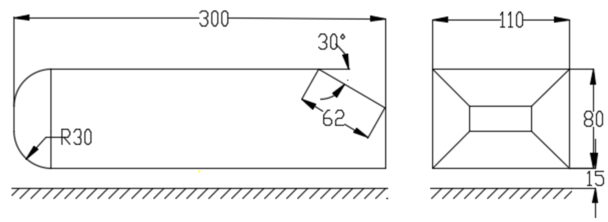


Figure 1 The scaled down model

The distance from the entrance of computational domain is twice as the length of Ahmed model, and the distance from the model's rear surface to domain exit is 6 times to the model length. The width and height of the domain are 9 times and 7 times to the model respectively.

5.21 million hexahedral meshes are generated in the computational domain, the minimum size in which is 0.1mm and maximum size is 0.5mm.

The boundary conditions in simulation are setting as below:

1. Entrance speed is 12m/s
2. The exit is set as outflow condition
3. Ahmed model surface is set as non-slip wall
4. Ground of the computational domain is set as moving wall, where the moving velocity is as same as entrance speed

In ref [8], a series of experiments was conducted in the small wind tunnel at Tokyo City University (TCU) by tsuhei Kohri, Teppei Yamanashi, and Takayoshi Nasu as shown in Figure 2, the TCU wind tunnel were conducted on the scaled down Ahmed models which are the same as what were just mentioned above and obtained the Drag coefficients of different rear angles . The boundary conditions in the verification calculation are consistent with this experiments, and ANSYS 17.1 was applied to complete the simulation for different rear angles. The simulation result and the experiments conclusion are compared and the comparison results are as follows.

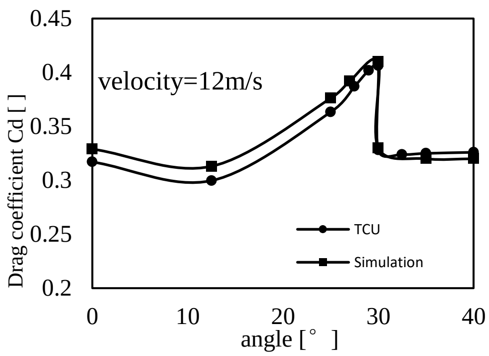
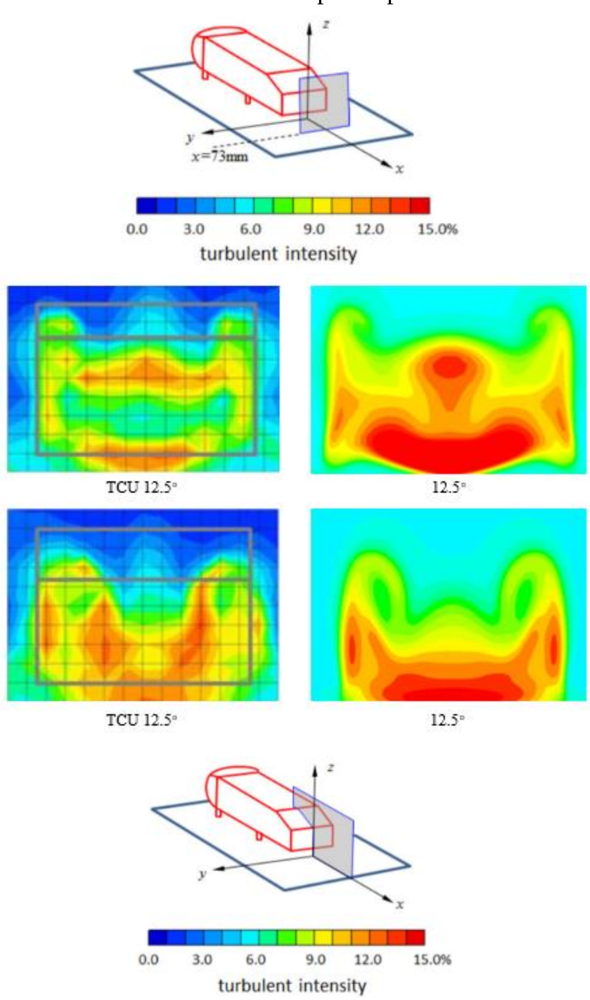


Figure 2 Cd comparison between TCU wind tunnel and simulation

The drag coefficient-angle curve in Figure 2 indicates that the simulation value is higher with angle from 0° to 30° and lower with angle over 30° compared with the TCU wind tunnel value, but the maximum deviation is approximately 3.7%, which is supposed to be acceptable.

According to the experiment's conclusion in the compared article, the same colour standard as the experimental results was selected. The turbulence cloud maps are performed as follows:



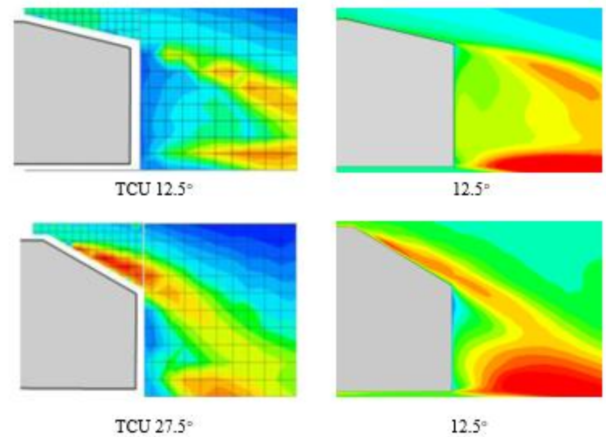


Figure 3 comparison in turbulent intensity between TCU wind tunnel and simulation

Through the comparison, although the colour of the simulating results is sort of different, the variation of the turbulence intensity is consistent with the experiment's. The tail turbulence still shows a high similarity to the actual situation obtained in the experiment. And although there are some distinctions in the details and the manifestations are kind of different, the overall turbulence intensity distribution exhibits consistent results. The above differences can be explained that different results were obtained through different researching methods, one is experiment, and the other is simulation. For one thing, the simulation process can't completely simulate the experiment's situation, for another, there was also deviation during the process of the experiments. Thus, the accuracy of the simulation method is acceptable.

DESIGN OF MODELS AND SIMULATION:

In order to find out the influence of the air entrance arrangement on the aerodynamic performance of the Ahmed model, a modified Ahmed model was designed as what was shown in figure 4, the dimensions of it are shown in the same figure. Meanwhile in the premise that the intake is bilateral symmetry, we designed three opening positions, that is, upper, middle and lower in vertical direction. Each position is located by the centre of the rectangular opening area. Additionally, we designed 5 rectangles in each position with β varying from 2 to 4. Thus, 15 models in all were established, and their precise positions are shown in Figure 4.

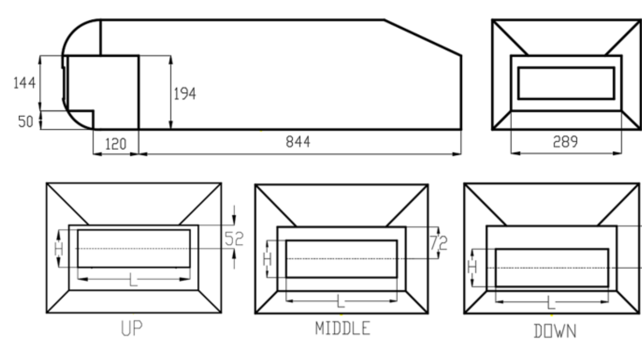


Figure 4 dimensions of models

In these simulations, the velocity of the inlet is set as 25m/s, therefore the Reynolds number is around 73100. The k-Ɛ turbulence model is adopted in the study. The turbulence intensity was calculated by the equation I=0.16*(Re)^(-1/8), and the result is 7.79%.

TREND AND RESULTS

The influence of positions

It can be seen clearly from Figure 5 that in each vertical position, the drag declines as the β grows. That indicates that for any vertical position with the β from 2 to 4, in premise of the same intake area, a flatter air entrance will result in less air resistance. This can be explained by comparing the simulation results of the down_4 and down_2 models: Figure 6 shows the velocity distribution at a distance of 0.5m from the front in x direction, it's easy to find out that an air entrance which is more flat in shape has the effect of increasing the speed of air flow at the bottom of the model. And the x-y surface streamline in Figure 6 was also analyzed. There is a large amount of the air separating from the body to both sides of the model with the β of 2 while air tends to flow through the bottom of the car to the tail rather than separate towards sides with the β of 4. This brings a result that the center of the tail vortex can be kept a farther distance from the vertical back wall. And the pressure on the back wall increases as it shows in Figure 6 and Figure 7.

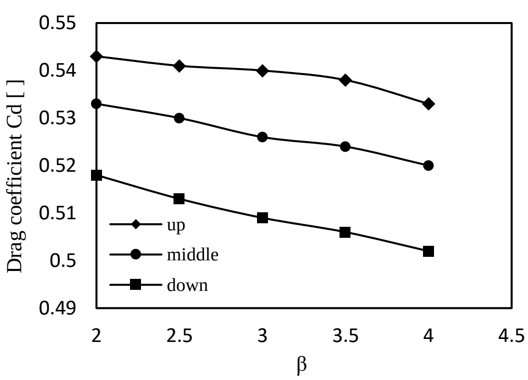


Figure 5 Drag coefficient with different β in up, middle, and down position

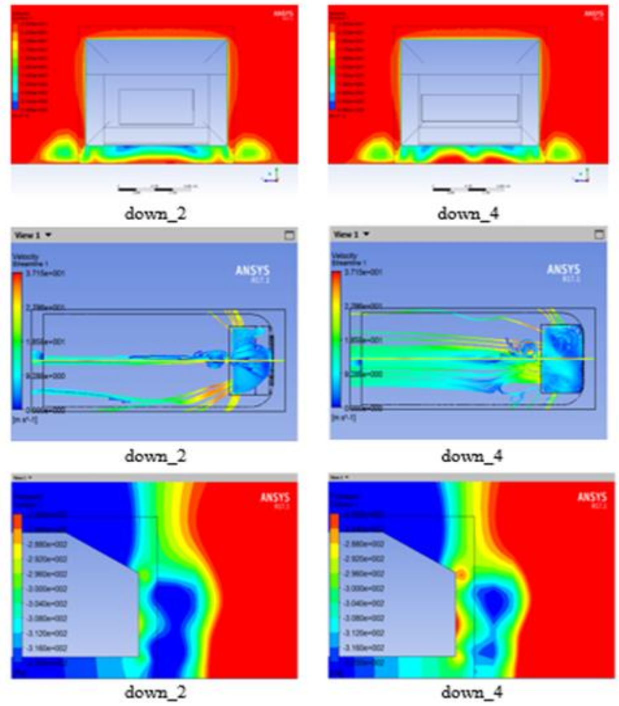


Figure 6 flow conditions around the down_2 and down_4 models

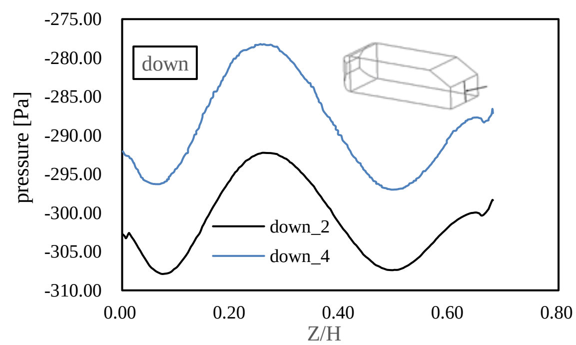


Figure 7 pressure in the vertical back

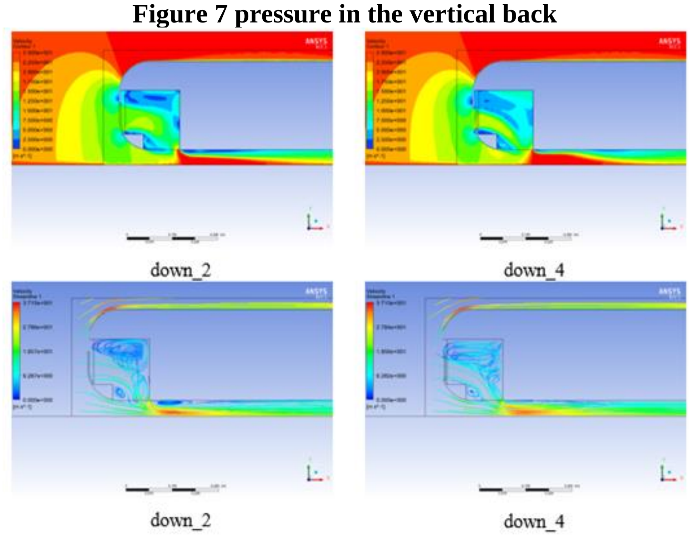


Figure 8 flow conditions in the engine compartment with down_2 and down_4 models

Besides, another regular rule was also easily to be find out that as the vertical position comes lower, the drag coefficient reduces visibly. Because in a lower opening height, the baffle phenomenon is weaker and the flow is more smooth which benefit the reducing of air resistance.

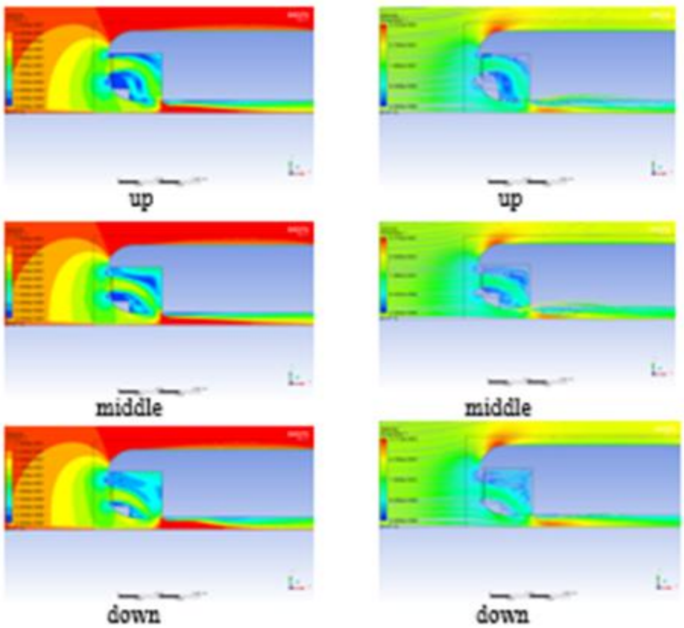


Figure 9 flow conditions in different vertical positions

The influence of porous media

The pores that are interconnected in the porous medium can be regarded as effective pores, and the fluid in the effective pore is free-flowing. In contrast, the pores that are partially blocked and can't be circulated are called dead ends, and the fluid flow in them is relatively stagnant. Porosity can be regard as an indicator of porosity. Presuming that the pores are uniform, the porosity can be simply defined as the ratio of the total volume of the pores in the porous dielectric material to the bulk volume, where the porous skeleton volume in the porous medium is recorded as Vs, then

$$\varepsilon = \frac{V_t - V_s}{V_t} \qquad (1)$$

Where ε is called porosity, its value is related to the structure, shape and arrangement of solid particles of porous media. In this study heat transfer was not taken into account, thus we use porous media to simulate the situation inside the engine compartment, and according to real condition, porosity value was set to 0.4.

In order to simulate the size of different objects inside the engine compartment, the average particle size D of the porous medium is introduced, and the degree of loosening of the internal structure is expressed by setting different particle diameter values. According to Ergun equation, the pressure drop calculation formula of porous media, in turbulent flows, packed beds are modelled by using both a permeability and an inertial loss coefficient. One technique for deriving the appropriate constants involves the use of the Ergun equation, a semi-empirical correlation applicable over a wide range of Reynolds numbers and for many types of packing:

$$\frac{|\Delta p|}{L} = \frac{150\mu}{D_p^2}\frac{(1-\varepsilon)^2}{\varepsilon^3}v_\infty + \frac{1.75\rho}{D_p}\frac{(1-\varepsilon)}{\varepsilon^3}{v_\infty}^2 \qquad (2)$$

When modelling laminar flow through a packed bed, the second term in the above equation may be dropped, resulting in the Blake-Kozeny equation

$$\frac{|\Delta p|}{L} = \frac{150\mu}{D_p^2}\frac{(1-\varepsilon)^2}{\varepsilon^3}v_\infty \qquad (3)$$

Thereby $\alpha = \frac{D_p^2}{150}\frac{\varepsilon^3}{(1-\varepsilon)^2}$, $C_2 = \frac{3.5}{D_p}\frac{1-\varepsilon}{\varepsilon^3}$, the viscous drag coefficient and the inertia Drag coefficient of the porous medium are calculated under different average particle diameters as follows:

D [mm]	ln(D) [mm]	α $[m^{-2}]$	C_2 $[m^{-1}]$
5	0.69897	33750000	6562.5
25	1.39794	13499999.53	1312.5
50	1.69897	337500	656.25
250	2.39794	134999.9953	131.25
500	2.69897	3375	65.625

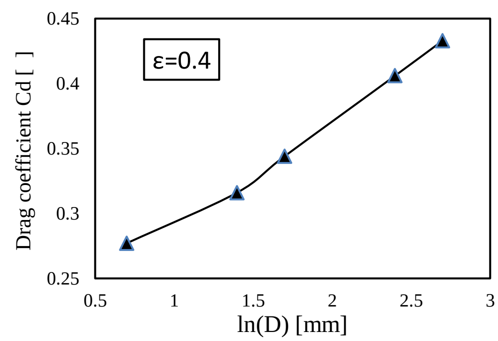


Figure 10 Drag coefficient with different particle diameters

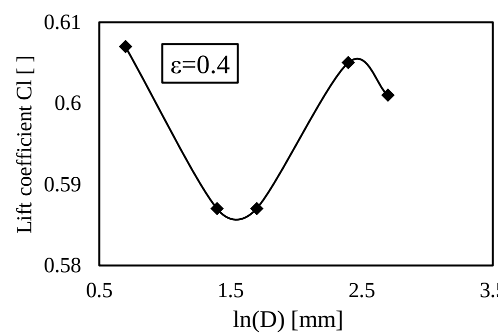


Figure 11 Lift coefficient with different particle diameters

The simulation results in Figure 10 and Figure 11 show clearly that with the decrease of the average diameter of the particles in the porous media, the drag coefficient become evidently smaller and gradually tend to the value of the actual Ahmed model without opening. If only taking reducing air resistance into consideration, the denser the layout is, the smaller the drag coefficient will be. However, it is easy to come up with the question that whether the detailed layout will bring the problem that the cooling system's requirements cannot be satisfied. Figure 12 shows vertical velocity distribution at the centreline of the air exit of engine compartment under different conditions of particle diameters. We can see from the figure that as the diameter decreases the average flow speed at the outlet falls, which means a decrease in the flow capacity of cooling air and this may cause the problem of inefficient cooling. Meanwhile, the corresponding lift coefficient will also rise to some extent. Considering the drag coefficient, the lift coefficient and the cooling effect, 50mm particle size arrangement is most suitable for the particle size set in this study. For actual vehicle engine compartment model, it may get best air resistance performance and cooling performance when its components and parts are of the equivalent size.

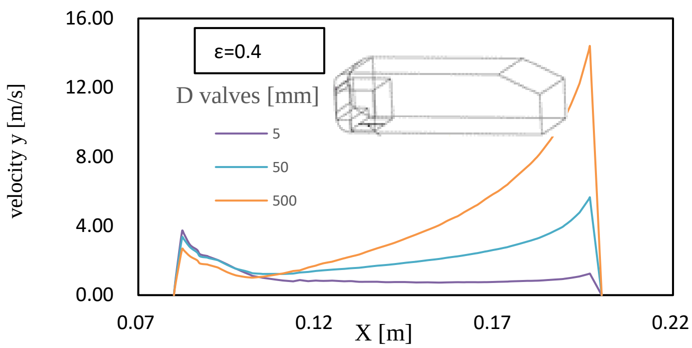


Figure 12 velocity in the central line of the air outlet

CONCLUSION:

Based on Ahmed body, this study discussed the influence of air entrance form of the engine compartment on the air resistance of the whole model. The following conclusions are drawn through numerical simulation method:

1. In the premise that the centre height of the rectangular entrance remains constant, an entrance with large β which means a flatter shape can reduce the air resistance because it helps to generate a weaker baffling effect and more air flows through the bottom of the vehicle to the tail in this situation. All these effects helps postpone the generation of the vortex.

2. With the same β, the rectangular entrance in the low position can form the airflow to be smoother, thus helps obtain a small drag coefficient.

3. Using porous media to simulate the objects inside the engine compartment in the simulations. It was found that small particle size can bring out small coefficient of resistance, which will reduce the vehicle's fuel consumption. However, what cannot be ignored is that small particle size will lead to slow airflow inside the engine compartment. This may have a great passive influence on the cooling effect of the conventional diesel trucks, and it is more suitable for the arrangement of electric vehicles which are less demanding in cooling system performance.

Through all the numerical simulation works, a rough analysis about the influence of the front air entrance arrangement on the drag coefficient of the Ahmed model was conducted. But this simulation results still need to be further verified by conducting experiments in the future. Due to the limitation of time and condition, simulation examples are still sort of undetailed and require to be refined. And this air resistance study inside the engine compartment is not well combined with the operation of cooling system. It was expected that there will be more work on the aerodynamic study and automobile design.

REFERENCE

[1] Krastev, V.K; Bella, G., On the Steady and Unsteady Turbulence Modeling in Ground Vehicle Aerodynamic Design and Optimization. *SAE Technical Papers*, September 2011.

[2] Daryakenari, Behrang; Abdullah, Shahrir; Zulkifli, Rozli ., Reducing Vehicle Drag Force Through a Tapered Rear Side Wall. *SAE International Journal of Commercial Vehicles.* Vol. 6,October 2013,pp. 582 – 588.

[3] Metka, Matthew; Gregory, James; Sassoon, Aaron; McKillen, James ., Scaling Considerations for Fluidic Oscillator Flow Control on the Square-back Ahmed Vehicle Model. *SAE International Journal of Passenger Cars - Mechanical Systems.* Vol. 1,April 2015,pp. 328 - 337

[4] Park, Hoonil; Cho, Jun-Ho; Lee, Joonho; Lee, Dong-Ho; Kim, Kyu-Hong ., Aerodynamic Drag Reduction of Ahmed Model Using Synthetic Jet Array, *SAE International Journal of Passenger Cars - Mechanical Systems,* Vol. 6,2013,pp.1-6.

[5] Huminic, Angel; Huminic, Gabriela., Computational Study of Flow in the Underbody Diffuser for a Simplified Car Model. *SAE Technical Papers*, 2010,

[6] Jama, Hussein; Watkins, Simon; Dixon, Chris ., Reduced Drag and Adequate Cooling for Passenger Vehicles Using Variable Area Front Air Intakes, *SAE Technical Papers,* 2006.

[7] Ding, Wei; Williams, Jack; Karanth, Dinakara; Sovani, Sandeep .,CFD Application in Automotive Front-End Design, *SAE Technical Papers*, 2006

[8] Kohri, Itsuhei; Yamanashi, Teppei; Nasu, Takayoshi; Hashizume, Yoshimitsu; Katoh, Daichi ., Study on the Transient Behaviour of the Vortex Structure behind Ahmed Body . *SAE International Journal of Passenger Cars - Mechanical Systems* , Vol,7,2014,pp. 586 - 602

[9] Chowdhury, Harun; Moria, Hazim; Ali, Abdulkadir; Khan, Iftekhar; Alam, Firoz; Watkins, Simon., A study on aerodynamic drag of a semi-trailer truck. *Procedia Engineering,*Vol.56,2013 ,pp.201-205.

[10] Rouméas, Mathieu; Gilliéron, Patrick; Kourta, Azeddine., Analysis and control of the near-wake flow over a square-back geometry, *Web of Science*, Vol,38, 2009, pp.60-70.