Optimization of Geometry of a New Device to Reduce Aerodynamic Drag on a Heavy Vehicle

Zafar Namazian
Department of Mechanical Engineering, College of Engineering, Yasooj branch, Islamic azad university, Yasooj, Iran, z.namazian@iauyasooj.ac.ir

***Abstract*— This study examines a new device aimed at reducing drag on commercial vehicles. The computational fluid dynamics (CFD) technique is utilized to the analysis of air flow around this device and also to optimize the geometry of device that has important effect on drag reduction. At first, simplified dimensions of a commercial vehicle are modeled as demonstrated in literature and the coefficient of the drag is measured through the analysis of the fluid flow. After choosing various different geometries with the addition of the new device and picking out the best geometry, the coefficient of the drag in the new position is measured and a comparison is made with the base model. The results indicate that there is about 22% drag reduction in the new state in comparison to the base model.**

***Index Term*— Aerodynamic -Heavy Vehicle-Drag Reduction Devices, Optimization.**

I. INTRODUCTION

Among the key areas of the automotive industry and in particular, the commercial vehicles sector is aerodynamics and its impact in lowering the force of air resistance, the consumption of fuel, the control of vehicle, and noise reduction. Besides using suitable aerodynamic designs on commercial vehicles to gain the abovementioned goals, including certain drag reducing equipments such as flaps, deflectors, and boat tails has also been found to have a huge impact on the total reduction of drag and consumption of fuel in this type of vehicles. Trail mobile funded the University of Maryland [1, 2, 3] in the 1950s to enhance trucks' consumption of fuel by examining the aerodynamics of trailers and tractors.

Past literature reveals multiple attempts undertaken in the study of base drag reduction in axisymmetric and two-dimensional bodies. Research by [4] assisted in defining the efficacy of base and ventilated cavities, multi-step after bodies, locked vortex after bodies, as well as after bodies that engage a non-axisymmetric boat-tail approach in net and base reduction of drag at various speed levels. These studies highlighted the wide attributes of the flow as well as the potential mechanism of fluid dynamics related to the device resulting in the reduction of the base drag.

The aerodynamic of commercial vehicles is affected by certain geometrical factors [5]. Measurements have been carried out in a wind tunnel at a low Mach to identify the impacts of different after bodies with boat-tails on the drag of a square section of a long body at a zero incidence. Having boat-tails at only two opposing sides resulted in a small drag reduction while in certain experiments, the drag increase was very high. Having boat-tails on four sides resulted in a higher drag reduction even if the boat-tail was short as in the case of an axisymmetric body.

This showed that beneficial drag reduction could achieved for trucks, containers and buses using short fairings at the rear ends [5]. [6] examined the potential for saving fuel due to drag reduced devices fitted on heavy vehicles. The examinations included testing realistic on-the-road operations using simulated routine driving routes for urban and long haul distributions while also considering the differences in the weight of the vehicles. Studies also numerically examined the utilization of lateral guide vanes as a means of drag reduction devices on vehicles [7].

Two types of ground vehicles were examined utilizing the dynamics of computational fluid namely a model of a simplified bus and a model of a simplified sports utility vehicle (SUV). Pressure recovery was improved by utilizing guide vanes to maneuver direct air into the low-pressured wake sections, which would in turn lower the drag form and thus lower the overall drag from the aerodynamic application. [8] developed three simple and low-cost devices using aerodynamic drag reduction to be applied on the trailer of a tractor-trailer truck. These three devices have been heavily tested operationally amassing more than 85,000 miles. The research shows a joint savings of fuel of up to 10% using an average speed of 47.5 mile per hour. This fuel economy enhancement is related to drag reduction to the equivalent of about 30% with a 0.45 drag coefficient.

Studies by [9] included theories and experiments on the impact of aerodynamic shape changes on a model scale of a tractor-trailer based on the aerodynamic drag and by enhancing the features of the aerodynamics using additional modifications to the body of the trailer. This experiment was carried out by utilizing Reynolds No. 106 wind tunnel and a model truck scale with the length of 1/30th of the Mercedes - Benz 1844 ACTROS Container-Trailer. Modifications included the addition of a base flap to the rear and a base flap with a splitter at four inclined angles (0˚, 10˚, 20˚, and 30˚). The results of the experiment revealed a high reduction of the aerodynamic drag by about 18% at inclination angles of 10˚. The addition of the splitter resulted in an improvement of the reduction in the aerodynamic based on the base flap behavior at β=20˚ with 21% reduction in drag. [10] developed four simple and low-cost devices using aerodynamic drag reduction to be applied on the trailer of a tractor-trailer truck. Two devices with vortex flow were heavily tested operationally amassing more than 85,000 miles.

The technology with the two vortex flow shows a joint savings of fuel of up to 8% using an average speed of 47.5 mile per hour. The trailers' base drag was reduced by using the developed devices with two base mounts. These devices were designed based on computational designs and testing in the wind tunnel. Both these two devices that were base mounted demonstrated a reduction in the drag of heavy vehicles by an average of 8% amounting to 4% of savings in fuel using speeds applicable on the highways. The joint savings in fuel of both the base devices and the vortex flow was more than 12% in total. More than two-thirds of the consumption of fuel by heavy vehicles at acceptable highway speeds was caused by the aerodynamic drag.

The efficiency of aerodynamics in the aft regions of heavy vehicles was normally sacrifices due to the functionality concerns. This results in a high separated flow on the lee side of the trailer trucks, as well as a related drag penalty. [11] carried out a thorough and extensive research strategy which integrated the development of the actuator, the dynamics of computational fluid, along with bench-tops and wind tunnel testing. There has been an increased emphasis in the research on improving aerodynamic efficacy of commercial vehicles as it reduces both the overall consumption of fuel and emissions. The benefits provided by a novel device on fuel savings for this type of vehicles was examined by [12]. This device utilized a surface that moved to channel the added kinetic energy to flow close to the surface of the roof.

The main aim of the research by [13] was to establish the effect of aerodynamics of different devices used for fuel saving in commercial vehicles such as the semi-trailer truck. A wind tunnel study was carried to calculate the vehicle's aerodynamic drag using a model that was 1/10th of an actual truck. The findings revealed that the external attachments such as the covering and the fairing had a substantial effect on the aerodynamic drag since they could lower the aerodynamic drag by about 26% compared to the base model according to the effects of the cross wind. The full-skirting (side skirting, front fairing, and gap filling) effect on the reduction of aerodynamic drag was maximum while using just the front fairing resulted in a minimum effect.

According to [14], the active flow control technique could be utilized to lower a vehicle's aerodynamic drag. This offers the potential of modifying the flow locally, removing or delaying the position of separation or reducing the recirculation zone development at the rear end and the separated twirling structure surrounding the vehicle. The findings demonstrated reductions in aerodynamic drag of nearly 15.83 % for the suction effect in addition to 14.38 % for the blowing effect. The road vehicle's aerodynamic drag is responsible for the main portion of a vehicle's consumption of fuel and results in almost 50% of the total consumption of fuel of the vehicle under normal highway speeds. [15] reported the study of research performance of passive and active flow controls of vehicles' aerodynamic drag reduction.

In recent years, there are a lot of works that investigated the drag reduction in heavy vehicles [16-19]. Some of them added particular afterbodies on these vehicle and some one used the different methods to do this work.

Base such consideration, in relation to the drag reduction in heavy vehicles, abundance of experimental and numerical works have been done but the geometry of device same as that is used in present work is not addressed in detail yet. And no one considers optimization of geometry of such device in its simulation.

In this study, a new device to reduce drag on commercial vehicles is investigated. At first, simplified dimensions of a commercial vehicle are modeled as extracted in literature and the coefficient of the drag is measured through the analysis of the fluid flow. After selecting various different geometries with the addition of the new device and picking out the best geometry, the coefficient of the drag in the new position is measured and a comparison is made with the base model.

II. ANALYSIS AND MODELING

For incompressible steady flow governing equations are Navier-Stocks and continuity equations.

The corresponding governing equations for fluid are:

Continuity

$$\frac{\partial \overline{u}_i}{\partial x_i} = 0 \tag{1}$$

Where $\overline{u}$ and x_i are time-average velocity and dimensional component respectively.

Momentum equation

$$\overline{u}_j \frac{\partial \overline{u}_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\upsilon \frac{\partial \overline{u}_i}{\partial x_j} + \upsilon_T(\overline{u_{i,j}} + \overline{u_{j,i}}) - \frac{2}{3}k\delta_{ij}\right) \tag{2}$$

Where ρ, $\overline{p}$, υ and υ_T are density, time-average pressure, kinematic viscosity and turbulence kinematic viscosity respectively.

For turbulence modeling standard *k-ε* Turbulence model is used.

Turbulence kinetic energy equation

$$\overline{u}_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j}\left[(\upsilon + \frac{\upsilon_T}{\sigma_k})\frac{\partial k}{\partial x_j}\right] - \left(\frac{2}{3}k\delta_{ij} - \upsilon_T(\overline{u_{i,j}} + \overline{u_{j,i}})\right)\frac{\partial \overline{u}_i}{\partial x_j} - \varepsilon \tag{3}$$

Where k , σ_k, σ_ε and ε are density, turbulent kinetic energy, constants in $k-\varepsilon$ model and turbulent dissipated energy respectively.

Turbulence dissipation energy equation

$$\bar{u}_j \frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j}\left[(\upsilon + \frac{\upsilon_T}{\sigma_\varepsilon})\frac{\partial \varepsilon}{\partial x_j}\right] - C_{\varepsilon 1}\frac{\varepsilon}{k}(\frac{2}{3}k\delta_{ij} - \upsilon_T(\overline{u_{i,j}} + \overline{u_{j,i}}))\frac{\partial \bar{u}_i}{\partial x_j} - C_{\varepsilon 2}\frac{\varepsilon^2}{k} \qquad (4)$$

Where $C_{\varepsilon 1}, C_{\varepsilon 2}, C_\mu$ are $k-\varepsilon$ model constants.

In this study, the system of equations has been numerically solved by the application of a 2D double precision model using CFD code based on finite volume procedure. The equations are discredited using Presto's scheme, which is similar to the staggered-grid scheme with a first-order upwind scheme for the convective terms. The Simple algorithm is used to solve the coupling between continuity and momentum through pressure. The convergence criterion in each case was $\left(\phi^{(i+1)} - \phi^{(i)}\right)/\phi^{(i)} < 10^{-8}$, where i denotes the iteration number and ϕ could stand for any of the dependent variables. It is worth noted that the mesh independency checked too. In all the simulations, the air is assumed to be incompressible, and density and viscosity are constant. The walls were adiabatic and no-slip wall conditions were used. A velocity inlet boundary condition was applied on the inlet, while on the outlet pressure outlet boundary condition was applied.

III. RESULTS AND DISCUSSION

In order to evaluate the model used in this work, obtained numerical results were compared with the experimental work of Wong and Mair [5].

The arrangement used in work of Wong and Mair [5] was as shown in Fig. 1. The model was basically a long rectangular block with a square cross-section. The elliptical nose was used in the planes of symmetry that the corners were rounded well.

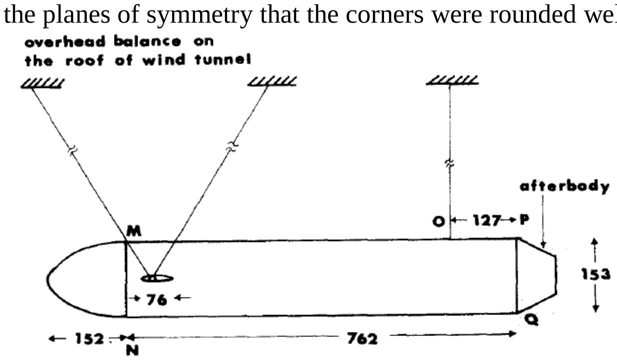

Fig. 1. Schematic of the model used in work of Wong and Mair [5].

The speed of wind was considered 42 m/s. This was consistent with the wind speed used by Wong and Mair [5], giving a Reynolds number of 4.4 × 10 [5] based on the maximum width of the body considered.

The Effect of short afterbody (Fig. 2) on base drag of a rectangular block at zero incidence has been compared with the experimental work of Wong and Mair [5] as shown in Fig. 3 and which shows a good compatibility.

There is a considerable drag reduction for values of β in the range 70-80 °.

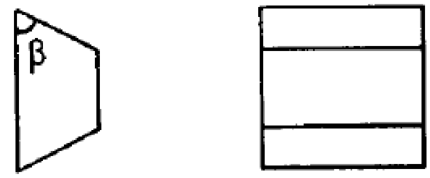

Fig. 2. Short afterbody used in experimental work of Wong and Mair [5]

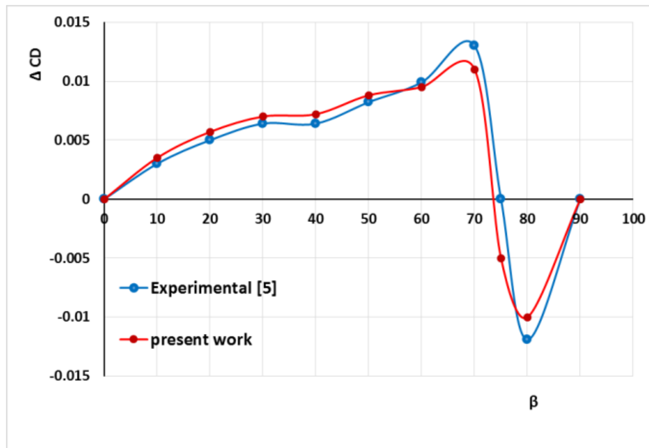

Fig. 3. Effect of short afterbodies on base drag of a rectangular block at zero incidence.

Fig .4. shows the Baseline tractor-trailer model used to compression. The standard dimensions (the tractor cab, wheels, semitrailer, ground clearance and the tractor-trailer gap are typical of real conditions as shown in Fig. 5.) have been selected based on work of Malviya et al [12].

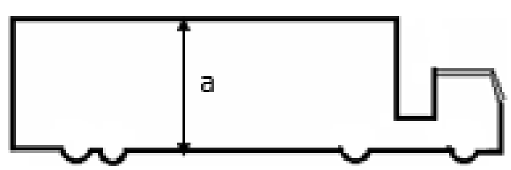

Fig. 4. Baseline tractor-trailer model.

This study utilized the velocity (V_{air}) of 19.5 m/s for the vehicles (thus free stream) as an appropriate base to compare with other devices on drag reduction.

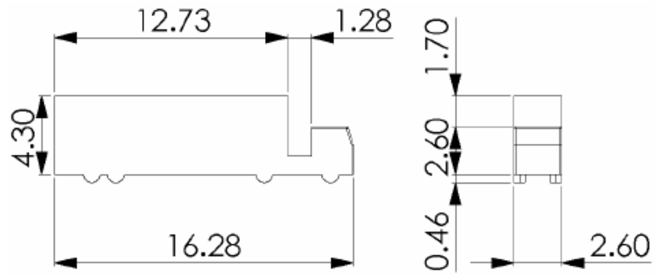

Fig. 5. Dimensions of baseline tractor-trailer model (m) used by Malviya et al [12].

The mentioned new device that attached to baseline tractor-trailer is shown at Fig. 6 and the b, c, r_1, r_2, and r_3 are the main parameters that were selected to investigate their influence on drag reduction of this new device. After selecting several different values for mentioned parameters as shown in table. 1, finally it is identified the maximum drag reduction equal 22%.

Fig. 6 shows the optimum values of mentioned parameters.

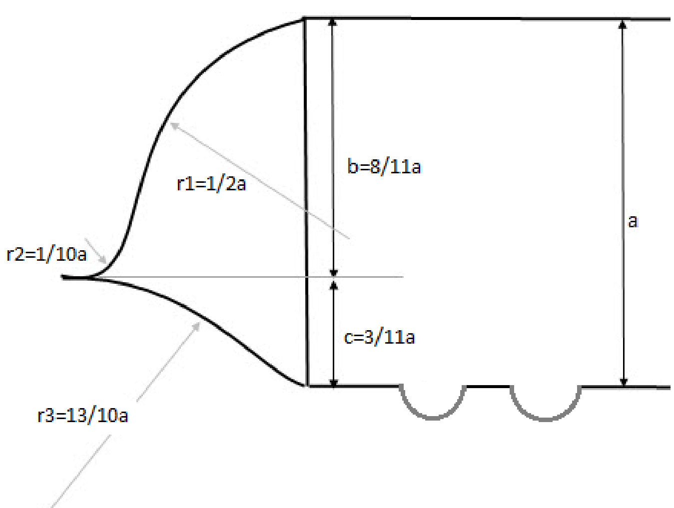

Fig. 6. Optimum values of parameters that were selected to investigate their influence on drag reduction of this new device.

TABLE I
PERCENTAGE OF DRAG REDUCTION FOR THE SELECTED PARAMETER RANGES

case	b	c	r1	r2	r3	Percentage of drag reduction
1	1/2a	1/2a	1/2a	1/12a	1/2a	10%
2	1/2a	1/2a	1/2a	1/7a	1/3a	8%
3	1/2a	1/2a	1/3a	1/9a	1/2a	11%
4	6/11a	5/11a	1/2a	1/10a	1/2a	13%
5	6/11a	5/11a	1/2a	1/8a	13/10a	15%
6	8/11a	3/11a	1/2a	1/10a	15/10a	18%
7	8/11a	3/11a	1/2a	1/10a	13/10a	22%
8	8/11a	3/11a	1/2a	1/10a	14/10a	20%
9	9/11a	2/11a	1/2a	1/10a	13/10a	15%

IV. CONCLUSION

In this study, a new device to reduce drag on commercial vehicles is investigated. The computational fluid dynamics (CFD) technique is utilized in the analysis of the airflow surrounding this device and the way in which the geometry of the device could be optimized to achieve a significant impact on the reduction of drag. Obtained numerical results of the model used in this work were compared with the experimental result and which showed a good compatibility. After choosing various different geometries with the addition of the new device and picking out the best geometry, the coefficient of the drag in the new position is measured and a comparison is made with the base model. The results indicate that there is about 22% drag reduction in the new state in comparison to the base model.

ACKNOWLEDGMENT

The author acknowledge Yasooj branch, Islamic Azad University for its financial support to this project.

NOMENCLATURE

$C_{\varepsilon 1}, C_{\varepsilon 2}, C_{\mu}$	k - ε model constant
g	acceleration of gravity [m/s^2]
i, j	grid coordinate [-]
k	turbulent kinetic energy [m^2/s^2]
$\bar{p}$	time-average pressure [N/m^2]
$\bar{u}$	time-average velocity [m/s]
x_i	dimensional component [m]
	Greek symbols
σ_k, σ_ε	constants in $k-\varepsilon$ model [-]
ε	turbulent dissipated energy [m^2/s^3]
ρ	density [kg/m^3]
υ	kinematic viscosity [m^2/s]
υ_T	turbulence kinematic viscosity [m^2/s]

REFERENCES

[1] DOT/SAE Truck and Bus Fuel Economy Measurement Study Report P59A. Report No. DOT/TSC – 1007, October, 1976.
[2] A. Wiley Sherwood - Wind Tunnel test of Trailmobile Trailers. University of Maryland Wind Tunnel Report No. 85. College Park, MD, April 1974.
[3] A. Wiley Sherwood - Wind Tunnel test of Trailmobile Trailers, 2nd Series. University of Maryland Wind Tunnel Report No. 85. College Park, MD, April 1974.
[4] 4. A. Wiley Sherwood P. R. Viswanath. (1996). Flow management techniques for base and afterbody drag reduction. Progress in Aerospace Sciences. 32, pp. 79-129.
[5] D. T. M. Wong and W. A. Mair, (1983). Boat-tailed afterbodies of square section as drag-reduction devices. Journal of Wind Engineering and Industrial Aerodynamics. 12, pp. 229-235.
[6] Z. M. Kassim, A. Filippone. (2010). Fuel savings on a heavy vehicle via aerodynamic drag reduction. Transportation Research Part D. 15, pp. 275-284.

[7] E. M. Wahba, H. Al-Marzooqi, M. Shaath, M. Shahin and T. El-Dhmashawy. (2012). in Aerodynamic Drag Reduction for Ground Vehicles using Lateral Guide Vanes. CFD Letters. 4, pp. 68-79.

[8] R. M. Wood and S. X. S. Bauer. (2003). Simple and Low-Cost Aerodynamic Drag Reduction Devices for Tractor-Trailer Trucks. SAE Technical Paper. pp. 1-18.

[9] R. Lopes, A. Delmas, and J. F. Sacadura, (2013). Improvement of Aerodynamics Characteristic of Heavy Trucks. 3rd International Conference on Trends in Mechanical and Industrial Engineering (ICTMIE'2013) January 8-9, 2013 Kuala Lumpur (Malaysia), pp. 246-255.

[10] R. M. Wood (2006), A Discussion of a Heavy Truck Advanced Aerodynamic Trailer System. 9th International Symposium on Heavy Vehicle Weights and Dimensions – USA – 2006. pp. 1-14.

[11] A. Seifert, O. Stalnov, D. Sperber, G. Arwatz, V. Palei. S. David, I. Dayan and I. Fono. (2001). the Aerodynamics of Heavy Vehicles II: Trucks, Buses, and Trains. 41, pp. 115-133.

[12] V. Malviya, R. Mishra and J. Fieldhouse. (2009). CFD Investigation of a Novel Fuel-Saving Device for Articulated Tractor-Trailer Combinations, Engineering Applications of Computational Fluid Mechanics. 3, pp. 587–607.

[13] H. Chowdhury, H. Moria, A. Ali, I. Khan, F. Alam and S. Watkins, (2013). A study on aerodynamic drag of a semi-trailer truck. Procedia Engineering. 56, pp. 201 – 205.

[14] Harinaldia, Budiarsob, R. Tarakkac and S. P. Simanungkalitd. (2011). Computational Analysis of Active Flow Control to Reduce Aerodynamics Drag on a Van Model. International Journal of Mechanical & Mechatronics Engineering IJMME-IJENS. 11, pp. 24-30.

[15] M. N. Sudin, M. A. Abdullah1 S. A. Shamsuddin, F. R. Ramli, Musthafah and M Tahir. (2014). Review of Research on Vehicles Aerodynamic Drag Reduction Methods. International Journal of Mechanical & Mechatronics Engineering IJMME-IJENS. 14. pp. 35–47.

[16] S. Li, J. Zhang, L. Ming, X. Liu and L. Jiang. (2011). Rear Add-on Device for Drag Reduction of Van body Truck. Advanced Materials Research. 308-310, pp. 1139-1142.

[17] L. Cao, X. Xie, J. Zeng and H. Huang. (2013). the Simulation Research on Air Drag Reduction of Tail Dome on Vans. Applied Mechanics and Materials. 345, pp. 48-53.

[18] D. G. Hyams, K. Sreenivas, R. Pankajakshan, D. S. Nichols, W. R. Briley and D. L. Whitfield. (2011). Computational simulation of model and full scale Class 8 trucks with drag. Computers & Fluids. 41. Pp. 27-40.

[19] M. Gazzola, O. V. Vasilyev and P. Koumoutsakos. (2011). Shape optimization for drag reduction in linked bodies using evolution strategies. Computers and Structures. 89, pp. 1224–1231.