ISSN: 2521-2292

Numerical Fluid Mechanics Study of Effect of Rear Slant Length on the Drag of an Idealized Road Vehicle

Pako Pollard Mogotsi , Thato Phologolo
Dept of Mathematics & Statistical Sciences
Botswana International University of Science & Technology
Palapye, Botswana
Pako.mogotsi@studentmail.biust.ac.bw
Thato.phologolo@studentmail.biust.ac.bw

N.Subaschandar
Dept. of Mathematics and Statistical Sciences
Botswana International University of Science and Technology, Palapye, Botswana
raos@biust.ac.bw

***Abstract*— In all kinds of shapes and form road vehicles use up energy in the form of fuel. It is interesting to note that on the highway more than half of the useful energy output of the engine is used to accelerate and heat the surrounding air molecules. Aerodynamic drag is the aerodynamic force that opposes motion and accounts for a large fraction of transportation energy consumption. The main objective of this paper is to provide a solution to reduce the drag of an idealized road vehicle. This is achieved by modifying the design of vehicle by changing one parameter and then verifying mathematically whether the modification has any effect in reducing drag. Idealized vehicle geometry has been used. It has been known that the rear top slant angle of about 6 degs reduces the drag. In this research, a numerical fluid mechanics study has been carried out by keeping the rear slant angle constant, but changing the rear slant length from 50% to 10% of the rear half of the top surface. The computational part consists of numerical simulation of the flow around the body employing CFD (Computational Fluid Dynamics) techniques. Computations were carried out with and without ground effects. It was observed that drag coefficient reduced as the slant length is reduced from 50% to about 80% and then the drag coefficient starts increasing with rear slant length from about 80% to 90%.**

Keywords— *Drag, turbulence, slant length, road vehicle*

I. INTRODUCTION

Studies show that a large fraction of goods worldwide are transported by road cargo transportation compared to other modes of transport. the excessive use of the road transport lead to numerous researches on how to increase the efficiency of the road cargo transportation. The industry of road vehicles is trying to develop various designs to reduce aerodynamic drag without constraints in the storage of the cargo. Also, the rapidly increasing fuel prices and the regulation of greenhouse gasses to control global warming give tremendous pressure on design engineers to enhance the current designs of the vehicle using the concepts of aerodynamics as to enhance the efficiency of vehicles.

Aerodynamic drag of a vehicle is directly related to the fuel consumption. In general, such vehicles are known to be aerodynamically inefficient compared to other ground vehicles due to their large frontal areas and bluff-body shapes. The inefficient aerodynamic shape results in excessive drag which leads to elevated fuel consumption rates. Contributions to heavy vehicle aerodynamic drag are mainly due to pressure drag, also known as form drag. It is estimated that the pressure drag on heavy vehicles accounts to more than 80% of the total aerodynamic drag [1], with frictional drag accounting for the remaining 20%. Any reduction in aerodynamics drag of a vehicle will help in reducing the fuel consumption thus helping in saving money and reducing the pollution caused by the burning of the fuel by the vehicles. With that motivation in mind, design engineers and manufacturers of vehicles, come up with different ways of reducing the aerodynamic drag of a vehicle. Retrofits for trucks aimed at improving aerodynamic performance and reducing the overall drag coefficient have recently shown great potential [2-4]. Examples of such drag-reducing devices include cab roof and side fairings, tractor and trailer side skirts, trailer-front fairings, vortex generators, and base-flaps. Studies for cab roof and side fairings show 9% to 17% drag reduction percentages on heavy vehicles [5-7]. Most of the studies carried out, as can be seen from the above references , are either experimental or real time empirical observations. Slant rear roof is one such technique used to reduce the drag of a road vehicle[1,3,6]. Most of the studies carried out, as can be seen from the above references, are either experimental or real time empirical observations. A numerical study presented in[8], estimated that a rear roof slant angle of 6^0 gave maximum reduction in drag of an idealized road vehicle. Also, their results showed that benefit occurred until the rear slant angle was about 12^0 and beyond that there was no benefit. However, the authors had fixed the rear roof slant length as 25% of the length of the vehicle. Since the flow over a three-dimensional road vehicle is very complex we consider an idealized configuration to represent the road vehicle. Also , because of the requirements of huge computing resources we

ISSN: 2521-2292

have modelled and carried out the research in a two dimensional space. The flow domain is assumed to be two dimensional and fluid is air. Steady, incompressible, constant viscosity and constant density assumptions have been made. Buoyancy and temperature effects have been neglected. The friction between the tires and the road, which happens in reality, has not been considered as this friction is common to all configurations and assumed to be uniform for all road vehicles. Also, drag due to the tractor in front of the trailer has not been considered as it is common to all configurations studied here. The two-dimensional CFD software Easy CFD has been used to carry out the computations and simulations and for data processing for all configurations[9].

Fig. 1 Picture of a road cargo transport vehicle without rear slant.

In the present study, the effect of the length of the rear slant length on the drag of an idealized road vehicle is presented. Fig. 1 shows the image of a road transport vehicle without rear slant. computational fluid dynamics(CFD) method has been used to carry out the research study. The computational fluid dynamics software EasyCFD has been used in this study. Easycfd is a computational fluid dynamics (CFD) software tool for the numerical simulation of two-dimensional and 3D axisymmetric fluid flow in a boundary fitted mesh[9].

II. COMPUTATIONAL DOMAIN

A two dimensional domain was used to study the effect of the rear slant length on the drag of an idealized road vehicle at a fixed angle of 6^0. The rear slant length started from the rear 50% of the vehicle and the slanted surface length decreased from 50% to 0% of the vehicle length, where 0% slant surface length represented a basic vehicle with any slant surface. For each configuration with different slant length the drag force acting on the truck was calculated and later used to calculate the drag coefficient which is defined by the Equation (1) bellow, where FD is the drag force, U_∞ is the airspeed, ρ is the air density, and A is the frontal area.

$$C_D = \frac{FD}{0.5 * \rho * U_\infty^2 A} \quad (1)$$

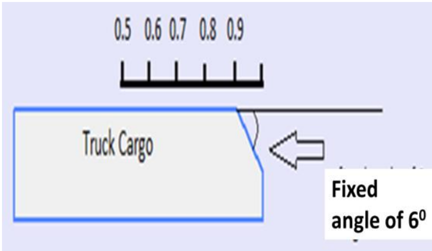

Fig 2: sketch of the geometry studied

III. MATHEMATICAL MODELLING

Governing Equations

1. Momentum Equations

- Horizontal Component

$$\frac{\partial(\rho u)}{\partial t} + \frac{\partial}{\partial x}(\rho u^2) + \frac{\partial}{\partial z}(\rho u w) = \frac{\partial}{\partial x}\left[\Gamma\left(2\frac{\partial}{\partial x} - \frac{2}{3}div\vec{V}\right)\right] + \frac{\partial}{\partial z}\left[\Gamma\left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right)\right] - \frac{\partial p}{\partial x}$$

- Vertical Component

$$\frac{\partial(pu)}{\partial t} + \frac{\partial}{\partial x}(\rho u w) + \frac{\partial}{\partial z}(\rho w^2) = \frac{\partial}{\partial z}\left[\Gamma\left(2\frac{\partial w}{\partial z} - \frac{2}{3}div\vec{V}\right)\right] + \frac{\partial}{\partial x}\left[\left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right)\right] - \frac{\partial p}{\partial z}$$

2. Continuity Equation

The conservation of mass law, or continuity equation, may be stated as:

$$\frac{\partial(\rho u)}{\partial t} + \frac{\partial}{\partial x}(\rho u) + \frac{\partial}{\partial z}(\rho w) = 0$$

ISSN: 2521-2292

3.The Standard k-ε Turbulence Model

The standard formulation of this turbulence model is described in [13].The turbulent viscosity is given by:

$$\mu_t = C_\mu \frac{\rho k^2}{\varepsilon}$$

The turbulence kinetic energy, k, as well as its dissipation rate, $\varepsilon[m^2/s^3]$, are computed with the following transport equations:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x}(\rho u k) + \frac{\partial}{\partial z}(\rho w k) = \frac{\partial}{\partial x}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial}{\partial x}\right] + \frac{\partial}{\partial z}[(\mu + \frac{\mu_t}{\sigma_k})\frac{\partial k}{\partial z}] + P_k - \rho\varepsilon$$

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial}{\partial x}(\rho u\varepsilon) + \frac{\partial}{\partial z}(\rho w\varepsilon) = \frac{\partial}{\partial x}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x}\right] + \frac{\partial}{\partial z}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial z}\right] + \frac{\varepsilon}{k}(C_1 P_k - C_2\rho\varepsilon)$$

The term P_1 represents the production rate of *k* as the results of the velocity gradients:

$$P_k = \mu_t[2\left(\frac{\partial u}{\partial x}\right)^2 + 2\left(\frac{\partial w}{\partial z}\right)^2 + (\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x})^2]$$

The remaining constants are:

$$C_\mu = 0.09, \quad \sigma_{k=} = 1.0 \quad \sigma_\varepsilon = 1.3$$

$$C_1 = 1.44 \quad C_2 = 1.9$$

where K is known as the turbulent kinetic energy and ϵ is known the dissipation rate of the turbulent kinetic energy.

Boundary conditions for this computational study are given as follows :

Inlet Velocity=Inlet= 40 m/s

Outlet = Pressure outlet 1 atm.

Wall (truck-trailer) Stationary wall with no slip condition 0 m/s

Outer Walls = Walls = no slip condition

Numerical simulation to the road vehicle for both the basic and modified profiles was obtained by using EasyCFD software. Meshing to the geometrical model was done by opting 'body fit' mesh insert technique to vary the mesh element size in the computational domain.. The computational domain was meshed with fine mesh near the model and coarse mesh away from the model. The fluid was assumed to be incompressible air and its properties were constant. The flow is assumed to be two Dimensional and turbulent. Standard K-ϵ turbulence model has been used whose equations are given above. The Navier-Stokes equations selected for the conservation of mass and momentum have been were also given above.

IV. RESULTS AND DISCUSSION

In this section we present the results of the current research. The computations have been carried out taking ground effects into consideration and without taking ground effects into consideration. However only results for configurations without ground effects are presented here.

In computational fluid dynamics , final results should not be influenced by the number of grid elements in the computational domain(mesh size). To eliminate the influence of mesh size on the final result, a grid independence study was conducted. For this, flow past a two dimensional rectangle without rear slant surface was computed by changing the mesh size and for each case drag coefficient was calculated. Table 1 gives the values of the drag coefficient for various mesh sizes. From this we can see the drag coefficient value lies in the range of the values quoted in standard literature[10] and also the difference in the values of drag coefficient for the last two meshes is around 0.2%, which may be considered very small. Hence all subsequent computations were carried out by keeping the mesh size around 3000 elements.

Table 1 : Grid Independence study

Number of elements	Coefficient of drag(C_D)	Difference(%)
2560	1.4791	-
2754	1.5903	2.12
2910	1.6064	1.01
3040	1.6096	0.2

ISSN: 2521-2292

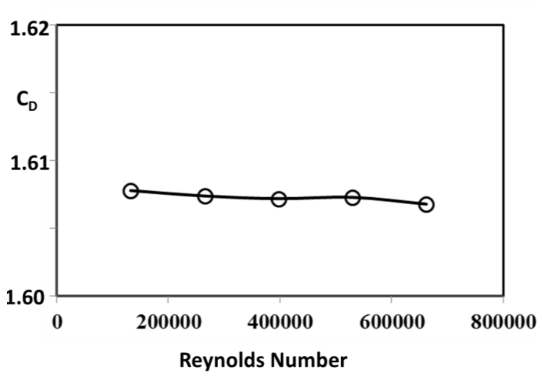

Fig. 3 : Variation of drag coefficient with Reynolds number

Figure 3 presents the variation of drag coefficient with Reynold number for the configuration without rear roof slant. Reynold number is defined as follows :

$$\text{Reynolds Number} = \frac{\mu U_{\infty} L}{\rho}$$

where L is the length of the vehicle , U_{∞} , μ is the viscosity of air and ρ is the density of air. From the figure 3 we can see that at low speeds the drag coefficient for a configuration without rear roof slant does not vary with Reynolds Number and the average value of the drag coefficient is matching with the drag values quoted in literature[10]

Table 2 : Variation of drag coefficient with slant length

Configuration	Slant Wall Length(s/L)	C_D/C_{D0}	Reduction Drag(%)
no slant	0	1.000	
90% slant	0.1	1.003	-0.29
80% slant	0.2	0.942	5.78
70 % slant	0.3	0.925	7.75
60 % slant	0.4	0.923	7.77
50 % slant	0.5	0.929	7.09

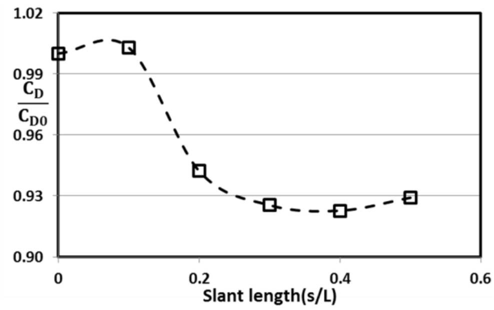

Fig. 4 Variation of drag coefficient with slant length

Table 2 and Figure 4 present the results of the current computations , namely , variation of drag coefficient with rear roof slant length. In Table 2 and Fig. 3, C_{D0} represents the drag coefficient when there no tailgate. From the table 3 and Figure 4 it can be seen that the drag coefficent initially increases and then starts decreasing as the rear roof slant length is increased reaching a minimum value of s/L = 0.4 which is equivalent to last 40 % of the vehicle length. After that it is increasing up to s/L=0.5 which is equivalent to the last 50% of the vehicle length upto which present computations were carried out.

V. CONCLUTIONS

The effect of rear roof length on the drag of an idealized road vehicle has been studied using computational fluid dynamics. The rear roof length was varied from s/L = 0 to 0.5. Effect of rear roof slant length on the drag coefficient with and without ground effect has been studied. Results on the effect of rear roof slant length on the drag coefficient without ground effect has been presented. It can be clearly seen from the results of this study that the rear roof slant length affects drag of an idealized road vehicle. The Drag coefficient initially increases with rear roof slant length and then starts decreasing recording a minimum value around s/L=0.4 which is equivalent to last 40% of the vehicle length. After that drag coefficient starts increasing reducing the benefit. The results show that a drag reduction upto 7.7 % can be achieved with a rear roof slant length of s/L=0.4. Studies with ground effect gave the same results.

REFERENCES

[1] Wood, R.M. Impact of Advanced Aerodynamic Technology on Transportation Energy Consumption. SAE Technical Paper 2004-01-1306, 2004, Washington.
[2] Wong H.Y., R.N. Cox, A. Rajan, Drag reduction of trailer-tractor configuration by aerodynamic means, Journal of Wind Engineering and Industrial Aerodynamics 1981, 9(1): p. 101–11.
[3] Hyams D.G., K. Sreenivas, R. Pankajakshan, D.S. Nichols, W.R. Briley and D.L. Whitfield. Computational simulation of model and full scale Class 8 trucks with drag reduction devices, Computers and Fluids 2011, 41(1): p. 27–40.

ISSN: 2521-2292

[4] Mohamed-Kassim Z and A. Filippone. Fuel savings on a heavy vehicle via aerodynamic
drag reduction, Transportation Research Part D 2010, 15(5): p. 275–284.

[5] Drollinger R.A. Heavy Duty Truck Aerodynamics. SAE Technical Paper 870001. 1986.
Washington, DC.

[6] Cooper K.R. Truck Aerodynamics Reborn-Lessons from the Past. SAE Technical Paper
2003-01-3376. 2003.

[7] Leuschen J., K.R. Cooper. Summary of full-scale wind tunnel tests of aerodynamic drag-

[8]. M.Mopipi, T.Phologolo and N. Subaschandar, "Numerical Fluid Mechanics study of effect of rear slant angle on the drag of an idealized road vehicle", Presented at the international Conference SAMSA2018, Southern African Mathematical Sciences Association, Palapye, Botswana, Nov 19-22, 2018.

[9]. http://www.easycfd.net/ , 2019, CFD software.

[10]. S.F. Hoerner, 1965, Fluid Dynamics Drag, Hoerner Publications, Bakersfield , CA , USA