ISSN: 2521-2292

STUDY OF EFFECTS OF TAILGATES ON THE DRAG OF AN IDEALISED ROAD VEHICLE USING NUMERICAL FLUID MECHANICS

Kenneth L. Bogatsu & Thato Phologolo
Dept of Mathematics and Statistical Sciences
Botswana International University of Science and Technology, Palapye, Botswana
kenneth.bogatsu@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 aerodynamic drag of road vehicle. This is achieved by modifying the design of vehicle by attaching a tailgate to the end of the trailer and then verifying mathematically whether the modification has any effect in reducing drag. Idealized vehicle geometry has been used. In this study, a numerical fluid mechanics study has been carried out extending the tailgate on one side as well as on both sides. The angle of the tailgate is changed to study the influence of the tailgate angle on the drag of the trailer. Computations were carried out with and without ground effects. It was observed that drag coefficient reduced as tailgate angle is increased upto 25^0 and after that the drag starts increasing reducing the benefit.

I. INTRODUCTION

Road vehicles have been considered as one of the major causes of air pollution due to high fuel consumption caused by burning a lot of fuel to overcome drag. During the designs of all attributes of various vehicles, reduction of fuel economy is considered to be the most important thing. In the process of car design, the aerodynamics must be seriously considered. Therefore improving the fuel economy of trucks will have tremendous impact on energy security, emission of greenhouse gases and cost of fueling when fuel price rises. According to Wolf-Heinrich Hucho, road vehicles are bluff bodies in very close proximity to the ground and have an extremely complex detailed geometry. As is typical for bluff bodies, drag (which is a key issue for most road vehicles) is mainly pressure drag which is estimated to account for more than 90% of the total drag. Improving vehicle aerodynamics is one of the factors that play crucial role for getting better performance including the handling of the vehicle especially at high speeds Energy efficiency of vehicles can be improved by altering the exterior body shape to reduce the aerodynamic 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. In the current we study the effect of presence of the tailgate at the end of the trailer. We study the addition of a single tailgate at the end of the truck trailer. Closing the tailgate actually improves fuel efficiency because it creates a type of airflow called a separated bubble within the bed of the truck. As air flows over the moving truck, that bubble of slow-moving air deflects it over the raised tailgate. By guiding surrounding air over the top of a truck, that vortex effect prevents added drag. If air flow gets complex as in flows over a bluff body, the flow becomes turbulent and it is impossible to solve Navier-Stokes and continuity equations analytically. To come up with a solution to this problem Computational Fluid Dynamics methods and software are widely used in the very recent years. 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 have modelled and carried out the research in two dimensional space.

II. DESCRIPTION OF THE PROJECT

ISSN: 2521-2292

In this section we describe the research work that has been carried out in this project. The tailgate angle has been changed and for each configuration the flow field over the road vehicle has been simulated and drag coefficient has been calculated. We monitor the angle of one tailgate from the horizontal top of the truck trailer, the effect of increasing or varying the angle between the horizontal top of the truck trailer and the tailgate flap. This is computed and simulated using the Computational Fluid Dynamics method which is a numerical method along with appropriate conditions for the results.

The tailgate is mounted at end of the trailer and at the top of as an extension to the trailer. In reality, the tailgate is designed in such a way that access to the doors is not impeded, they sleep on the door surfaces, so opening doors will not be a problem. Tapering the back top end of a long vehicle will increase its base pressure by providing pressure recovery of the surrounding air flow before it leaves the sharp back top edge. The increase in base pressure provides a lowered overall pressure difference from the front to the back of the truck semi-trailer combination. The tailgate works by streamlining the trailer geometry. The air follows the flaps of the tailgate around the corner..

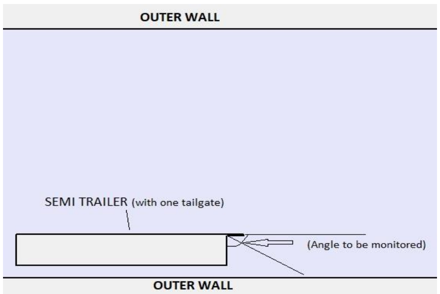

Fig 1: Sketch of the computational domain

Fig. 1 shows an idealized semi-trailer with a single tailgate aligned at the top of the horizontal part of the trailer, where the angle between the horizontal top of the trailer and the tail-gate flap is $\theta=0^{O}$. The computations have been simulated with and without ground effect, but the results have been presented with for configurations with ground effects because in reality road vehicles are very close to the ground. Hence ground effect plays a big role in aerodynamics of road vehicles. The inlet velocity is kept constant at 50m/s, which is about 180km/h.

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[8].

III. MATHEMATICAL MODELLING

THE NAVIER-STOKE EQUATIONS :

Horizontal component;

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

Vertical Component;

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

Continuity Equation

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

Turbulence Modelling Equations

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

K-Equation

ISSN: 2521-2292

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

ϵ- Equation

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial}{\partial x}(\rho u\varepsilon)+\frac{\partial}{\partial z}(\rho w\varepsilon)=\frac{\partial}{\partial x}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x}\right]+\frac{\partial}{\partial z}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial z}\right]+\frac{\varepsilon}{k}(C_1P_k-C_2\rho\varepsilon)$$

The modelling constants are :

$$C_\mu=0.09, \sigma_{k=}=1.0, \sigma_\varepsilon=1.3, \quad C_1=1.44, C_2=1.9$$

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 tailgate 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

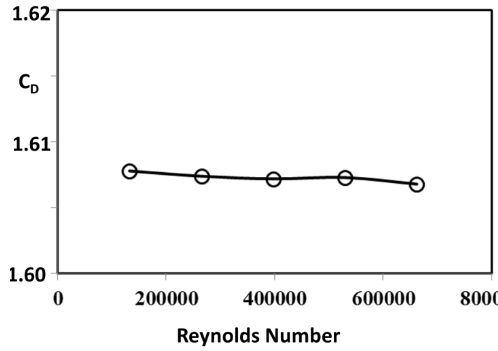

Fig. 2 : Variation of drag coefficient with Reynolds number

ISSN: 2521-2292

Figure 2 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 standard literature [9]

Table 2: Showing drag coefficient at different angles

Tailgate Angle	C_D/C_{D0}
0	1.000
15	0.942
25	0.937
30	0.946
45	0.984

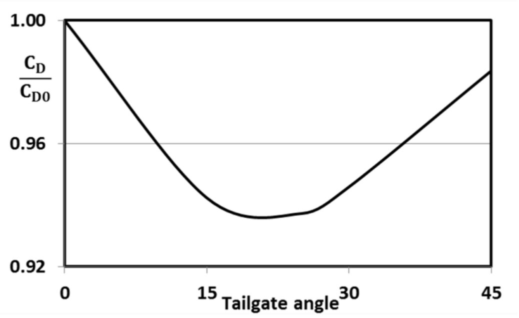

Fig 3: Drag coefficient variation with different angles of the tail-gate.

From Table 2 and Fig 3 we can notice that the drag coefficient was high at an angle of 0^0 and started decreasing as we increase the angle of the tail-gate with the horizontal top of the semi-trailer. In Table 2 and Fig. 3, C_{D0} represents the drag coefficient when there no tailgate. The drag coefficient kept decreasing up to an angle of about 25^0, then it started to increase with the increase of the angle of the boat-tails. At the tailgate angle of about 25^0 , the reduction in drag is about 6.3%. This reduction has been achieved for one tailgate. If we placed tailgates on all sides the resulting drag reduction will be more. However, in reality the flow is more complicated and the configurations are three-dimensional.

V. CONCLUSIONS

The effect of one tailgate on the drag of an idealized road vehicle has been studied using computational fluid dynamics. The tailgate angle was varied from 0^0 to 30^0. Effect of tailgate on the drag coefficient with and without ground effect has been studied. Results on the effect of tailgate on the drag coefficient with ground effect has been presented here. From all this simple modelling study we can see that tail-gating phenomenon is indeed possible and has a good impact on the drag coefficient an idealized road vehicles. It is shown that one tailgate mounted or placed at an angle of 25^0 will lower the amount of drag of an idealized road vehicle as much as by 6.3%. We can also conclude that the drag coefficient will be reduced further if tailgates are mounted on all sides at the end of the road vehicle.

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.
[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]. http://www.easycfd.net/ , 2019, CFD software.
[9]. S.F. Hoerner, 1965, Fluid Dynamics Drag, Hoerner Publications, Bakersfield , CA , USA