Effect of Surface Roughness on the Aerodynamic Performance of an Articulated Truck-Trailer Assembly

T. ASIM[a], K. UBBI[b], R.MISHRA[b], B. NSOM[c]

a. School of Engineering, Robert Gordon University, UK (t.asim@rgu.ac.uk)
b. School of Computing & Engineering, University of Huddersfield, UK (k.s.ubbi@hud.ac.uk r.mishra@hud.ac.uk)
c. Université de Bretagne Occidentale, IUT de Brest, IRDL UMR CNRS 6027, France (blaise.nsom@univ-brest.fr)

Abstract :

Aerodynamic drag is the most dominant resistive force to the motion of a vehicle, several methods of drag reduction have been applied over the years; these mainly include shape modification and add-on devices. A novel method to improve drag response of a truck is to modify the flow so that the flow remains uniformly attached to the vehicle. This will result in a smaller wake of the vehicle and consequently reduces drag. The present study investigates the feasibility of employing one such flow modifying technique. Surface roughness has been systematically introduced on the tractor-trailer unit in the form of spherical-shaped structures and the drag of the vehicle has been measured. Computational Fluid Dynamics (CFD) based techniques have been employed to numerically simulate the flow of air over the tractor-trailer unit. The results show that the surface roughness parameters affect the overall fluid flow around the truck-trailer unit, and hence have a significant effect on the aerodynamic drag force of the vehicle. Drag force has been observed to increase by 1% when valleys were introduced as surface roughness. However, drag reduction of 1.9% has been recorded in case of roughness peaks. The results suggest that although the peaks act as a restriction to the local flow, they contribute more towards reducing the overall drag. In case of the roughness valleys, the drag force increases because these valleys introduce more non-uniformity in the flow without any benefit in drag reduction.

Mots clefs : Computational Fluid Dynamics, Truck-Trailer, Surface Roughness.

1 Introduction

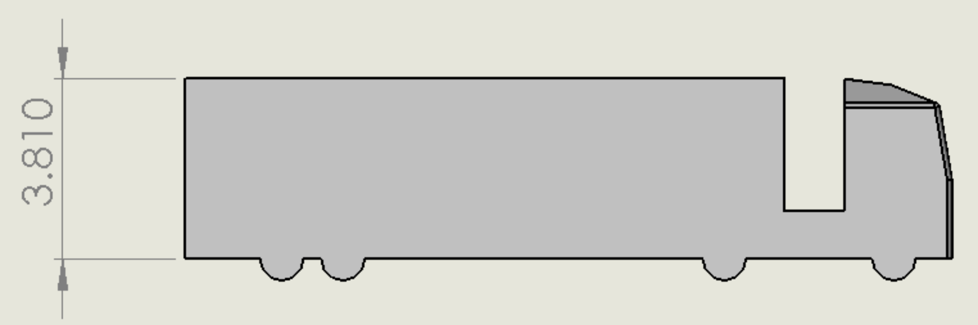
Heavy goods vehicles play an important role when it comes to inland goods transportation around the world. According to Malviya et al [1], around 70% of all the goods are transported by articulated tractor-trailer units. A typical large freight operator in the United Kingdom has approximately 600 tractor units and twice as many semi-trailers. Such a fleet would annually cover about 56 million miles, which translates to approximately £19.25 million in fuel. Hence, fuel economy of tractor-trailer units account for a major share of the overall fuel consumption in transportation sector. At speeds above 30mph, aerodynamic drag is the most significant resistive force and up to 50% of enigne power is comsumed to overcome this resistance. Even a small fraction achieved in fuel savings could mean savings of thousands of pounds.

The first aerodynamic development to reduce aerodynamic drag was the obvious rounding of the edges of the vehicle [2]. Due to the utilitarian nature of commercial vehicles and the need for internal loading space, aerodynamic development of such vehicles has historically been focused on add-on devices. The influence of side skirts and rear-end tapering of semi-trailers on the drag coefficient was found to be between 6% and 12% in various configurations [3]. The cab roof fairing has been the most significant contribution to reduction in aerodynamic drag of commercial vehicles [4]. Other aerodynamic flow modification devices like cab side collars, vortex trap devices [5] and A-pillar deflectors [6] have been researched extensively. Research has also been reported in the area of vehicle platoons (closely grouped vehicles) which shows promising reduction in aerodynamic drag and hence fuel consumption [7-8]. This concept however requires a large scale collaborative effort and extensive logistics scheduling for harnessing significant benefit.

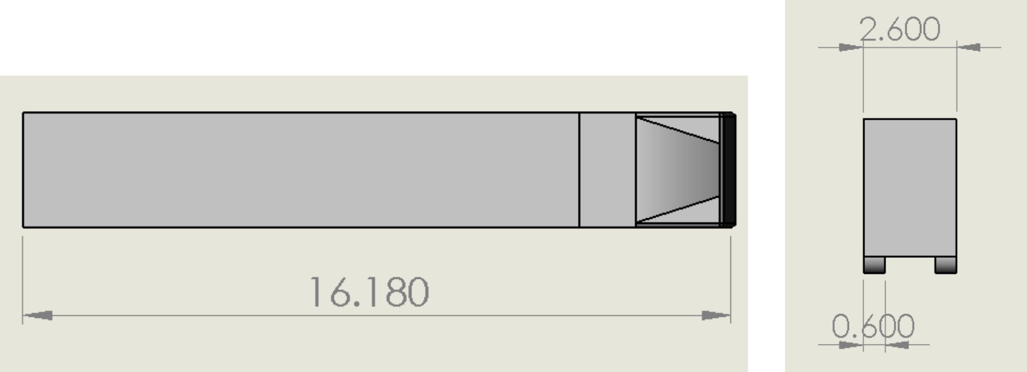
In the present study, surface roughness concept, similar to golf balls, has been been used to analyse its effects on the drag force experienced by a truck-trailer unit. The scale of surface roughness being considered in the present study is large i.e. addition/extraction of material from the surface of the unit, in the form of semi-spheres. Novel computational techniques have been employed to numerically simulate the flow around the vehicle. Three separate models of the vehicle have been created; first being the baseline model containing no surface features, second model that uses convex structures (peaks) and the third model that uses concave structures (valleys).

2 Methodology

A simplified model of a truck-trailer combination has been numerically modelled in a commercial CFD package known as Ansys. This 3D model has been imported into a large flow domain consisting of 3.84 million discrete volume elements. Flow of air has been numerically simulated at 25m/sec (56mph) through this discretised volume in order to predict pressure and turbulence fields on the surface of the truck-trailer uinit. Drag and lift forces acting on the models have been measured. Figure 1 show the side, top and rear views of the baseline tractor-trailer model respectively, where the different geometrical entities and the main dimensions (in m) of the baseline model are shown.



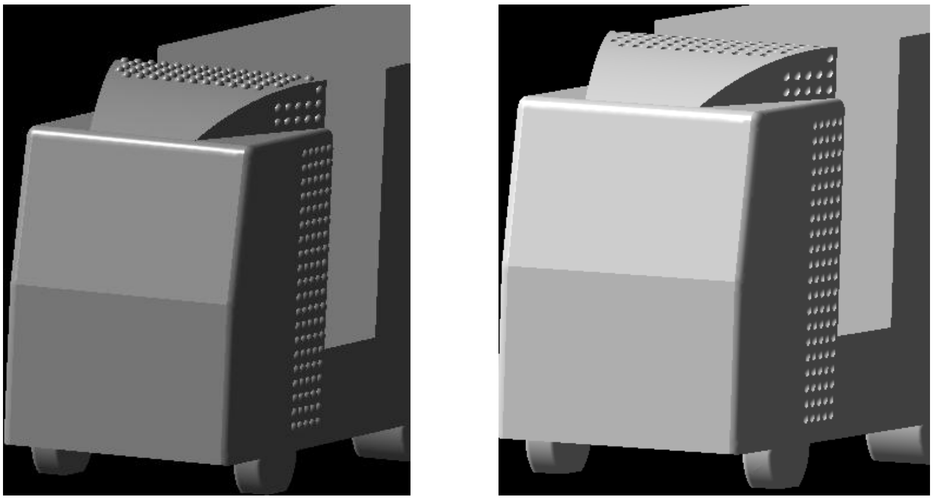
(a)



(b) (c)

Figure 1. Geometrical details of the baseline truck-trailer model (a) Side view (b) Top view (c) Rear view.

Figure 2 depicts the peaks and the valleys model. It can be clearly seen that the surface roughness effects have been included on the top and side surfaces of the fairing (cab deflector), and on the side surfaces on the cab. For effective comparison purposes, the number, location and the size of these surfaces have been kept the same for these two models, where the diameter and spacing between the spheres has been kept constant to 0.05m.



(a) (b)

Figure 2. (a) Peaks model (b) Valleys model.

3 Solver Settings

Three dimensional Navier Stokes equations, alongwith the continuity equation, have been numerically solved in an iterative manner for steady flow of air within the flow domain. Air turbulence has been modelled using Shear Stress Transport k–ω two equation turbulence model due to its superiority in predicting flow separation amongst other two equation turbulence models. Isothermal conditions have been considered in the present study, while the different surfaces of the truck, alongwith the various faces of the domain (except the bottom one) have been modelled as stationary walls, satisfying no-slip boundary condition. SIMPLE pressure-velocity algorithm has been specified in the present study, while 2nd order upwind spatial discretisation schemes have been used for pressure, velocity and turbulence properties.

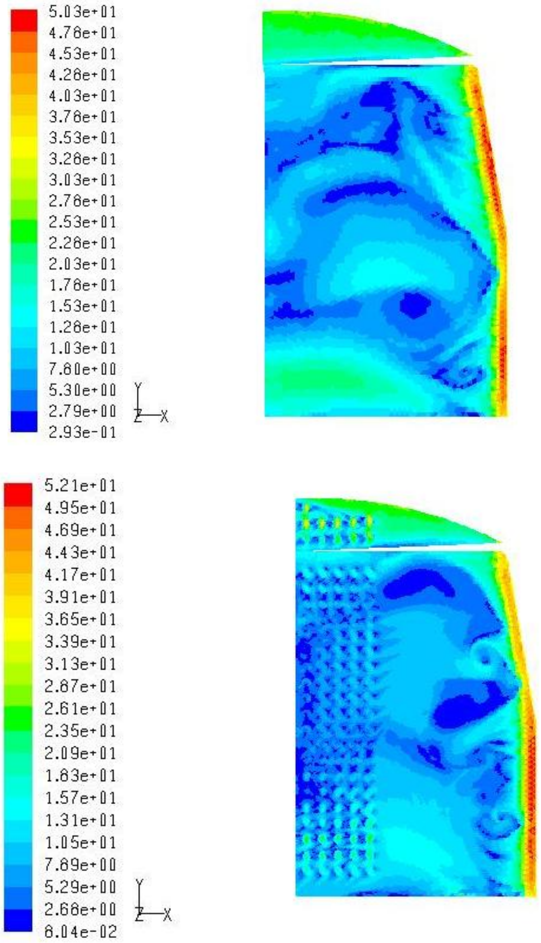
4 Results and Discussions

The 3D models of the three configurations of the truck-trailer unit have been numerically simulated to predict the aerodynamic response of the models, particularly drag. Flow velocity magnitude distribution on the different surfaces of thee models have been critically analysed.

4.1 Effects of Surface Roughness on Aerodynamic Drag

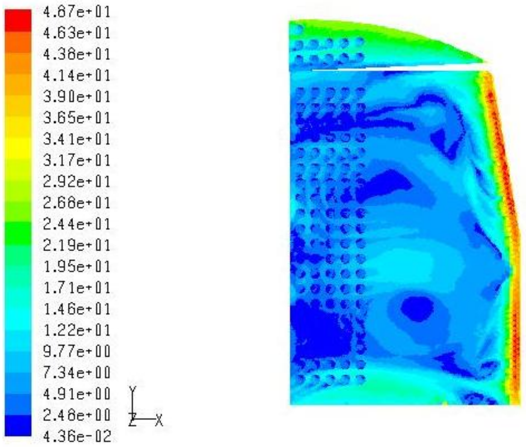
The flow field analysis carried out here allows better understanding of the flow behaviour and features responsible for aerodynamic drag acting on the vehicle. The flow behaviour over the key surfaces, which have been modified to introduce surface roughness, has been discussed in detail. Side faces of the cab form significant surfaces due to the addition of surface roughness features on them. Figure 3 depicts the velocity magnitude contours over these cab side surface for the baseline, peaks and valleys models respectively. Due to symmetry of geometry and flow, only one side has been discussed.

The velocity distribution over the cab side surface for the baseline model shows low velocity, predominantly between 5 and 12m/sec, over the entire surface. There are however small regions of velocity below 5m/sec shown by dark blue patches. The dominant velocity over the same surface in case of the peaks model is also similar to that observed in case of the baseline model i.e. between 5 and 12m/sec. However, it can be seen from this velocity distribution that the introduction of surface roughness peaks causes disruption in the normal flow behaviour. Low velocity regions are seen to form upstream of almost every peak on this surface.



(a)

(b)

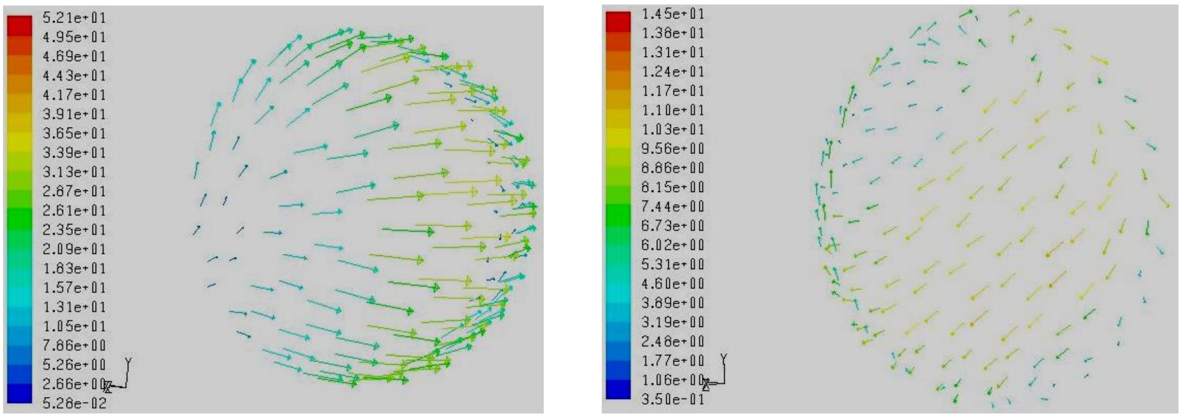


(c)

Figure 3. Velocity magnitude distribution over side faces of cab and fairing for (a) baseline model (b) peaks model and (c) valleys model.

In case of the valleys model, the velocity distribution over the cab side surface is nearly the same as the baseline model. This is primarily due to the surface roughness valleys which do not cause excessive disruption to the general flow over the surface. The dominant velocity over this surface is again seen to be between 5 and 12m/sec; however, the low velocity regions below 5m/sec are seen to be much larger in case of the valleys model. These regions dominate the rear part of the cab side where the surface roughness has been applied.

Figure 4 depicts the flow velocity behaviour corresponding to the peaks and the valleys respectively. Velocity magnitude vectors have been included for one peak and one valley. In these figures, the free stream direction is from left to right. It can be clearly seen that in case of the peaks, the flow decelerates to about 3m/sec before it flows over the peak, and accelerates to about 20m/sec as it flow over the peak geometry. On the other hand, when the flow passes over the valley, it is seen to travel in the opposite direction. Moreover, the maximum velocity of the flow in the valley is seen to be about 7m/sec. This behaviour can be attributed to causing a reduction in the overall velocity in the vicinity of the roughness valleys.

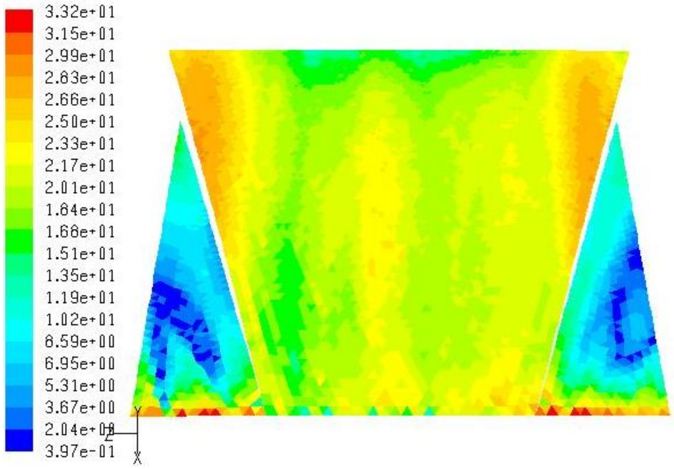


(a) (b)

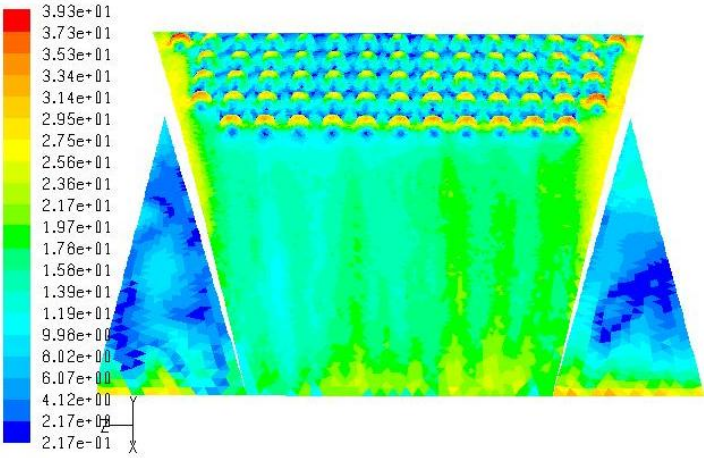
Figure 4. Flow velocity magnitude vectors on/within a unit roughness feature (a) peak (b) valley.

Figure 5 depicts the velocity magnitude contours on the truck's top surface for the baseline, peaks and valleys models respectively. It is seen that there is no appreciable difference between the velocity distributions over the cab top surfaces on either side of the fairing, with velocity being less than about 6m/sec over these surfaces. There is however noticeable difference between the velocity distribution over the top surface of the fairing. By preliminary visual inspection, it can be clearly seen that the net velocity magnitude is the lowest in the case of peaks model. In case of the baseline model this velocity near the surface is seen to be predominantly between 15 and 23m/sec. In case of the both the peaks and the valleys models it is observed to be between 10 and 17m/sec.

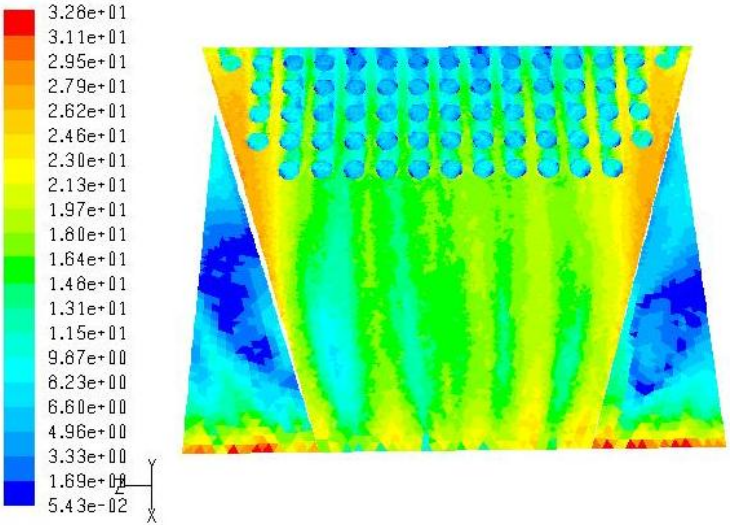
In case of the baseline model, the velocity over the extended portions on either side of the rear part is observed to be significantly high, with values between 25 and 30m/sec. This trend is also observed in case of the valleys model as the surface roughness valleys do not present a direct obstruction to the flow. On the contrary, in case of the peaks model, the roughness peaks directly obstruct the flow path, thus restricting the maximum velocity near the edges to about 27m/sec. The presence of peaks also causes a reduction in the extent of this high velocity region. Velocity higher than 35m/sec is observed at the topmost part of the peak structures as the flow accelerates over the spherical shape of these peaks. However, this also causes low velocity patches immediately upstream of the peak structures.



(a)



(b)



(c)

Figure 5. Flow velocity magnitude distribution over cab and fairing top surfaces for (a) baseline model (b) peaks model and (c) valleys model.

Overall velocity over the fairing top surface is observed to minimum (between 10 and 27m/sec) in case of the peaks model due to this obstruction to flow.

The overall drag force experienced by all these three configurations has been compared to identify key surfaces contributing to the aerodynamic drag acting on the vehicle. Table 1 shows a comparative itemisation of the drag force acting on the main surfaces of the vehicle. It can be seen from this table that surfaces that are most affected by adding surface roughness to the baseline model are the front and side faces of the cab, top faces the cab and fairing, and the front face of the trailer.

Table 1. Itemised comparison of drag force acting on key surfaces of the vehicle.

Vehicle Section	Surface	Drag Force		
		Baseline (N)	Peaks (N)	Valleys (N)
Truck (tractor)	Front	1892	1921	1898
	Sides	-701	-673	-663
	Back	761	736	776
	Top	-160	-133	-153
Trailer	Front	-603	-673	-651
	Sides	58	57	58
	Back	658	634	658
	Top	24	20	23
Total	(entire vehicle)	2221	2179	2244

It can be further noticed that variation in drag acting on the front, sides, back and top surfaces of the truck (tractor cab) are up to 1.5%, as low as -5.4%, between -3.2 and 1.9% and as low as -16.8% respectively. At the same time the effect on the same surfaces for the trailer is seen to be up to 11.6%, as low as -1.7%, -3.6% and -16.6% respectively. These values suggest that the addition of peaks and valleys to the baseline model mainly affects the truck's sides, top, back and the trailer's front surfaces. The overall drag for the baseline model is found to be 2221N. This is seen to increase by 1 % to 2244N for the valleys model, and decrease by 1.9 % to 2179N for the peaks model.

5 Conclusions

The effect of the surface roughness parameters has been investigated. Three different models were considered for the CFD analysis. Firstly, the baseline model with no surface roughness parameter introduced was discussed and the flow behaviour analyzed. Then the surface roughness was introduced in the model; as the semi-circular peaks and as the semi-circular valleys. The results show that the surface roughness parameters affect the overall fluid flow around the truck–trailer and hence have an effect on the aerodynamic drag force of the vehicle. Pressure and velocity magnitude distributions have been extensively investigated, in conjunction with the velocity magnitude vectors, to explain the flow behaviour around the models. Drag force was seen to increase by 1% when valleys were introduced as surface roughness. However, drag reduction of 1.9% was obtained in case of roughness peaks.

The results suggest that although the peaks act as a restriction to the local flow; they contribute to reducing the overall drag. In case of the roughness valleys structures, the drag force increases because these valleys introduce more non-uniformity in the flow without any benefit in drag. Hence, the effectiveness of the peaks for drag reduction in a truck-trailer is significant and can prove to be of vital importance for the reduced fuel consumption of such articulated vehicles. Peaks of different roughness heights and spacing can be used to further optimise their effectiveness for reducing overall aerodynamic drag.

Références

[1] V. Malviya, R. Mishra, J. Fieldhouse, CFD Investigation of a Novel Fuel-Saving Device for Articulated Tractor-Trailer Combinations, Engineering Applications of Computational Fluid Mechanics3 (2009) 587-607
[2] W.H. Hucho, Aerodynamics of Road Vehicles, 4th ed, Warrendale, PA (USA): Society of Automotive Engineers, 1998
[3] S. Matěj, N. Jiří, Aerodynamic Devices to Reduce the Base and Underbody Drag of Semitrailer Unit, Proceedings of 4th International Conference on Advanced Engineering Design, Glasgow, UK, 2004
[4] F.W. Haegert, Aerodynamic structure for semi-trailer trucks, US Patent 4750772, 1988
[5] R. Wood, S. Bauer, Simple and low-cost aerodynamic drag reduction devices for tractor-trailer trucks, Proceedings of International Truck & Bus Meeting & Exhibition, Ft. Worth (TX), USA, 2003
[6] M.M. Sullivan, One piece A-pillar air deflector and windshield molding, 2005
[7] M. Hammache, Aerodynamic Forces on Truck Models, Including Two Trucks in Tandem, University of California, Berkeley, USA, 2001
[8] F. Browand, Reducing Aerodynamic Drag and Fuel Consumption, 2005