

Numerical and Experimental Investigation on Vehicles in Platoon

Paolo Schito and Francesco braghin
Politecnico di Milano

ABSTRACT

Many studies have been carried out to optimize the aerodynamic performances of a single car or a single vehicle. In present days the traffic increases and sophisticated technologies are developing to guarantee the drivers safety, to minimize the fuel consumption and be more environmentally friendly. Within this research area a new technique that is being studied is Platooning: this means that different vehicles travel in a configuration that minimizes the aerodynamic drag and therefore the fuel consumption and the longitudinal space. In the present study platoons with different vehicles and configurations are taken into account, to analyze the influence of car shape and relative distance between the vehicles. The research has been carried out using CFD techniques to investigate the different flow fields around different platoons, while wind tunnel tests have been used to validate the results of the CFD simulations. The results show that vehicles with an estate back, like station wagons and sport utility vehicles, give a drag reduction up to more than 50% when running in platoon, while vehicles with a fast back, like sedan, that have better performances than sharp-back cars when running isolated, cannot reach the highest drag reduction obtained by the estate back cars. Trucks show negligible drag reduction when running in platoon with other cars, while the vehicles behind the truck experience a very low drag force. The distance between vehicles plays a very important role since smaller distances between the cars give lower aerodynamic forces, and may be minimized only by developing better technologies to control and maintain the position of the vehicle in the platoon without any risk for the passengers.

CITATION: Schito, P. and braghin, F., "Numerical and Experimental Investigation on Vehicles in Platoon," *SAE Int. J. Commer. Veh.* 5(1):2012, doi:10.4271/2012-01-0175.

INTRODUCTION

The definition of the aerodynamic characteristics of commercial cars has been an important field of research. There are many open points, and a lot of studies must be performed on the aerodynamic interactions between different cars. The growth of world population points out the need of advanced infrastructures and more efficient transport systems. The approach used until now is the construction of new infrastructures, but this solution cannot be applied forever. A different approach that has been taken into account by California PATH (California Partners for Advanced Transit and Highways) is the Automated Highway System (AHS) [1]. In this system the vehicles travel in rows becoming platoons to use the aerodynamic interaction between the different vehicles in the convoy to lower the fuel consumption. To ensure the safety of the entire convoy the vehicles must be equipped with Intelligent Transport Systems (ITS) like distance sensors, adaptive braking systems (ABS), Global Positioning System (GPS) since the distance between the cars can be lower than a car length. Before optimizing the

ITS it is important to verify the aerodynamic performances of vehicles travelling at close distances.

Some studies on this topic have been performed by the railway community to improve the aerodynamic performances of freight trains (Watkins et al. [17]). PATH conducted some research more in detail on road vehicles. It is important to note that the advantages of the reduction of aerodynamic drag will reduce fuel consumption mainly for vehicles running at higher speeds and constant velocity, since for lower velocities the aerodynamic drag is negligible while when changing the speed inertial forces play a significant role in fuel consumption. The study of the aerodynamic behavior of a car is typically realized with wind tunnel tests or in full scale road tests: wind tunnel tests allows to perform sensitivity analysis for different configurations with almost known boundary conditions, but on the other hand these tests are expensive and for most cases it is necessary to realize scaled models that could not have all characteristics of the real vehicle; full scale tests will experience the real conditions, but are sensible to ambient conditions (wind

intensity and direction, rain) that causes a wide spread in the measured results. Nowadays it is possible to use CFD with a good confidence in order to reduce costs and time-to-market but it is important to validate the numerical model with experimental tests to ensure the quality of the result. Aerodynamic tests on a simplified geometry of a vehicle (the Ahmed Body) were performed by Gillieron and Chometon [2] that compared experimental results and numeric calculations for the back angle of 25 and 35 degrees using a standard k- ϵ calculation. The numerical simulation predicted in a good way the vortices and the drag trend: the higher drag values were imputed to a higher calculated back pressure. Makowski and Kim [3] conducted numerical simulations with standard k- ϵ and with RSM (Reynolds Stress Model) on an Ahmed Body with a backlight of 30°: standard k- ϵ over-predicts viscosity where turbulence is non-isotropic, while RSM reproduces correctly all characteristics observed in the experimental tests; Strachan et al. [4] compared the experimental results of an Ahmed Body wake measured with a Laser Doppler Anemometer with a k- ϵ ReNormalization Group simulation: the counter rotating vortices were captured in a good way, but there is a discrepancy in the position that has been imputed to a misalignment of the model in the wind tunnel tests. Unsteady LES (Large Eddy Simulations) simulations where conducted by Krajnovic and Davidson [5] and by Hinterberger et al.[6] showing that very accurate results can be obtained with this technique that is quite time and energy demanding. Kapadia et al. [7] compared the results of a DES (Detached Eddy Simulations) with Gillieron and Chometon [2] and experimental data showing a higher capability of DES to capture the instable structures of the wake with respect to RANS equations. First wind tunnel tests on car platoons have been presented by Zabat et al. [13] and [14], where they measured that the mean drag on a 4-vehicle platoon at half a vehicle distance is 38% lower than the non platoon value, while for a closer distance of 0.2 vehicle the drag reduction is 44%. The investigations on the wake interaction between different vehicles have been performed by Abdel Azim and Abdel Gawad [8], Barnard [9], Watkins and Vino [10], California PATH [11]. Gotz [12] studied the aerodynamic interaction between commercial vehicles at 80 km/h and 40 m spacing, predicting a drag reduction of 20% for the second vehicle and a 30% for the rest of the platoon. Watkins and Vino[10] in particular made experiments on a platoon of two Ahmed Bodies noting that for a close distance the drag of the trailing vehicle had a higher value than the one of the first vehicle. California Path [11] studies denoted that a closer distance in the platoon decreases the drag coefficient for each vehicle in the platoon, the last vehicle of a platoon has a higher drag than the other vehicles and that platooning is not efficient for vehicle distances higher than 10L (ten times the length of a vehicle). Bonnet and Frits [15] studied the drag reduction of a platoon of two trucks at different distances, and they measured that the fuel consumption reduction at a speed of 80 km/h and a distance of 10m was 21% for the tail vehicle. Browand and Hammache [16]

demonstrated that the interaction of two vehicles in platoon is influenced not only by the inter-car distance, but also by the shape of the leading and trailing part of the vehicle.

DESCRIPTION OF THE WORK

This work has been conducted analyzing two different problems: the first one is the application of the platoon concept to cars modeled as Ahmed Bodies, while the second is the proposal of some platoons that can be easily observed on a common road almost worldwide.

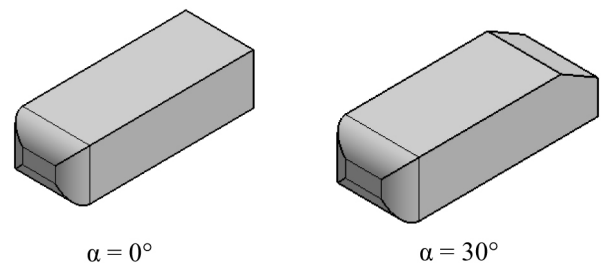


Figure 1. Geometry of the considered Ahmed Bodies: on the left the Ahmed Body with 0° backlight angle; on the right the Ahmed Body with 30° backlight angle.

The study proposes at first the aerodynamic performance of a single vehicle that will be compared with the behavior of the same vehicle in a platoon. Ahmed Body results will be compared with literature results while for the car models dedicated wind tunnel tests have been conducted to validate CFD simulations results.

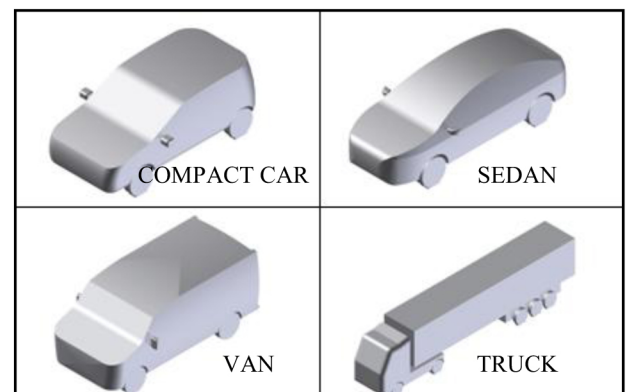


Figure 2. CAD geometry of the considered vehicles. The geometries are simplified with respect to the real cars, but some details are still present.

The aim of the work is the survey of the value of the drag of the vehicles. In particular, this particular aerodynamic force will be reported in non-dimensional form as a drag coefficient:

$$C_D = \frac{F_D}{\frac{1}{2}\rho V^2 A}$$

where F_D is the aerodynamic force, ρ the air density, V the relative fluid velocity and A the frontal area of the vehicle. The aerodynamic performance of the vehicle is also reported as reduction of drag coefficient with respect to the single isolated vehicle as:

$$\Delta C_D = C_D / C_{D0} \quad (2)$$

where C_D is the drag coefficient of the vehicle in the platoon and C_{D0} is the drag coefficient of the isolated vehicle.

NUMERICAL MODELING

This study has been conducted through finite volume Computational Fluid Dynamic (CFD) approach. To solve the Unsteady Reynolds Averaged Navier-Stokes (URANS) equations a domain for the calculation is defined and a typical example is shown in Figure 3. The domain realized to reproduce the flow around the single car or the platoon of vehicles is big enough to not affect the result of the calculation, but small in order to save computational nodes and for this reason limit the time needed to solve the problem. It is important to note that only half of the domain is reproduced: the setup is symmetrical as well as the expected flow, and suitable boundary conditions are imposed on the symmetry plane. Typical dimensions of the domain is a height of 5 times the height of the vehicle, and a width of 10 times the width of the vehicle. The longitudinal length depends on the number of vehicles that composes the platoon, but the space behind the cars allows to fully develop the wake and has typically 5 vehicle lengths before the first vehicle and 14 vehicle lengths after the last one. The elements close to the surface of the vehicles have a small size in order to be able to capture the influence of the details of the vehicles while moving away from the vehicles the size of the cells becomes larger because the variations in the flow are lower and don't need a very refined representation. Y^+ on the vehicle is 100 and therefore standard wall functions are used. The total number of cells of the grid is between 700.000 and 4 million, depending on the analyzed configuration. The cell used to discretize the domain are tetrahedral in order to fit the complex geometry that is reproduced. Due to the typical unsteady characteristics of the separation behind bluff bodies (as cars can be considered), the simulations solve the unsteady Reynolds Averaged Navier-Stokes equations with the k- ϵ realizable turbulence model. The time-step used for the calculations is 10^{-3} seconds and the time history that was used for the averaging of the forces is 20 seconds (20000 time-steps). The software used to solve the flow equations is the commercial Ansys code.

Appropriate boundary conditions are fixed on the domain boundaries, defining the velocity at the inlet (30 m/s), imposing a zero velocity on the vehicle and on the ground, while a zero normal velocity has been imposed on the top and the sides of the domain. The boundary condition on the ground can be considered "wrong", since in the real world the

vehicle moves with respect to the ground, but, since in this work a comparison between numerical and experimental data will be performed, the simulations are performed in analogy with the experimental setup, where the wind tunnel tests have been performed fixing the vehicles on a flat plate. The length of the Ahmed body is 1m, while the dimension of the real car models is the correct dimension full-scale (3.8m the compact car, 4m the van, 4.8m the sedan and 16.4m the truck). The Reynolds number for the simulations is $Re=8 \cdot 10^6$.

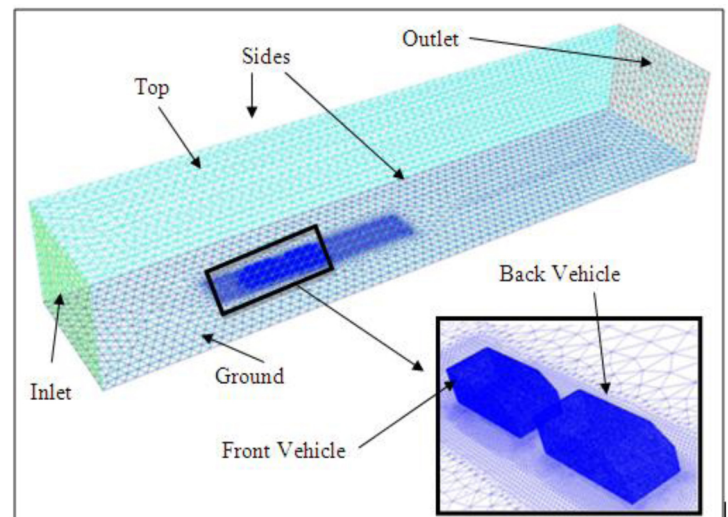


Figure 3. Example of the typical grid used for CFD simulations. It is possible to note the coarser outer domain, the fine grid in the zone close to the vehicles also reported in the enlargement.

EXPERIMENTAL SETUP

Experimental wind tunnel tests have been conducted on 1:40 scaled models of the vehicles. The car models have been realized in Styrofoam and covered with a hard finishing in order to reproduce the correct surface roughness. On the lower part of the car is fixed a support in order to hold the vehicle in position and, if the drag must be measured, to fix the vehicle himself to the load cell that will measure the drag force on the model. The vehicles are positioned on a flat plate (as can be seen in Figure 4) that is divided in two in order to allow every distance between the vehicles during the tests.

The facility used for this study is a closed circuit wind tunnel in use at the Aerospace Engineering Department of Politecnico di Milano. The tests section is 1.5m high, 1m wide and 3m long. The range of velocities that is possible to test goes up to 50m/s. In this case the wind tunnel tests have been performed on 1:40 scale models with a wind speed of 20m/s and therefore the Reynolds number of the experiments is around $Re=1.3 \cdot 10^5$. A pitot tube has been positioned at the reference height of the vehicle in order to measure the reference velocity and obtain the correct drag coefficient. The load cell positioned under the flat plate has been shielded from the incident flow to keep it protected from the pressure