

Effect of the Frontal Edge Radius in a Platoon of Bluff Bodies

Thomas Gheysens
TU Delft

Gandert Van Raemdonck
Wabco

ABSTRACT

This paper investigates the effect of the frontal edge radius of the Generalized European Transport System (GETS) model on the aerodynamic behavior of three different vehicles in a platoon. Due to the increasing awareness around harmful gases and depleting oil sources, more sustainable transport systems are needed. The efficiency of long-haul, heavy-duty vehicles can be increased by driving in platoon formation in order to decrease aerodynamic drag and to increase fuel mileage. The drag behavior of drafting vehicles is already studied but differences in aerodynamic drag reductions occur on the trailing vehicle. Some studies indicate a drag increase while others predict a drag decrease. In this study, a numerical investigation was performed solving Reynolds Averaged Navier-Stokes equations with the aid of a commercial package. Four different inter-vehicle distances were tested for the vehicles with several different frontal edge radii. The simulations were validated using wind tunnel experiments on an isolated GETS model. The drag of the lead vehicle showed an asymptotic behavior, increasing towards the value in isolation when the inter-vehicle distance is increased. The drag of the trailing vehicle showed very different behavior. The frontal edge radius was critical in determining the drag increase or decrease of the vehicle. Due to the platoon, the drag of the front part of the trailing vehicle was increased when the frontal edge radius was small. This lead to a total drag increase for the trailing vehicle at specific inter-vehicle distances.

CITATION: Gheysens, T. and Van Raemdonck, G., "Effect of the Frontal Edge Radius in a Platoon of Bluff Bodies," *SAE Int. J. Commer. Veh.* 9(2):2016, doi:10.4271/2016-01-8149.

INTRODUCTION

In the last years, there has been an increasing awareness of climate change, as demonstrated in the 2015 climate conference in Paris recently [1]. The most polluting road transport mode in the European Union are the long-haul tractor-trailer combinations, they are responsible for 30% of the on-road CO₂ emissions [2]. Platooning can be a successful concept for decreasing the fuel consumption of these heavy-duty vehicles. Track tests with European tractor semitrailer combinations showed fuel savings up to 21% for the trailing vehicle and 7% for the leading vehicle, with respect to the value in no traffic conditions, with a 10 m spacing and a velocity of 80 km/h [3]. Platooning of heavy duty vehicles is becoming more and more an additional manner to further improve the fuel consumption. In 2016, the European Truck Platooning Challenge took place to stimulate the implementation of platooning on the public roads [4].

The principle of platooning is that the following vehicle drives in the wake of the vehicle in front. This way, the low pressure in the lead vehicle's wake increases while the high pressure in the stagnation region of the following vehicle decreases. Both vehicles experience reduced drag forces due to the platoon [5]. Literature indicated a broad range of sometimes contradicting drag reductions when driving in a platoon [6], [7].

By testing 1/8 scale models of a Van in the wind tunnel, the Partners for Advanced Transportation Technology (PATH) project [8] showed that for a 2-vehicle platoon, at spacing greater than one vehicle length, the lead vehicle is almost unaware of the presence of the trailing vehicle. This trailing vehicle however, already experiences a decrease in drag. Lowering the spacing below one vehicle length decreases the drag of both vehicles below 70% of the value without a platoon and in no traffic conditions. For a spacing below 0.5 vehicle length, the drag of the trailing vehicle increases abruptly and becomes larger than the lead vehicle drag. In platoons with more than two vehicles, the interior vehicles experience the lowest drag.

Using Computational Fluid Dynamics (CFD) based on the time accurate Lattice-Boltzmann Method, Smith et al. showed that while the drag of the lead truck always decreases with decreasing spacing, the drag of the trailing truck started to rise again for the shortest spacing. The reason for this is that besides reducing the stagnation pressure on the trailing vehicle, the wake of the leading vehicle also reduces the suction pressure on the curved frontal surfaces of the trailing vehicle. This was observed for yaw angles of zero and six degrees. These findings correspond to trucks like the Peterbilt Model 386. It must be noted that most trucks in Europe do not resemble this geometry [9].

Hammache and Browand [10] tested a tandem of both rounded and blunt trucks. Using these two different bluff body shapes, four different platoon configurations were tested in the wind tunnel. For spacings between zero and one vehicle length, there is a strong interaction. For larger gap widths there was a low interaction region. Every configuration contributes to lower drag coefficients compared to an isolated value. In the strong interaction region, the relative drag coefficient of the rear body starts at a value of 20% (blunt) or 40% (rounded) of the isolation value. When enlarging the spacing, a steep increase in drag coefficient is noticed for the trailing body. It was observed that applying a rounded edge on the leading body always reduces the drag on that body as well as on the total platoon. Rounding the front of the rear body however, increases the relative drag coefficient of the rear body and of the total platoon due to the loss in suction force on the leading edges. The most beneficial configuration is thus a rounded body followed by a blunt body. The highest relative drag coefficient occurs for the platoon with a blunt front body and a rounded rear body. The blunt-blunt and the rounded-rounded configurations behave alike in the strong interaction region, but in the low interaction region, the rounded-rounded configuration has a higher relative drag coefficient and it even becomes higher than for the blunt-rounded configuration after a spacing of 2.5 vehicle length. In all the configurations except for the rounded-rounded configuration, the trailing vehicle experienced significantly lower drag forces. For the rounded-rounded configuration, the drag of the trailing body was lower than that of the front body, but it was the other way around in the low interaction region. It must be noted that the non-dimensional gap width, $G/\sqrt{A}$, where G is the spacing and A the frontal area of the model, was normalized using the squared frontal area instead of the vehicle length [10]. A remark has to be made that these results should not be interpreted as absolute drag values. The drag coefficient is normalized using the drag coefficient in isolation, which is higher for the blunt body than for the rounded geometry. This can cause the rounded body to experience a higher relative drag coefficient.

The current study investigates the effect of changing the forebody shape of a vehicle on the behavior of the drag of a three model platoon. Also the effect of applying a boattail to the vehicle is studied. Both the drag of the individual vehicles as well as the platoon averaged drag will be considered. The objective is to have an understanding of the flow characteristics that cause some following vehicles to experience an increase in drag but other following vehicles to experience a decrease in drag.

NUMERICAL SET-UP

Computational Geometry and Domain

Heavy-duty vehicles have complex geometries. In order to reduce the computational costs and effort for the meshing, a model with a simplified geometry is used, i.e. the Generalized European Transport system (GETS) model [11]. This model suffices in capturing the primary effects and minimizes secondary effects due to the complex details of heavy-duty vehicles. This approach of simplifying the vehicle geometry has already been carried out successfully in other studies [7],[12]. During the simulation, all surfaces of the model are categorized into the following parts: front, sides, rear and boattail

surfaces (if applied). It should be noted that the side surfaces also include the top and bottom surfaces. The geometry of the model can be seen in Figure 1.

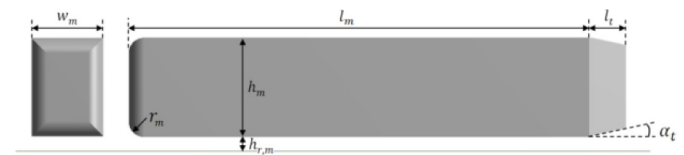


Figure 1. Geometry of the used GETS model.

The length of the model, l_m , is 16.5 m. The width, w_m and height, h_m , are respectively 2.6 m and 3.5 m. The ride height, $h_{r,m}$, is 0.5 m. The frontal edge radius, r_m , is varied between 0.135 m, 0.27 m and 0.54 m. This model is used to set up different platoon configurations. Four different inter-vehicle distances are used, they are derived from four time intervals between the vehicles, namely 0.15 s, 0.30 s, 0.60 s and 0.90 s. Using the driving velocity of 25 m/s to convert them to distances and the vehicle length to normalize them leads to the normalized inter-vehicle distances of 0.23, 0.45, 0.91, 1.36. These distances will be presented by the parameter D .

The model is placed in a computational domain with a width of twelve times the model width and a height of six times the model height. The domain extents up to six model lengths after the model and two model lengths in front of the model. For the condition without cross wind, the blockage ratio of the model was 1.34%.

Computational Grid

A hybrid grid is generated with both prism, pyramid and tetrahedral cells. In order to capture the flow phenomena in the boundary layer, the cells were refined close to the model. The dimensionless wall unit, y^+ , was kept between 30 and 300 so that wall functions could be used [13]. Since the dimensionless wall unit is based on the wall shear stress, this unit will not be larger than 30 on the entire surface of the model. Therefore enhanced wall treatment is used. This function uses wall functions if the y^+ value is within the valid range and uses the near wall treatment when it is small enough.

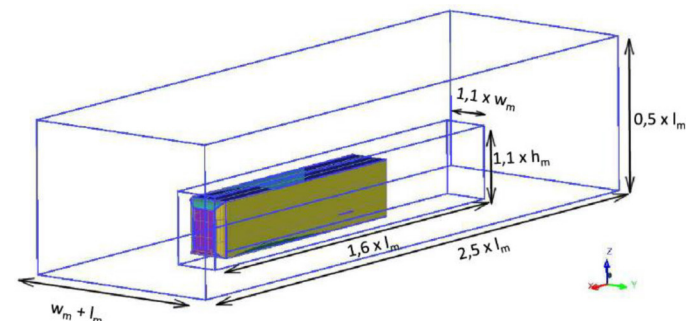


Figure 2. Dimensions of the mesh density boxes around the GETS model.

To capture the relevant flow characteristics and still keep the amount of cells reasonable, the mesh is only refined in two areas. To do so, two density boxes are used in the domain. One contains the immediate surroundings of the model and its wake, while the other surrounds a larger area around the models. The dimension of these boxes can be seen in Figure 2. The mesh sensitivity study was

performed to determine the dependency of the results on the refinement of the mesh. An asymptotic behavior of the drag coefficient with increasing number of cells was clearly seen [14].

Boundary Conditions

At the inlet of the domain, a uniform velocity of 25 m/s is imposed. The turbulence intensity was set to 5%, this value was found in extensive road tests [15]. The back of the domain act as an outlet with outflow properties. The side surfaces and top surfaces are symmetry planes in the case of symmetric wind conditions. When cross wind conditions were simulated a lateral velocity component was added to the incoming flow. The right side acted as an extra inlet and the left side as an extra outlet. The outflow through both inlets was weighted such that mass conservation in every direction was preserved. This method was already applied successfully [12] and is more time efficient than rotating the model and re-meshing the domain. The bottom of the domain was a moving wall in order to create the ground effect. This wall only had a movement in longitudinal direction to represent the driving velocity without interference of wind conditions.

Using the described model and boundary conditions results in a Reynolds number of 5.1×10^6 with the square root of the frontal area of the model as reference length. This is the Reynolds number that is also experienced by trucks in real life situations. The boundary layer can be considered fully turbulent and is modeled as such.

Governing Equations and Solver Settings

The 3D steady Reynolds Averaged Navier Stoker (RANS) equations were solved. To close this system of equations, the Menter $k-\omega$ SST model was used. It was observed that the Menter $k-\omega$ SST model performed the best when compared to four other two equation models [16].

The pressure-based coupled solver is used in ANSYS Fluent since this yields superior performance to the segregated solver. This is also recommended by SAE International guidelines [13]. The gradients of the solution variables at the cell centers are determined using the Green-Gauss Node-Based approach [17]. This approach minimizes false diffusion and is thus recommended for tetrahedral meshes. Both the momentum, turbulent kinetic energy, specific dissipation rate and pressure are discretized with the default scheme (1st order / standard) for 150 iterations. After these initial iterations, the pressure is discretized with a second order scheme. The other variables use a second order upwind scheme for the spatial discretization [13]. The required limits for the convergence residuals are also adapted in order to have a fully converged flow. The criteria are set to 10^{-5} for the continuity and velocity residuals and to 10^{-4} for the turbulent kinetic energy, k , and the turbulent dissipation rate, ω . The flow field is initialized using the flow parameters from the velocity inlet.

Validation of the Results

The numerical results from this study are evaluated using the data from a sub-scale wind tunnel test on the single GETS model by Van Raemdonck [11]. The Reynolds number of the windtunnel

experiment was 8.2×10^5 , which is almost one order of magnitude lower. However, a Reynolds sweep was performed by Van Raemdonck [11] and it was seen that above the windtunnel experiment's Reynolds number, the drag of the model remained constant with increasing Reynolds number.

Both the forces on the model as well as the pressure distribution over the model surfaces are compared. The force coefficients and the pressure distribution along the frontal area and top surface are well captured by the simulations. The corrected drag coefficient was measured to be 0.284 in the wind tunnel while the simulations resulted in a drag coefficient of 0.288. The pressure distribution along the frontal and top surfaces is shown in Figure 3.

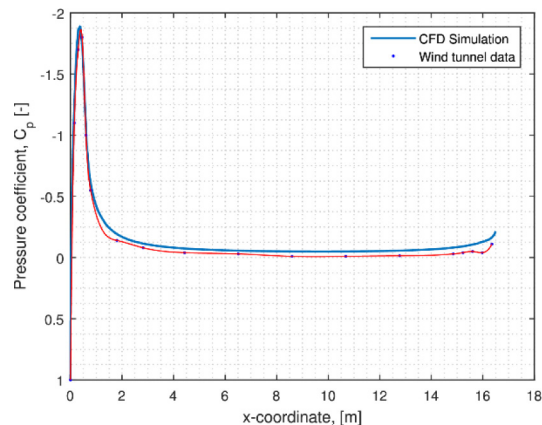


Figure 3. Pressure coefficient distribution over the top surfaces of the model for CFD simulations and experiments [11].

The simulated pressure distribution on the rear surface did not resemble the measured values well, this is shown in Figure 4. This is due to the highly unsteady nature of the wake structure, which is characterized by a large turbulence intensity. These characteristics cannot be captured by a steady RANS solver. However, for the scope of this project, the simulation results for the single vehicle are sufficient in order to capture the most important effects.

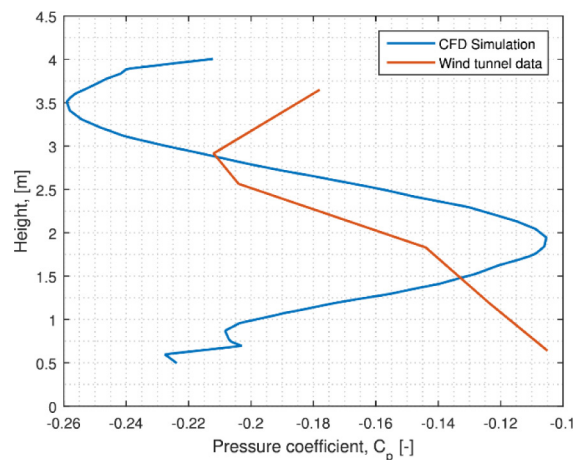


Figure 4. Pressure coefficient distribution over the symmetry line of the rear surface of the model for CFD simulations and experiments [11].