

# Computational Prediction of a Vehicle Aerodynamics Using Detached Eddy Simulation

Natalia Castro, Omar D. Lopez and Luis Munoz  
Universidad de Los Andes

## ABSTRACT

In the present paper, Computational Fluid Dynamics (CFD) simulations of the aerodynamics of a station wagon using DES (Detached Eddy Simulation), based on the Spalart-Allmaras turbulence model, are discussed and compared with experimental results. DES is a non-zonal hybrid turbulence model that uses both Reynolds Averaged Navier-Stokes (RANS) and Large Eddy Simulation (LES), enabling a better equilibrium between the accurateness and the computational cost of the solution.

Simulations were run in parallel using the commercial software ANSYS/FLUENT v13.0, and required a computational grid of approximately 50 million cells. Some flow characteristics such as boundary layer separation, recirculation zones, and the entire pressure and velocity field were also obtained and analyzed. Computational results of drag force were compared with experimental results based on the SAE J1263 recommended practice (which describes a way to calculate the resistive loads that act on a vehicle during non- forced deceleration). Changes were made in the experimental estimation of the drag force, compared to the procedure stated in SAE J1263 (e.g., a differential formulation of the system was used, and the mass of the vehicle was altered by loading extra bodies into the vehicle). A selection among different sets of experimental conditions (vehicle velocities and vehicle weights) was made to improve the accuracy of the prediction.

Even though computational and experimental approaches involved different models and simplifications, results from each of them showed a fair agreement. Congruence was driven by the iterative improvements made on the boundary conditions and meshing parameters on the CFD approach, and the careful experimental design performed. The aerodynamic drag coefficient calculated by CFD was of 0.460 compared to 0.404 when calculated by experimental means, which represents a 13.8% difference or 56 drag counts.

**CITATION:** Castro, N., Lopez, O. and Munoz, L., "Computational Prediction of a Vehicle Aerodynamics Using Detached Eddy Simulation," *SAE Int. J. Passeng. Cars - Mech. Syst.* 6(1):2013, doi:10.4271/2013-01-1254.

## INTRODUCTION

The use of Computational Fluid Dynamics (CFD) has become a standard practice in the automotive development in the last decades, increasing the understanding of flow dynamics around road vehicles and improving the prediction of aerodynamic forces and moments. One of the biggest challenges of CFD is the simulation of turbulent external flows. Specifically in the case of ground vehicles, several characteristics such as: 3D flow, large separation regions, complex vortical structures, ground effects, among others, increases flow complexity. One of the several models used for simulating ground vehicles' aerodynamics is Reynolds Averaged Navier Stokes (RANS) which has shown to be reliable and robust, but fails to predict the complexities involved in these flows. The increase of computational resources in the last years has allowed the research on Large-

Eddy Simulation (LES) models for ground vehicle aerodynamics. LES has shown improved capabilities as well as a high computational cost, in particular close to the walls where Quasi-Direct Numerical Simulation (QDNS) is required. One way to overcome this difficulty is to use wall treatment methods; one of these methods is Detached-Eddy Simulation (DES). DES model incorporates both, RANS and LES models, using RANS in near wall regions and LES in turbulent zones (e.g., vehicle wake). This combination enables a better equilibrium between the accurateness and the computational cost of the solution.

According to Krajnovic et al [1], the computational cost of performing a full LES simulation of a road vehicle is still expensive and beyond the capabilities in the near future. Nevertheless, LES can be applied to the study of vehicle aerodynamics, but for simple geometrical configurations such

as Ahmed body at low Reynolds number. These studies allow the understanding and exploration of the flow dynamics around a vehicle, given details of the transient flow that cannot be reproduced in experiments or RANS simulations. Hinterberger et al [2] performed a simulation of the flow around an Ahmed body with an slat angle of  $25^\circ$  and  $Re=2.8 \times 10^6$  achieving promising results with satisfactory agreement with experimental data, despite some discrepancies in the lower part of slant back. Temmerman et al [3] also reported some differences between LES computations of the flow around an Ahmed body and the experimental results at  $35^\circ$  of slant angle.

Hybrid RANS-LES models capabilities (over RANS and LES) have been tested in CFD simulations of aerial vehicles especially at high angles of attack in which massively separated flow is expected [4], [5]. According to Squires [4], in massively separated flow the division between RANS and LES regions is very clear so that DES is suitable. The potential of DES in the simulation of the flow around bluff bodies have been widely proven providing satisfactory results at a lower computational cost in comparison to LES. The fact the DES is suitable for massively separated flows and flow around bluff bodies makes it very attractive for the simulation of the flow around road vehicles. For example, Islam et al [6] discuss the implementation of DES in an open source CFD toolbox called OpenFOAM showing that many of the relevant issues of the vehicle aerodynamics are well predicted with DES. In this study, it is also pointed out the importance of using a parallel CFD code in simulations using DES for design purposes due to its computational cost. Maddox et al [7] performed a DES simulation of a simplified Ground Transportation System with a Reynolds number of  $2 \times 10^6$  using the commercial software Cobalt. Numerical results shows a substantial better agreement in the drag coefficient (at  $0^\circ$  yaw) predicted by DES than RANS. Discrepancies were observed at high yaw angles, which were related to the actual laminar to turbulent transition not predicted by the model.

Another numerical method that is commonly used in the simulation of the flow around ground vehicles is Lattice-Boltzmann (LBM). PowerFLOW and Xflow (commercial softwares) have been proven to be successful in the simulation of automotive applications using LBM. For example, in the field of aeroacoustics, Uchida et al [8] performed a computational simulation of the flow around a simplified vehicle A-pillar with PowerFLOW. Using grids of 8 to 17M lattices, Uchida et al were able to compute and predict the surface fluctuating pressures. Even though, the peak sound pressure was overpredicted, the overall trend of the power spectra was well captured from the LBM simulations. Fares [9] developed a simulation of the flow around an Ahmed body using PowerFLOW v4.0, using as a turbulence model VLES (Very Large-Eddy Simulation) and D3Q19 type lattices. The slant angle of the Ahmed body was simulated for  $25^\circ$  and  $35^\circ$ , especial interest was focused on

the separation behavior of the wake. Excellent agreement is reported between the computational and experimental results. The capabilities of the implemented VLES were demonstrated in both capturing the unsteady behavior of the flow and the averaged flow phenomena. Drage et al [10] used the commercial software XFlow based on LBM, in order to study the unsteady vehicle aerodynamic phenomena, including moving geometries. Three examples were shown: A simplified reference vehicle called ASMO with pitching and vertical motion due to cross wind fluctuation; a race car with pitching and vertical motion due to aerodynamics forces at high speed; and an overtaking maneuver between a car and a truck. The potential and capabilities of LBM was shown and proven without the time consuming remeshing process needed in a classical method based on finite volume.

In order to validate the computational results, experimental measurement of the drag coefficient has to be performed. There are two tests to accomplish this measurement: wind tunnel or coast down tests, the former is preferable due to its controlled and repeatable conditions. Steady state CFD simulations are comparable with wind tunnel results. Differences between wind tunnel and coast down results have been previously reported in the literature. For example Hong et al [11] studied the drag coefficient of two vehicles in tandem with both wind tunnel and coast down tests, finding differences up to 17% of the drag coefficient between them. Howell et al [12] measured the drag coefficient of a compact SUV with both on-road and wind tunnel tests. The MIRA wind tunnel was used at a nominal test speed of 100 km/h, while coast-down test were performed at a starting speed of 120 km/h for about 70 to 80 seconds. The speed of the SUV at the end of the coast-down test was between 20 and 40km/h. After performing corrections to the wind tunnel data, it was found an absolute difference between the methods of 0.01, and the cause of this discrepancy was not understood or explained. Several efforts have been done to correctly correlate wind tunnel with coast-down results. Walter et al [13] proposed a methodology to infer the coast-down drag coefficient, based on wind tunnel results for the Lockheed-Martin Low-Speed Wind Tunnel. This was achieved by a careful uncertainty analysis of the experimental data and the obtained correlation. Eaker [14] performed a similar study for the General Motors Aerodynamics Laboratory achieving good agreement between both methodologies with a drag coefficient difference of less than 2%. In the present study, difference between the experimental and computational results are expected since the CFD simulations were performed with a constant flow speed of 70km/h, while the coast-down tests were performed at a maximum starting speed of 78 km/h. The vehicle final speed was between 30 to 20 km/h.

The present paper discusses the computational simulation of the flow around a commercial station wagon using DES in order to explore its capabilities in the prediction of the drag coefficient. Numerical results are compared with experimental results based on SAE J1263 recommended

practice. The Reynolds number in the computational model is  $2.7 \times 10^6$  based on the wheel based. The simulation was performed with the commercial software ANSYS FLUENT v13.

## THEORETICAL BACKGROUND

Turbulence modeling in CFD can be achieved with three different approaches: direct numerical simulation (DNS), LES and RANS. According to Spalart [15], DNS will not be available for aerodynamic applications in the next 60 years.

On the other hand, LES can be used today but it requires very large computational resources. RANS and hybrid RANS-LES are the most promising turbulence models in recent years, especially for computational aerodynamic problems. In a RANS model the Navier-Stokes (NS) equations are averaged (in time or ensemble), which produces new terms called Reynolds stresses that need to be modeled. Several models have been proposed for modeling the Reynolds stresses and they are classified according to the number of extra differential equations that have to be solved besides the NS equations. Among these models, the Spalart-Allmaras (SA) [16] is one of the most frequently used for the simulation of external flows, and aerodynamics flows in particular. The simplicity and reduced computational cost, with satisfactory numerical results, are the main characteristics that make the SA model very attractive for aerodynamic applications. The extra differential equation used for this model is called the SA equation and it represents a transport equation for the model variable  $\tilde{v}$  (see equation 1).

$$\frac{\partial \tilde{v}}{\partial t} + U_i \frac{\partial \tilde{v}}{\partial x_i} = \frac{1}{\sigma_{\tilde{v}}} \left[ \nabla \cdot ((\nu + \tilde{v}) \nabla \tilde{v}) + C_{b2} (\nabla \tilde{v})^2 \right] + P(\tilde{v}) - D(\tilde{v}) \quad (1)$$

where  $P(\tilde{v})$  and  $D(\tilde{v})$  represent the production and destruction terms respectively (given by equations 2 and 3) while  $\sigma_{\tilde{v}}$  and  $C_{b2}$  are model constants.

$$P(\tilde{v}) = C_{b1} \tilde{v} \left( S + \frac{\tilde{v}}{\kappa^2 d^2} f_{v2} \right) \quad (2)$$

where  $S$  is the magnitude of the vorticity,  $f_{v2} = 1 - \frac{\chi}{1 + \chi f_{v1}}$

where  $\chi = \frac{\tilde{v}}{\nu}$  and  $f_{v1} = \frac{\chi^3}{\chi^3 + C_{v1}^3}$ . The destruction term is given by

$$D(\tilde{v}) = C_{w1} f_w \left( \frac{\tilde{v}}{d} \right)^2 \quad (3)$$

where  $f_w = g \left[ \frac{1 + C_{w3}}{g^6 + C_{w3}} \right]^{1/6}$ ,  $g = f + C_{w2}(r^6 - r)$  and  $r = \frac{\tilde{v}}{5\kappa^2 d^2}$ . It is important to notice that the destruction term depends on the distance to the wall ( $d$ ) as expected in any RANS model.

Finally, the model is closed with the calibration and definition of the constants as follows:  $C_{b1} = 0.1355$ ,  $C_{b2} =$

$0.622$ ,  $\sigma_{\tilde{v}} = 2/3$ ,  $C_{v1} = 7.1$ ,  $C_{w1} = \frac{C_{b1}}{\kappa^2} + \frac{1 + C_{b2}}{\sigma_{\tilde{v}}}$ ,  $C_{w2} = 0.3$ ,  $C_{w3} = 2$  and the Karman constant  $\kappa = 0.4187$ . The turbulent viscosity and the model variable are related by equation 4.

$$\nu_t = \tilde{v} f_{v1} \quad (4)$$

DES is obtained by redefining the distance to the wall as  $\tilde{d} = \min(d, C_{DES} \Delta)$  where  $C_{DES}$  is a model constant equal to 0.65 and  $\Delta$  represents the LES filter. This new definition of  $d$  allows the model to behave as RANS close to the wall (attached boundary layer) and as LES in the rest of the domain. The transition between RANS and LES is controlled by the mesh.

## Experimental Approach Formulation

The experimental tests performed to estimate the aerodynamic drag coefficient were based on the SAE J1263 [17]. This norm describes unforced (natural) deceleration tests that allow estimating the forces opposing to the motion of a vehicle (i.e. aerodynamic drag force and rolling resistance). The equation of motion for the longitudinal dynamics of the vehicle considers a reduced order model for the drag force, as well as a reduced order model for the rolling resistance [18]. The model used for the drag force is shown in equation 5, and it neglects any wind interaction.

$$D_A = 1/2 \rho C_D A V^2 \quad (5)$$

where  $D_A$  is the drag force acting on the vehicle,  $\rho$  is the air density,  $C_D$  is the drag coefficient of the vehicle,  $A$  is the frontal area and  $V$  is the velocity of the vehicle. The drag coefficient  $C_D$  can be seen as a measure of the efficiency of the shape of the vehicle and depends on the vehicle geometry, the surface rugosity and the flow's Reynolds number. For the experimental process,  $C_D$  was assumed to be dependent only on the geometry.

The model used for the rolling resistance is shown in equation 6. It neglects any lift force and also neglects the vertical dynamics of the vehicle.

$$R_X = f_r W \quad (6)$$

where  $R_X$  is the rolling resistance force acting on the vehicle,  $f_r$  is the rolling resistance coefficient and  $W$  is the weight of the vehicle. Rolling resistance coefficient  $f_r$  depends on the road characteristics and the tires characteristics and inflation pressure [19].

The equation of motion for the longitudinal dynamics considers as opposing forces the combination of the drag and rolling resistance forces. For a given condition on a flat road