

Validation of Aerodynamic Simulation and Wind Tunnel Test of the New Buick Excelle GT

Fuliang Wang, Zhangshun Yin, Shi Yan, and Jia Zhan
 Pan Asia Technical Automotive Center Co

Heinz Friz, Bo Li, and Weiliang Xie
 Exa Corporation

ABSTRACT

The validation of vehicle aerodynamic simulation results to wind tunnel test results and simulation accuracy improvement attract considerable attention of many automotive manufacturers. In order to improve the simulation accuracy, a simulation model of the ground effects simulation system of the aerodynamic wind tunnel of the Shanghai Automotive Wind Tunnel Center was built. The model includes the scoop, the distributed suction, the tangential blowing, the moving belt and the wheel belts. The simulated boundary layer profile and the pressure distribution agree well with test results. The baseline model and multiple design changes of the new Buick Excelle GT are simulated. The simulation results agree very well with test results.

CITATION: Wang, F., Yin, Z., Yan, S., Zhan, J. et al., "Validation of Aerodynamic Simulation and Wind Tunnel Test of the New Buick Excelle GT," *SAE Int. J. Passeng. Cars - Mech. Syst.* 10(1):2017, doi:10.4271/2017-01-1512.

INTRODUCTION

In today's economic environment, the consumer demand for fuel-efficient vehicles has never been higher. Vehicle aerodynamic design has a critical impact on fuel efficiency, through reducing wind resistance of the vehicle's exterior shape and reducing losses associated with requirements for cooling flow through the engine compartment. The challenge faced by vehicle manufacturers in each design stage is the urgent need for information about how to improve the design. Aerodynamic information can be costly and difficult to obtain, traditionally involving building a detailed model or prototype of the vehicle and testing the model in a wind tunnel. Usually, it also needs to take significant care to obtain reasonable aerodynamic results in a wind tunnel test [1-3]. Prototype testing for aerodynamics is usually a major contributor to vehicle development costs and design cycle time.

Aerodynamics simulation changes the vehicle development process, reducing both vehicle development costs and design cycle time. Because of its inherent advantages over physical testing methods, simulation can bring much more feedback about the design performance into each stage of development, improving the ability for designers and engineers to innovate in balancing design aesthetics with aerodynamics. Simulation provides cost savings and improvement in the final design.

The need for accuracy and timely delivery of design feedback places a high demand on aerodynamic simulation quality, because the simulation results will be used to predict the aerodynamic performance of the vehicle in a wind tunnel experiment as well as under real road driving condition. Usually, it is expected that the difference between the simulated aerodynamic drag coefficients (C_D) and the data obtained in a wind tunnel experiment need to be as small as possible, and the delta C_D of different vehicle designs also need to be well predicted by simulation. This is a major concern of many automotive aerodynamic engineers. In order to achieve high accuracy for the validation of simulation and wind tunnel test, the engineers need to have high fidelity to the simulation model, compared to the real vehicle used in the wind tunnel test, and to include the major information of the wind tunnel test environment into the simulation model, such as the suction system and the boundary layer removal system (BLRS).

In this paper, a digital floor model of the aerodynamic wind tunnel of the Shanghai Automotive Wind Tunnel Center (SAWTC) was built which included the models of the scoop, the distributed suction, the tangential blowing, the moving belt and the wheel belts. The model was first simulated without the vehicle and the suction rate of the distributed suction and the speed ratio of the tangential blowing were tuned to obtain a good match of the pressure distribution and boundary layer profile test data. A full detailed simulation model of the new Buick Excelle GT was then built and simulated in the wind tunnel floor model. The baseline vehicle configuration and a number of design iterations were simulated and the calculated drag coefficients (C_D) were compared to test results.

METHODOLOGY

Vehicle Information

The vehicle used in this study is the new Buick Excelle GT [4]. It is a compact sedan manufactured by SAIC General Motors Corporation (SAIC-GM) [5]. The new Buick Excelle GT was released to Chinese market on March 2nd, 2015. The aerodynamic development work is done by the aerodynamic team of the Pan Asia Technical Automotive Center (PATAC) [6], which is the technical center of the SAIC-GM. For validation purpose, the vehicle used in this study is a production vehicle.

Test Method

In this study, the wind tunnel test was done in the aerodynamics wind tunnel of the Shanghai Automotive Wind Tunnel Center (SAWTC). It is a closed-loop, full scale automotive wind tunnel. Its sketch is shown in Figure 1 and its parameters are listed in Table 1.

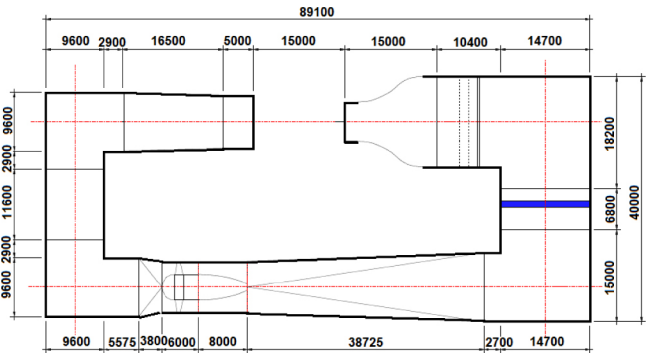


Figure 1. Sketch of the aerodynamic wind tunnel of SAWTC.

Table 1. SAWTC aerodynamic wind tunnel parameters.

Nozzle area	approx. 27m ² (6.5m(w) x 4.25m(h))
Wind speed range	25~250kph (7~70m/s)
Test section length	15m
Plenum volume	17m(w)x22m(l)x12m(h)
Layout Size	91.2m x 44.5m

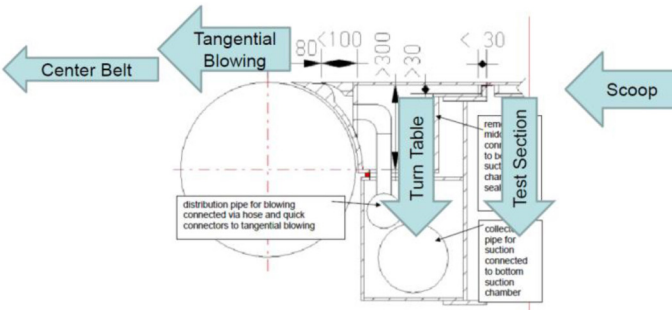


Figure 2. The floor boundary layer removal system (BLRS) of the wind tunnel.

It is common to use ground simulation techniques [7-9] to remove the boundary layer in front of the test vehicle in order to represent the flow condition of the vehicle on the road. The SAWTC aerodynamic

wind tunnel has a floor boundary layer removal system (BLRS) which includes a scoop, a distributed suction and a tangential blowing system. The sketch of the BLRS is shown in figure 2.

The wind tunnel also has a five-belt road simulation system (one center-belt and 4 mini-belts for the wheels), as shown in Figure 3 together with the vehicle. It was reported [10] that this BLRS could generate very good flow profile, compared to other automotive wind tunnels. The measured horizontal pressure distribution and the boundary layer profile presented in the report [10] are also used to calibrate the wind tunnel floor simulation model in this study.



Figure 3. The vehicle and the moving belt system of the wind tunnel.

In the wind tunnel test, the vehicle was mounted by the struts which were clamped on the rocker-panel, as shown in Figure 4. The struts were also connected to the balance to measure the aerodynamic forces. Another function of the struts was that they prevent the change of the vehicle posture when the wind is on. The vehicle was fixed by the struts so the ride height would not change during the wind tunnel test.



Figure 4. The vehicle was fixed to the wind tunnel floor by the struts clamped on the rocker-panel.

Simulation Method

Exa's commercial software PowerFLOW is used to perform transient flow simulation. This software uses a special discretization of the Lattice Boltzmann method (LBM) on a variable resolution Cartesian volumetric grid, which is automatically generated by the software. The key concept of the LBM is very briefly outlined here. For more details, please refer to literature [11-13].

Theory

The Boltzmann equation describes a representation of the particles and how the particles evolve in a fluid. The equation can be written as following,

$$\frac{\partial f}{\partial t} + \vec{v} \cdot \frac{\partial f}{\partial \vec{x}} = \Theta \quad (1)$$

where f is the velocity probability distribution function and Θ is the collision operator, which satisfies the necessary conservation laws. $\vec{v}$ is velocity vector, t is time and $\vec{x}$ is the position vector. In the lattice form, the Boltzmann equation can be expressed as a set of algebraic equations of the probability distribution function at each state, f_i ,

$$f_i(t + \Delta t, \vec{x} + \vec{e}_i \Delta t) = f_i(t, \vec{x}) + \Theta_i(t, \vec{x}) \quad (2)$$

In which $\vec{e}$ is the unit normal vector. Here the lattice BGK collision term is,

$$\Theta_i(t, \vec{x}) = -\frac{\Delta t}{\tau} [f_i(t, \vec{x}) - f_i^{eq}(t, \vec{x})] \quad (3)$$

where τ is the relaxation time and (f_i^{eq}) is the lattice equilibrium distribution function.

The macroscopic variables such as density ρ and velocity $\vec{v}$ can be expressed by the distribution function as following,

$$\rho(t, \vec{x}) = \sum_i f_i(t, \vec{x}) \quad (4)$$

$$\vec{v}(t, \vec{x}) = \frac{1}{\rho(t, \vec{x})} \sum_i f_i(t, \vec{x}) \vec{e}_i \quad (5)$$

turbulence modeling in PowerFLOW this study uses turbulence theory to model only the dissipative and inertial ranges, modeling the dynamics of sub-grid scale turbulence by two additional equations derived from an extended RNG (Renormalization Group theory) formulation, as following,

$$\rho \frac{Dk}{Dt} = \frac{1}{\sigma} \frac{\partial}{\partial x} \left[(\mu + \mu_t) \frac{\partial k}{\partial x} \right] + P - \rho \varepsilon \quad (6)$$

$$\text{And where, } \varepsilon \equiv \nu \frac{\partial v'_i}{\partial x_k} \frac{\partial v'_i}{\partial x_k} \quad (7)$$

Here κ is turbulence kinetic energy, μ is dynamic viscosity, μ_t is turbulence viscosity, P is pressure, ε is the dissipation of the turbulence kinetic energy, and ν is the kinematic viscosity. For more details, please refer to literature [15] and [16]. This approach is also known as Very Large Eddy Simulation (or VLES).

Wind Tunnel Floor Model

The simulation cases in this paper were performed in a digital wind tunnel model which has a blockage ratio of 0.1%. The geometry of the nozzle and the collector of the SAWTC aerodynamic wind tunnel were not available to the authors so they were not included in the simulation model.

Previous studies [17,18] have shown that in order to obtain good validation results between simulation and wind tunnel test, it is important to include the road simulation system into the simulation model, especially when studying the design changes on the underbody of the vehicle. In this study, we built the road simulation system of the SAWTC aerodynamic wind tunnel into the simulation model. The geometry of the floor model is shown in Figure 5.

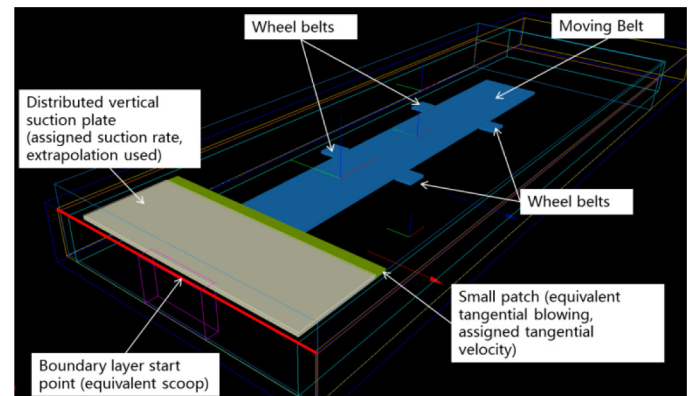


Figure 5. The floor model of the road simulation system of the wind tunnel.

The shapes, sizes and locations of the components of the road simulation system were measured in the wind tunnel and then the geometry models were built in the simulation model. In the simulation model, the scoop was modeled by setting the starting point (or location) of the boundary layer on the floor, as shown by the red line in Figure 5. The floor lying upstream of the starting point was set to be frictionless so it would not generate any boundary layer. The gray rectangle in Figure 5 was the distributed suction (DS). A vertical mass flow outlet boundary condition was set on the DS and was tuned to get a good match of the horizontal pressure distribution, which will be discussed later. At the downstream of the DS was the tangential blowing (TB), which is a green rectangle in Figure 5. Because the actual flow rate of the TB was unknown to the authors, the TB was modeled by a velocity wall boundary condition to add additional momentum to the flow near the floor. The velocity magnitude was then tuned to obtain a good agreement to the measured floor boundary layer profile. Further downstream were the center-belt and the wheel belts. In the cases when these belts were moving, they would be modeled as sliding wall boundary condition and if not, they would be modeled as standard wall with friction.

Because the exact suction rate of the DS is unknown to the authors, the first step is to adjust the suction rate to get a good match of the horizontal pressure distribution to the measured data in [10]. We simulated different DS suction rate and the results are shown in Figure 6, where the value of x axis shows the location of the measured points in flow direction, $x=0$ is the location of the center, the y axis (C_p) is the measured static pressure coefficient. In