

Identification of Vortical Structure that Drastically Worsens Aerodynamic Drag on a 2-Box Vehicle using Large-scale Simulations

Takafumi Makihara, Takahiro Kitamura, Taro Yamashita, and Kazuhiro Maeda
Toyota Motor Corporation

Chisachi Kato
University of Tokyo

Tsutomu Takayama, Kohei Yamamoto, and Yoshinobu Yamade
Mizuho Information and Research Institute

Yasumasa Suzuki
Nihon University

ABSTRACT

It is important to reduce aerodynamic drag for reducing fuel consumption. Conventionally reduction of aerodynamic drag has been carried out by shape optimization of each part of a vehicle based on the investigations of the time-averaged flows around the vehicle. However, the general tendency of drag reduction has been saturated recently and it is required to develop a new flow-control technique to achieve further reduction in aerodynamic drag. We therefore focus on the unsteadiness of the flow around a vehicle to achieve it because the aerodynamic drag of a vehicle fluctuates over time due to repetitions of generation, growth, merging and disappearance of various sizes of vortices around it. These vortices are formed by flow separations, for which the longitudinal coherent vortices inside turbulent boundary layers on vehicle surfaces are presumably playing an important role. However, there have been few studies on these vortices due to the difficulty in performing such a high-resolution flow simulation that is able to capture these small structures. In fact, the size of these vortices in the boundary layers on vehicle surfaces is estimated to be as small as 0.4 mm when the vehicle is running at 100 km/h. In the present paper, we carried out wall-resolving LES (Large Eddy Simulation) for 1/4-scale simplified 2-box car models. The Reynolds number based on the vehicle speed and the representative length of 1.06 m is set to about 0.71 million. By using K computer, which accommodates approximately 80,000 CPUs, we have realized large-scale flow computations by using approximately 2.9 billion grids. At first, we investigated appropriate grid resolution by considering the size of the longitudinal coherent vortices in the boundary layers and we have confirmed the importance to accurately resolve turbulent boundary layers in the under-floor flow of a car model. Next, we have identified critical differences in the vortical structures behind car models that have high and low aerodynamic drag. Especially, we have found that the hone-shaped vortical structures are formed behind the car model with a drastically worsened aerodynamic drag, and proposed a formation process of such vortical structures. This new finding is expected to help us achieve further reduction in aerodynamic drag of 2-box vehicles. In addition, we have found that the under-floor flow plays an important role in the above-mentioned formation process and it is essentially determined by the under-floor shape of a car model and is little affected by the flows over the roof, C pillar, or side surfaces of a car model.

CITATION: Makihara, T., Kitamura, T., Yamashita, T., Maeda, K. et al., "Identification of Vortical Structure that Drastically Worsens Aerodynamic Drag on a 2-Box Vehicle using Large-scale Simulations," *SAE Int. J. Passeng. Cars - Mech. Syst.* 9(2):2016, doi:10.4271/2016-01-1585.

INTRODUCTION

For reducing CO₂ emission and saving energy, further improvement in fuel efficiency of vehicles has become extremely important in recent years. Reduction of aerodynamic drag is considered essential to this end as aerodynamic drag accounts for about 2/3 of running resistance of a vehicle when it is travelling at around 100 km/h. Reduction of aerodynamic drag has so far been accomplished based on the investigations of the time-averaged flows around the vehicle

and shape optimization of the under floor and other parts of the vehicle. However, drag reduction seems to have been saturated in recent years. In particular, controlling the flows around 2-box vehicles is difficult. On the other hand, customer's diverse demands with regard to styling of 2-box vehicles are growing year by year. Especially, they want to have rounded vehicle shapes, which can easily worsen the aerodynamic drag. For this reason, it is necessary to

breakthrough is needed for controlling the flows around a vehicle and thus preventing the aerodynamic drag from increasing for a vehicle with rounded shapes.

In the flows around a vehicle, countless vortices are born, grow, merge and disappear repeatedly as seen in Figure 1, which results in fluctuation of aerodynamic drag over time. Separation of the flow, which influences the formation of these vortices, is greatly affected by the development of a turbulent boundary layer on the vehicle surfaces. In a turbulent boundary layer, longitudinal coherent vortices near the walls are playing a dominant role. Therefore, if the behavior of such vortices in the turbulent boundary layers can be analyzed in detail, the entire process of boundary layer separation and formation of the rear-flow structures can be identified. If this is achieved, new technology for controlling flows, applicable to rounded roof-end styling, can be developed. Few researchers have attempted to resolve the longitudinal coherent vortices in the turbulent boundary layers of a car [1, 2], but they did not give detailed discussion on the growth and separation of the boundary layer and formation of the large-scale vortical structures. In the present study, highly-accurate large-scale Large Eddy Simulation (LES) [3] was conducted by using K computer, one of the world's fastest supercomputers, to analyze the flows around car models. From these computations, firstly, computational conditions to obtain accurate flow fields have been made clear. Secondly, the vortical structures [4, 5] behind the car models were identified respectively for car models with a favorable drag coefficient (C_D) and for the one with a poor drag coefficient. Finally, a formation mechanism of vortical structures was proposed for a car with a poor drag coefficient.

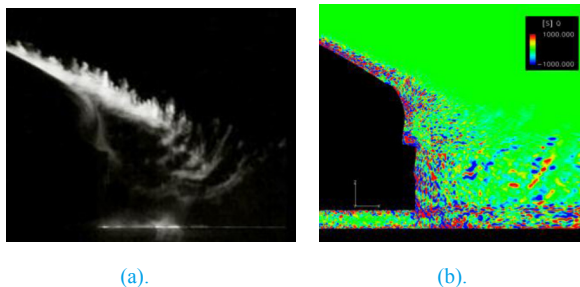


Figure 1. Visualizations of rear flow of 1/4-scale car model by (a) experiment using smoke and (b) computation using Second invariant of velocity-gradient tensor.

WIND TUNNEL TESTING

Outline of Wind Tunnel

Wind tunnel testing was performed to validate the computational accuracy in the present study. For the experiments, we used a low-speed wind tunnel (see Figure 2) at the Research Center for Aerodynamics of Nihon University. This wind tunnel is of Göttingen type. The nozzle opening has dimensions of 2 m by 2 m, and the measurement section with fixed walls has a length of 5.3 m. To avoid the effects of the nozzle boundary layers on the aerodynamics of the

car models, a ground plate with a length of 4.4 m is installed in the measurement section at a height of 0.66 m from the wind tunnel floor. The car model is placed at 1.03 m downstream of the front edge of this ground plate. The turbulence intensity when no car model is present is 0.1% or less.

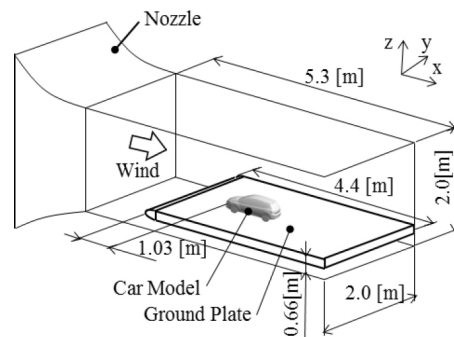


Figure 2. Wind tunnel dimensions.

Test Conditions and Flow Field Measurements

Detailed descriptions on the car model are given in the next section. Here, test conditions are outlined. The wind velocity, U , was 30 m/s, the coefficient of kinematic viscosity, ν , was $1.5 \times 10^{-5} \text{ m}^2/\text{s}$ and the Reynolds number based on a total length of the car model, L , of 1.06 m, was about 2.1×10^6 . We measured the six components of the force and moment exerted to the test model and profiles of the streamwise velocity around the model. The former was measured by connecting the model to the wind tunnel balance via tire struts. The latter was measured by installing a hot-wire probe on the wind tunnel's ancillary traverse system.

PRELIMINARY COMPUTATIONS AND VERIFICATIONS OF THEIR ACCURACY

Car Models

For the present study, 2-box car models were considered. The baseline model, Case A, shown in Figure 3, had a slant angle of the back-door glass of 47 degrees, a spoiler installed at the rear end of the roof, and upward slope of the rear floor toward the rear end. The model size was of 1/4-scale of the real car, with a length of 1.06 m, a height of 0.37 m, a width of 0.44 m, and a frontal projected area of 0.138 m². For the sake of simplicity, we did not consider the flow through the engine room and removed wipers, outer mirrors and door handles. In addition, the under-floor shape was made flat. In addition to this baseline model, computations and experiments were also performed for the following three cases for comparisons. Case B had a slant angle of the back-door glass of 30 degrees and a rounded roof end. Case C did not have the upward slope of the rear floor. Case D did not have the upward slope of the rear floor, had a slant angle of the back-door glass of 30 degrees and a rounded roof end. Figure 4 shows the rear-end shapes of these car models. All of the models shared the same front-half shape.

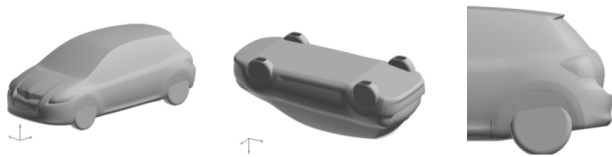


Figure 3. Baseline car model (Case A).

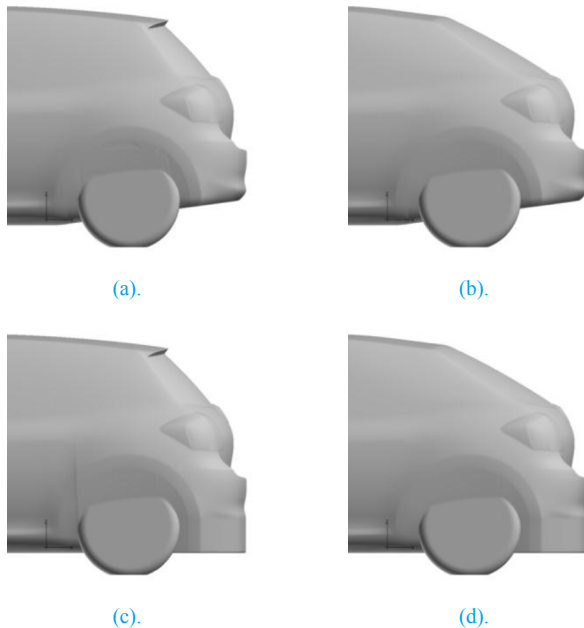


Figure 4. Rear-end shapes of each car model: (a) Case A with a slant angle of the back-door glass of 47 degrees, rear spoiler and upward slope of the rear floor, (b) Case B with a slant angle of the back-door glass of 30 degrees and upward slope of the rear floor, (c) Case C with a slant angle of the back-door glass of 47 degrees and rear spoiler, and (d) Case D with a slant angle of the back-door glass of 30 degrees.

Flow Solver

For the present study, a general-purpose incompressible flow solver, Front Flow/blue (FFB) [3] was used. FFB employs Finite Element Method (FEM) for spatial discretization and Dynamic Smagorinsky model for the subgrid-scale model of LES. It has been developed for performing wall-resolving LES for industrial flows with complicated geometries, which normally require use of a computational mesh with billions of grids. It has been successfully applied to the computations of internal flows and aeroacoustics of turbomachinery and ship hydrodynamics [see 6 for example]. Table 1 presents the main computational methods of FFB.

Table 1. Main computational methods of FFB.

Items	Method
Turbulence model	Dynamic Smagorinsky model
Spatial discretization	Finite element method
Time integration method	Crank-Nicolson method
Pressure Coupling method	Fractional-step method
Matrix solver	BiCGSTAB method

Computational Domain and Boundary Conditions

The 1/4-scale car models were set in a semi-cylindrical domain that had a radius of 2.13 m and a total length of 26.9 m, as shown in Figure 5. The origin of the coordinate was set at the front end of the car model. The inlet boundary was at $x = -3.1$ m, and the outlet boundary at $x = 23.8$ m. The car model was placed with its center line aligned to that of the computational domain. The boundary conditions were set as follows. A uniform flow of $U=30$ m/s in the +x direction was given at the inlet boundary and the moving wall was given at the lateral face of the semi-cylinder, and non-slip conditions were prescribed at the ground plate and the car surface. The outlet boundary was set such that the fluid tractions be zero. The fluid was assumed to be air at the room temperature, and the Reynolds number was set to about 2.1×10^6 , which was the same as that of the wind tunnel testing.

The present computational domain did not include the wind tunnel nozzle and the ground plate, and was expected to reproduce a flow around the car model that was placed on an infinitely large ground plate in an open air. We had confirmed by preliminary computations that the wind tunnel nozzle had little effects on the flow around a car model although the ground plate had some effects on it and produced a difference in the flow that approached to the lower part of the car model. However, after some distance downstream of the leading end of the car model, the ground plate had little influence on the flow around the car model and the velocity profiles computed by using the present computational model matched well with the measured ones. Therefore, to avoid use of unnecessary computational grids and keep a good grid quality, which is crucially important for performing an accurate LES, we used the computational domain described above.

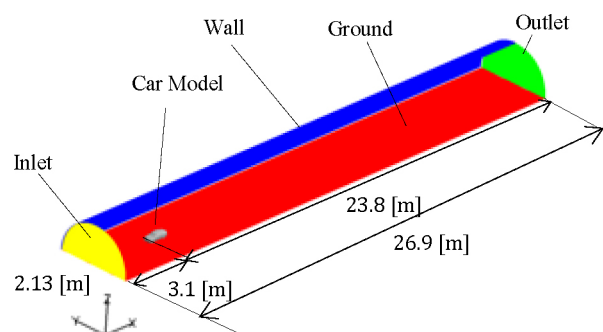


Figure 5. Computational domain.

Computational Grid

To increase the accuracy of computations based on a previous study [7], the aspect ratios of the computational grid were carefully checked and properly adjusted. The computational grid was composed of a total of 81 million elements, with a minimum dimension near the wall of 0.64 mm in the wall-normal direction and 2.4 mm in the directions along the wall. Figure 6 shows the computational grid near the rear end of the roof. Note here that this computational grid did not have a sufficient resolution for the coherent vortices in the turbulent boundary layers. We will improve the grid resolution as well as the boundary conditions, which will be described in the next section.

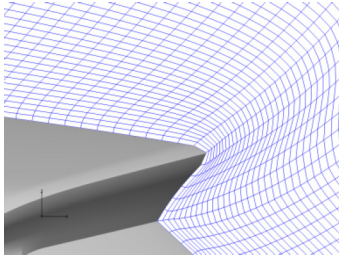


Figure 6. Computational grid near the rear end of the roof (Case A and Case C).

Comparison between Computational and Experimental Results

Table 2 presents the coefficients of aerodynamic drag (C_D) and lift (C_L) obtained by the computations and experiments for each case. The computed absolute values are different from those of measurements; especially the aerodynamic drag for Case C is considerably different. Namely, the measured drag is higher than those of Case A and B while the computed drag is lower than them.

Table 2. Computational and experimental results (preliminary computations).

Model	Computation		Experiment	
	C_D	C_L	C_D	C_L
A	0.260	-0.023	0.300	-0.156
B	0.261	0.290	0.283	0.101
C	0.240	0.110	0.309	0.326
D	0.271	0.542	0.344	0.537

Figure 7 shows distributions of total pressure C_{pt} in the plane of symmetry ($y = 0$ m) and static pressure C_p on the rear surface computed for each case. The computed flow over the roof for Case C separates at the rear end of the roof without turning to the downward direction, similar to Case A. To the contrary, the experimental result

of C_L shows a positive value for Case C, which implies that the flow over the roof turns to the downward direction at the rear end of the roof.

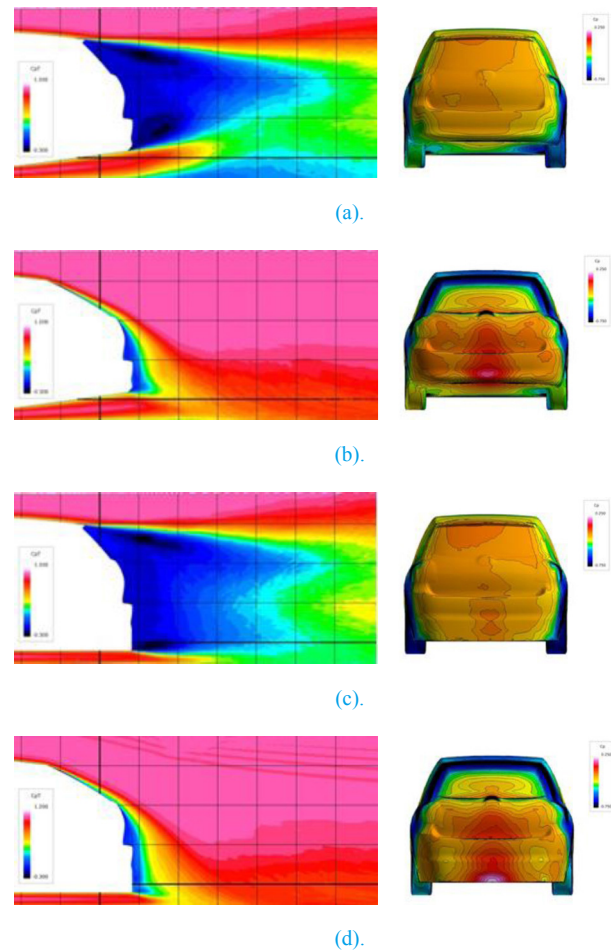


Figure 7. Distributions of total pressure C_{pt} in the plane of symmetry (left) and static pressure C_p on rear surface (right) for (a) Case A, (b) Case B, (c) Case C, and (d) Case D.

To identify the cause of this discrepancy, we have compared the computed and measured results of the flow fields in details, and found that remarkable differences existed between the computed and measured velocity profiles of the under-floor flows. As shown in Figure 8, the computations considerably overestimated the underfloor flow velocity (and underestimated the velocity defect) in all cases although they well predicted boundary layer profiles at the rear end of the roof. We have assumed that this overprediction of the under-floor flow velocity has resulted in the separation of the over-roof flow in the computation of Case C.

The reason for this discrepancy between the computational and experimental results, which occurred only for Case C, may be explained as follows. In Case A, the under-floor flow velocity was relatively high due to the upward slope at the rear floor, and the high-momentum under-floor flow was most likely to prevent the over-roof flow from turning to the downward direction at the rear end