

Flow Structures above the Trunk Deck of Sedan-Type Vehicles and Their Influence on High-Speed Vehicle Stability

2nd Report: Numerical Investigation on Simplified Vehicle Models using Large-Eddy Simulation

Takuji Nakashima
Hiroshima University

Makoto Tsubokura
Hokkaido University

Takahide Nouzawa and Takaki Nakamura
Mazda Motor Corporation

Masashi Ichimiya
Tokushima University

Copyright © 2009 SAE International

ABSTRACT

In the present study, two kinds of simplified vehicle models, which can reproduce flow structures around the two sedan-type vehicles in the previous study, are constructed for the object and the unsteady flow structures are extracted using Large-Eddy Simulation technique. The numerical results are validated in a stationary condition by comparing the results with a wind-tunnel experiment and details of steady and unsteady flow characteristics around the models, especially above the trunk deck, are investigated. In quasi- and non-stationary manner with regard to vehicle pitch motion, unsteady flow characteristics are also investigated and their relations to an aerodynamic stability are discussed.

INTRODUCTION

In the previous study [1], it was found that two kinds of sedan-type vehicles with almost the same aerodynamic lift coefficient happened to show a drastic difference of straight-ahead stability, and suggested the existence of flow structures above the trunk deck, which stabilize the vehicle vertical motion in on-road driving. In conclusion

of the study, unsteady characteristics of a flow field around the vehicle, especially above the trunk deck, have a great importance in an evaluation of vehicle's straight-ahead stability in addition to the conventional steady aerodynamic evaluation.

In recent years, importance of unsteady flow characteristics, which is expected to relate to the vehicle stability, becomes a hot topic in a vehicle aerodynamics field. Guilmineau et al. [2] performed numerical and experimental analysis of unsteady flow field behind a car model oscillating in yaw direction and they investigated unsteady aerodynamic characteristics, such as phase shift phenomena of the aerodynamic forces. Aschwanden et al. [3] studied the influence of pitch and heave motions on the vehicle dynamics using a manipulator in a wind tunnel experiment. For an investigation of such unsteady phenomena, the numerical analysis is expected to be a powerful tool avoiding some difficulties in the measurements. Regarding the numerical technique in a simulation of turbulent flow around a vehicle that is in motion, the Large-Eddy Simulation (LES) is suitable [4] because the method is based on spatial filtering of flow field and it does not need any time-average for turbulent modeling.

The objective of this study is to validate and generalize the analogy of the high-speed aerodynamic stability mechanism proposed in the previous study through the limited wind tunnel and on-road data. For this purpose, two kinds of simplified vehicle models, which represent the two sedan-type vehicles investigated in the previous study in terms of the reproduced flow structures supposed around the vehicles, are constructed for the object and three dimensional unsteady flow structures around them are extracted using Large-Eddy Simulation technique. Pitch motion of the vehicle, which was supposed to influence the running stability of the vehicle in the previous study, is set to be a target in the present study on the interaction of vehicle motion and unsteady flow structure relating to the running stability.

The numerical analysis is first conducted in a stationary condition with a pitch angle of zero degree. For a validation of the numerical analysis, the results are compared with results of a wind-tunnel measurement from viewpoints of steady and unsteady characteristics. Based on the validation, details of steady and unsteady flow characteristics around the vehicle models and especially on the trunk deck are investigated also in quasi- and non- stationary manner with regard to vehicle pitch motion. Dominant flow structures and their difference around the vehicle models generated by slight difference of partial geometry are first extracted from the time-averaged flow field. Then, their unsteady flow characteristics are discussed. Aerodynamic pitch moments acting on the vehicle oscillating in the pitch direction are also discussed with regard to their influence on the vehicle aerodynamic stability.

OBJECTIVE VEHICLE MODELS

SIMPLIFIED VEHICLE MODELS WITH DIFFERENT AERODYNAMIC STABILITIES - In order to investigate flow structures around the sedan type passenger cars in the previous study [1], especially above their trunk deck, the target simplified vehicle models have been designed. They are about 1/20 scale of the original passenger sedan and the body size is 210mm length (L), 80mm width (W), and 65mm height (H) respectively. Their main differences are appearing in the curvatures of front and rear pillars as shown in Fig. 1. In the previous study, the original sedan-type vehicles of these vehicle models have different straight-ahead stabilities at high speed. The vehicle model, which has a lower stability, is shown in a top figure of Fig. 1 and it is called as "Model A" in this paper. The other model, which has higher stability, is shown in a bottom of Fig. 1 and is called as "Model B".

WIND TUNNEL EXPERIMENT OF THE SIMPLIFIED MODELS [5] - experimental measurements of flows around the simplified vehicle models were conducted by Ichimiya *et al.* [5] in a wind tunnel facility in Tokushima University. Figure 2 shows a photograph of experimental setup. The vehicle model was supported by four struts located at positions of front and rear wheels. The rear struts had been designed to move in vertical direction

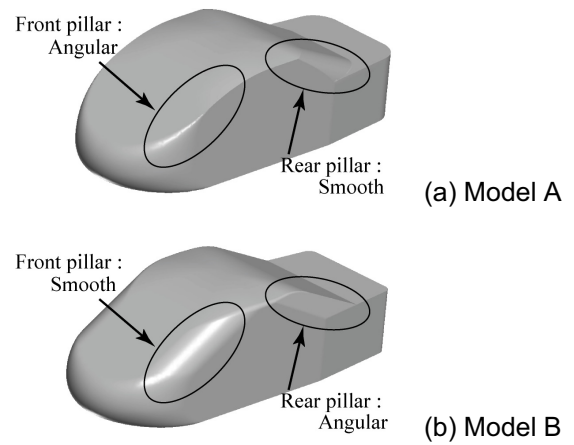


Fig.1 Schematic view of the simplified vehicle models



Fig.2 Photograph of the experimental setup.

and to push the simplified vehicle model for a forced oscillation in the pitch motion. In the wind tunnel experiment, velocity field around the trunk deck of the vehicle model was measured by hot-wire anemometry. Static pressure at points on the trunk deck was also measured by a pressure sensor. Measured flow field and its unsteady characteristics around the simplified geometries showed the same tendency as the real sedan-type vehicles investigated in the previous study.

NUMERICAL METHOD

GOVERNING EQUATIONS - The governing equations adopted in the present LES method for a vehicle aerodynamics prediction are the spatially filtered continuity and Navier-Stokes equations as follows;

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \quad (1)$$

$$\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial}{\partial x_j} \bar{u}_i \bar{u}_j = -\frac{\partial \bar{P}}{\partial x_i} + 2 \frac{\partial}{\partial x_j} (\nu + \nu_{SGS}) \bar{S}_{ij} \quad (2)$$

$$\bar{S}_{ij} = \frac{1}{2} \left(\frac{\partial \bar{u}_j}{\partial x_i} + \frac{\partial \bar{u}_i}{\partial x_j} \right) \quad (3)$$

$$\bar{P} = \bar{p} / \rho + 1/3 (\overline{u_i u_i} - \bar{u}_i \bar{u}_i). \quad (4)$$

Here, u_i , p , ρ , and ν represent flow velocity in i -direction, pressure, density, and dynamic viscosity of the fluid, respectively. An over-bar ($\bar{}$) means the spatial filtered value of the physical value. Sub-Grid Scale(SGS) eddy viscosity ν_{SGS} appeared in Eq. (2) must be modeled by a turbulent model. The standard Smagorinsky model [6] is adopted to estimate ν_{SGS} as follows;

$$\nu_{SGS} = (C_s f_s \Delta)^2 \sqrt{2 \bar{S}_{ij} \bar{S}_{ij}} \quad (5)$$

Here, Δ means the spatial filter width that is evaluated by a cubic root of cell volume. C_s is the Smagorinsky constant and is given as 0.15, which is the standard value for external flow simulations. To represent a damping of SGS eddy viscosity in near-wall region, the Van-Driest type damping function f_d in Eq. (5) is introduced as follows;

$$f_d = 1 - \exp(-y^+ / 26) \quad (6)$$

where y^+ is the wall unit.

COMPUTATIONAL CODE AND DISCRETIZATION SCHEMES - To conduct the present numerical simulations, the CFD code "FrontFlow/red ver. 2.8" has been modified and optimized for an unsteady vehicle aerodynamic simulation. The code "FrontFlow/red" had been developed at the Institute of Industrial Science of the University of Tokyo, as parts of the project of Frontier Simulation Software for Industrial Science [7] and the project of Revolutionary Simulation Software [8].

These governing equations are discretized based on the vertex-centered unstructured finite volume method and SMAC algorithm. The central finite difference scheme with the second order accuracy is adopted for the spatial discretization except for the convective term in which 5% of the first order upwind component is blended for a numerical stability. The second order Adams-Bashforth scheme is adopted for the time integration.

NUMERICAL CONDITIONS - In the present numerical simulations, the simplified vehicle model is set into the rectangular duct as shown in Fig. 3 (to the left). Origin and directions of coordinates are defined as shown in Fig. 4. They are based on the experimental geometry though the struts are neglected in the present simulations. The origin is defined at the central position of the two front struts in the experiment. Clearance under model's floor is 0.23H as same as the experiment. At an inlet boundary, uniform and constant flow velocity $U_0=16.7\text{m/s}$ is assumed. Reynolds number, based on a vehicle length and an inlet velocity, is about 2.3×10^5 .

Regarding the boundary conditions, the vehicle body and a part of floor, which is light-colored in the left of Fig. 3, are treated by the wall-model assuming a fully developed turbulent boundary layer. The dusk floor, which is treated as free-slip wall boundary, represents a suction of a boundary layer developed on the floor. The vehicle model is located 0.14L downstream from this suction. The side and top walls are also treated as free-slip boundary.

The computational domain is divided to about seven million elements and 1.5 million nodes in each case. The computational grids on the vehicle model surfaces are shown in Fig. 5 and the grids on the central cut-plane of the vehicle model B is also shown in Fig. 6. The minimum distance between the nodes on the surface is 0.5mm on the edges. Eight layers of prism mesh are inserted on the vehicle body and thickness of the first layer is about 0.1 mm. The other region is divided into tetrahedral meshes except for the prism-layer on the outlet boundary.

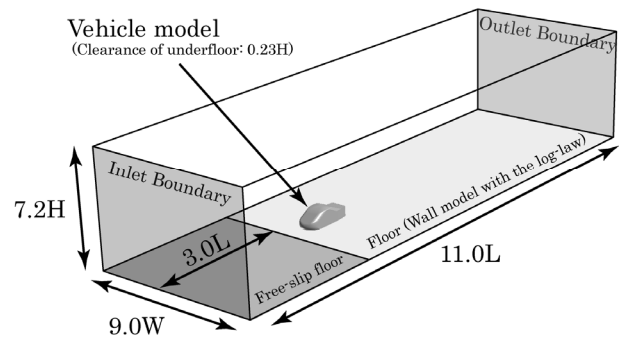


Fig.3 Schematic view of a computational domain

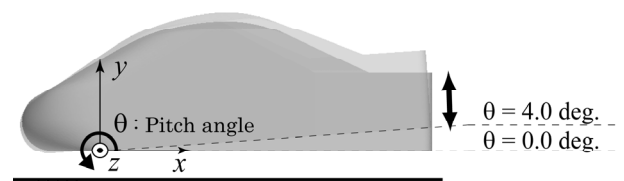


Fig.4 Definition of coordinates and pitch motion.

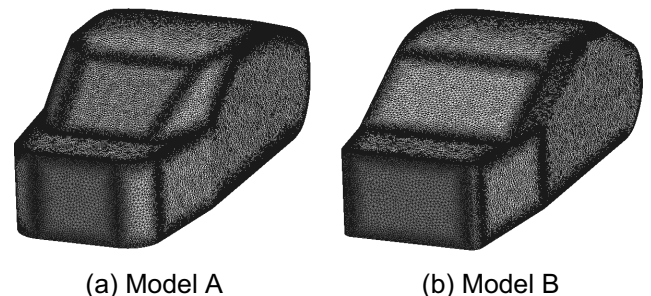


Fig.5 Computational grids on the vehicle model surfaces

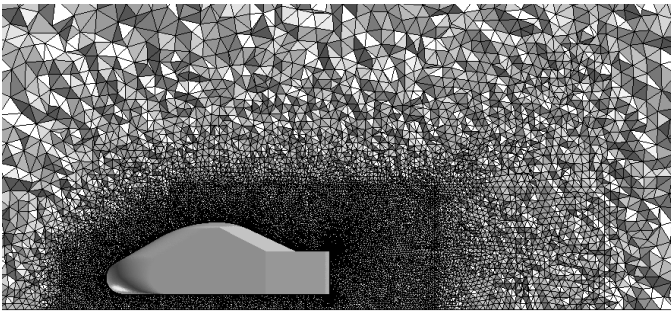


Fig. 6 Computational grids on a cut-plane at the center of the vehicle model B.

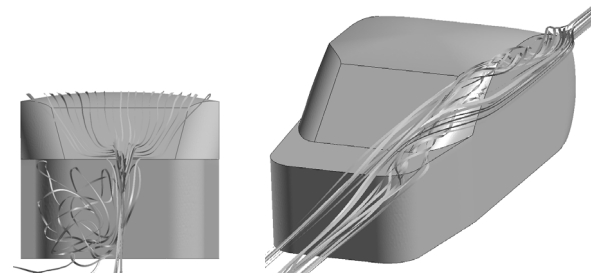
LARGE-EDDY SIMULATION ON SIMPLIFIED VEHICLE MODELS

SIMULATION IN A STATIONARY CONDITION - First, the LES of flows around the vehicle models in a stationary condition is conducted. Pitch angle is set to be zero and the steady and unsteady flow characteristics around the vehicle models are investigated.

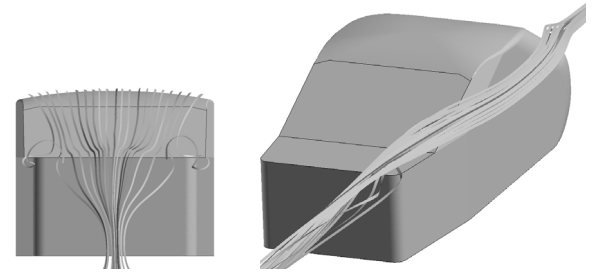
Figure 7 shows time-averaged flow structures around the vehicle models visualized by flow ribbons. In both cases, the angular pillars generate clear longitudinal vortices. Thus, in the model A with an angular front pillar, the front pillar vortex is generated and goes onto the rear shield. The vortex is paired with the other vortex generated from the rear pillar. On the other hand, in the model B, the approaching flow goes around the smooth front pillar and the angular rear pillar generates a clear longitudinal vortex on the side of the trunk deck. These features of the pillar vortices are also clearly shown in Fig.8, which visualize the time-averaged vortex structures as iso-surfaces of vorticity magnitude $\bar{\omega} = 1.5 \times 10^3 [1/s]$. and their rotational directions as arrows. The front pillar vortices are observed only in the model A having angular front pillar.

These flow structures characterize flow field above the trunk deck of the vehicle models. Figure 9 shows streamwise velocity distributions on vertical cut-plane at 5mm downstream from a trailing edge of the trunk deck. Different flow features are observed in the velocity distributions and the model A shows a non-uniform distribution of the velocity in the spanwise direction on the trunk deck, while the model B generates a almost uniform distribution in the direction.

In order to validate the present Large-Eddy Simulations to represent steady flow structures, these distributions are compared with the experimental data. The streamwise velocity profiles on five vertical lines shown in Fig.9 are compared in Fig. 10. At $z=0\text{mm}$ above the center of the trunk deck, deceleration of streamwise velocity in the model A is stronger than in the model B and this tendency agrees with the experimental data,

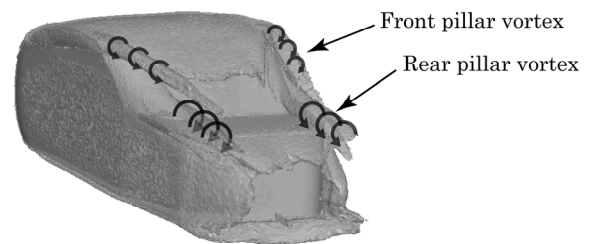


(a) Model A

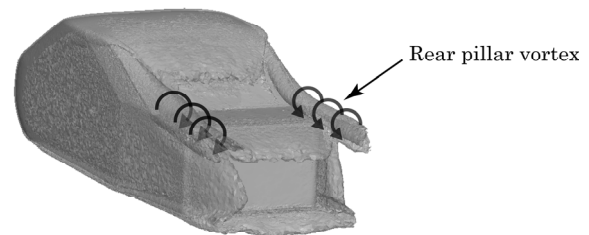


(b) Model B

Fig. 7 Time-averaged streamlines



(a) Model A



(b) Model B

Fig. 8 Iso-surfaces of time-averaged vorticity magnitude $\bar{\omega} = 1.5 \times 10^3 [1/s]$. Arrows indicate rotate direction of vortex.

though the deceleration of the velocity in the model A is quantitatively overestimated in the simulation. The front and rear pillars vortices appear as slow velocity regions in the profiles on the lines $z=20\text{mm}$ and 30mm . Such profiles qualitatively agree with the experimental data, though the vertical positions of the vortices are closer to the deck than the experimental data.