

# Unsteady Vehicle Aerodynamics during a Dynamic Steering Action: 2nd Report, Numerical Analysis

Makoto Tsubokura and Yuki Ikawa  
Hokkaido Univ.

Takuji Nakashima  
Hiroshima Univ.

Yoshihiro Okada, Takashi Kamioka and Takahide Nouzawa  
Mazda Motor Corp.

## ABSTRACT

Unsteady aerodynamic forces acting on vehicles during a dynamic steering action were investigated by numerical simulation, with a special focus on the vehicles' yaw and lateral motions. Two sedan-type vehicles with slightly different geometries at the front pillar, side skirt, under cover, and around the front wheel were adopted for comparison. In the first report, surface pressure on the body and total pressure behind the front wheel were measured in an on-road experiment. Then the relationships between the vehicles' lateral dynamic motion and unsteady aerodynamic characteristics during cornering motions were discussed. In this second report, the vehicles' meandering motions observed in on-road measurements were modeled numerically, and sinusoidal motions of lateral, yaw, and slip angles were imposed. The responding yaw moment was phase averaged, and its phase shift against the imposed slip angle was measured to assess the aerodynamic damping. It was found that the vehicle model with a higher sensory rating of high speed stability in the on-road test showed higher aerodynamic damping for the slip-angle change. The difference in flow structures on the sides of the vehicles and their contributions to the yaw and lateral motions were also investigated by visualizing the transient numerical results. Observations revealed that the vehicle with higher stability had thinner side flow structures, and flows around the front wheel house and rear-end corner contributed to restraining the rotating motion during the meandering motion. As a result, the discussions and analogy included in the previous report based on the restricted on-road measurements were confirmed in the present numerical analysis.

**CITATION:** Tsubokura, M., Ikawa, Y., Nakashima, T., Okada, Y. et al., "Unsteady Vehicle Aerodynamics during a Dynamic Steering Action: 2nd Report, Numerical Analysis," *SAE Int. J. Passeng. Cars - Mech. Syst.* 5(1):2012, doi: 10.4271/2012-01-0448.

## INTRODUCTION

In the automotive industry, the aerodynamic performance of road vehicles under development is mainly determined through the averaged aerodynamic forces and moments measured in wind-tunnel facilities. This is also true for the vehicle's high-speed stability, even though it is also strongly related to the vehicle's dynamic pitching or yawing motion. Recently, we reported that two sedan-type vehicles with almost the same drag and lift coefficients showed drastic differences in pitching stability in on-road tests and that flow structures above the trunk deck affect vehicle pitching motion [1]. Subsequently, simplified vehicle models were developed that reflect the geometric differences in the upper bodies of

these two vehicles, and a method to assess the aerodynamic damping factor was proposed by imposing forced oscillation on the body [2][3]. To study the vehicles' aerodynamic stability during dynamic motion, we conducted a dynamic coupling analysis of flow around the vehicles and their dynamic motion by using large-eddy simulation (LES) [4]. We further investigated the aerodynamic pitching stability using the developed numerical method, and we explained the mechanism of the aerodynamic damping [5][6]. To investigate the effect of a vehicle's pitching/heaving motion on the aerodynamic performance, Aschwanden et al. [7] developed the model motion system called "shaker" for a scale wind-tunnel model and discussed the greater effects of

dynamic motion than those suggested by the static measurements. They further investigated the unsteady or transient aerodynamic effects by utilizing the transient wind-tunnel data obtained by the “shaker” in vehicle dynamics simulations [8] and suggested their importance on the stability and handling characteristics of vehicles.

It is widely acknowledged that a vehicle's straight-ahead stability, especially in the context of its drivability, is strongly related to its yaw stability. This matter has mainly been discussed from the viewpoint of crosswind sensitivity or the on-road turbulence effect, and many research results have been reported. Some of these studies address on-road measurements of turbulence [9][10], wind-tunnel improvement for transient yaw behavior by incoming turbulence [11], or dynamic motion systems [12][13], numerical simulation [14], and reviews of these topics [15][16]. In contrast to crosswind stability, less research has been conducted on vehicle stability during dynamic steering actions. The main reason for the lack of academic knowledge on this topic is the difficulty in treating the coupled yaw and lateral motions. Thus, little is known about the relationships between the vehicle's dynamic motion during steering and unsteady aerodynamics around the vehicle body.

Accordingly, the objective of this study is to clarify the unsteady aerodynamic characteristics of vehicles during steering actions and their effect on the vehicle's motion stability. For this purpose we focus on two sedan-type vehicles with slightly different geometrical shapes in the front pillar, side skirt, under cover, and around the front wheel. In steering action tests conducted at high speed at Mazda proving ground, different sensory ratings were given by test drivers. In the first report [17], pressure distributions on the sides of vehicles and total pressure distributions behind the front wheel house were measured during high speed driving with a sinusoidal steering input. Then the relationships between the vehicle's dynamic motion and unsteady forces on the side of the body were analyzed. Differences in the unsteady side force and the yaw moment estimated from the side pressure were discussed, and flow structures causing these differences were suggested. In this second report, we have adopted the coupling numerical simulation method of flow and vehicle motion to further investigate the unsteady aerodynamics during the sinusoidal steering action to confirm the discussion and analogy held in the previous report. The vehicle's meandering motion is modeled based on the on-road experiments described in the first report, and is directly reproduced in the numerical flow simulation. We have also estimated the aerodynamic damping of the vehicle's slip angle motion, and we discuss here the difference in aerodynamic damping in the context of the flow structures between the two vehicles.

## NUMERICAL METHODS

### VEHICLE MODELS

Two mid-size sedan-type models were investigated in the present study, as shown in Figs. 1 and 2. The model called “Vehicle-A” is the original model, and “Modified Vehicle-A” was developed from Vehicle-A to improve its straight-ahead stability at high speed. The main areas of improvement are: (1) a larger curvature on the front pillar, (2) a shorter gap between the front wheel arch and tire, as shown in Fig. 2, (3) the addition of a side skirt, (4) a flat engine under cover, and (5) a center floor cover. An on-road test was conducted, and we confirmed that these improvements were effective based on a higher sensory rating of high speed stability at 120 km/h and 180 km/h during steering actions that were achieved by Modified Vehicle-A. The shape of the models for the present simulation was simplified slightly from the original vehicles, and no engine compartment was considered.

Figure 3 defines the forces and moments with coordinate axes fixed on the vehicle. The vehicles' length, width, and height are given as  $L = 4.7\text{m}$ ,  $W = 1.8\text{m}$ , and  $H = 1.4\text{m}$ .

### GOVERNING EQUATIONS

The governing equations adopted in this study are given by the spatially filtered continuity and Navier-Stokes equations for incompressible Newtonian flow, which are expressed in tensor notation as

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

$$\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial \bar{u}_i \bar{u}_j}{\partial x_j} = -\frac{\partial \bar{P}}{\partial x_i} + 2 \frac{\partial}{\partial x_j} (\nu + \nu_{sgs}) \bar{S}_{ij} = 0, \quad (2)$$

where the over-bar denotes the spatially filtered quantity, and  $u_i$ ,  $\nu$ , and  $\rho$  are the  $i$ -th velocity component, kinetic viscosity, and fluid density, respectively. The strain rate tensor and the filtered pressure in Eq. (2) are

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

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

The effect of subgrid-scale (SGS) turbulence on the grid-scale turbulence motion is represented by the SGS eddy viscosity, which is modeled by Smagorinsky [18] as

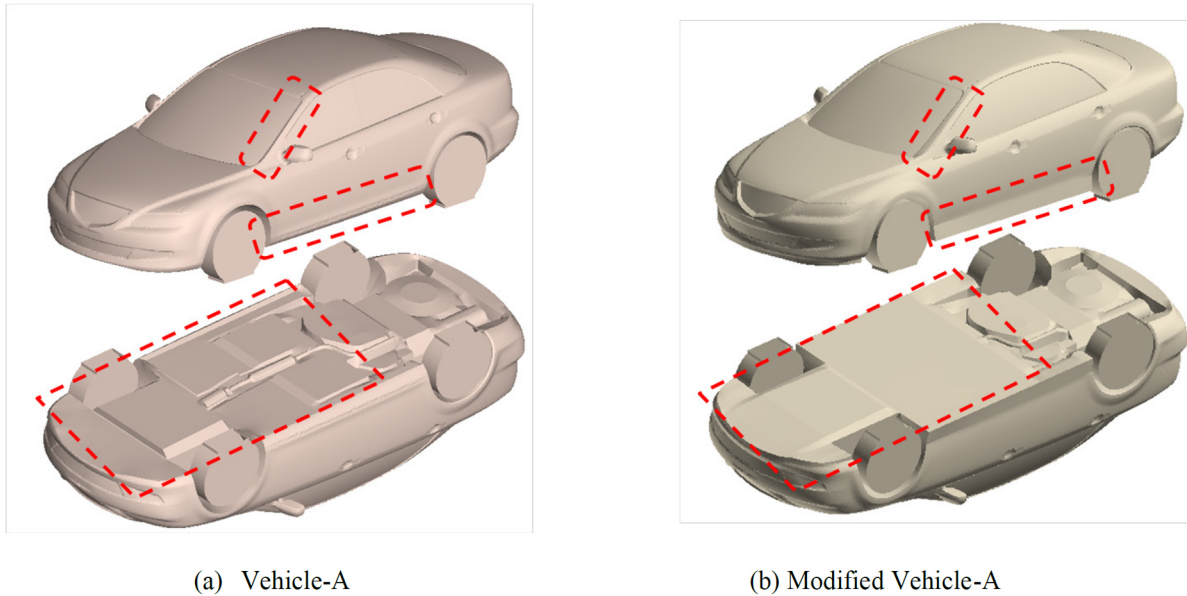


**Figure 1. Target vehicle models and the differences in their front pillar, side skirt, and underbody configurations.**

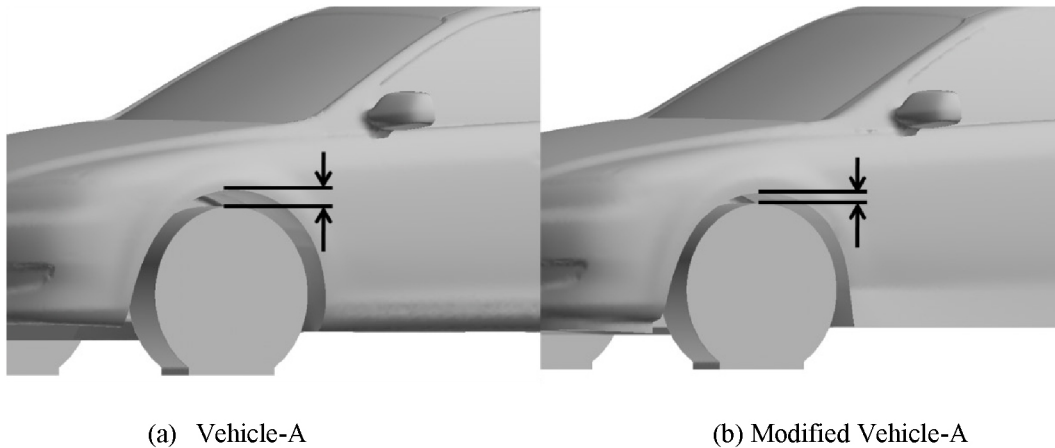


**Figure 2. Target vehicle models and the difference in front wheel well.**

$$\nu_{sgs} = \{C_s g(l^+) \Delta^{1/3}\}^2 \sqrt{2\bar{S}_{ij}\bar{S}_{ij}}, \quad (5)$$

where  $\Delta$  is the volume of the generic numerical element. The model coefficient  $C_s$  is given as 0.15, which is typically used in simulations of flow around a rectangular cylinder. The eddy viscosity is damped in the vicinity of a solid wall using a Van-Driest function:

$$g(l^+) = 1 - \exp\left(\frac{-l^+}{25}\right), \quad (6)$$

where  $l^+$  is the distance from the wall in wall coordinates.

## DISCRETIZATION

The governing equations were discretized in space using a vertex-centered unstructured finite volume method. The second-order central differences were applied for the spatial derivative, blended with the first-order upwind scheme for the convective term in the Navier-Stokes equations to avoid excessive numerical oscillation appearing at coarse tetrahedral elements. It should be noted that the dissipation property of upwind schemes is desirable to a certain extent for engineering applications of the LES on unstructured meshes. As a compromise, the contribution of the upwind discretization to the convective fluxes was set to be as low as 5%. The diffusive fluxes on the volume surface were treated based on the deferred correction formula suggested by Muzafarjia [19] to avoid a checkerboard type oscillation. The monotone upwind scheme for conservation laws (MUSCL)

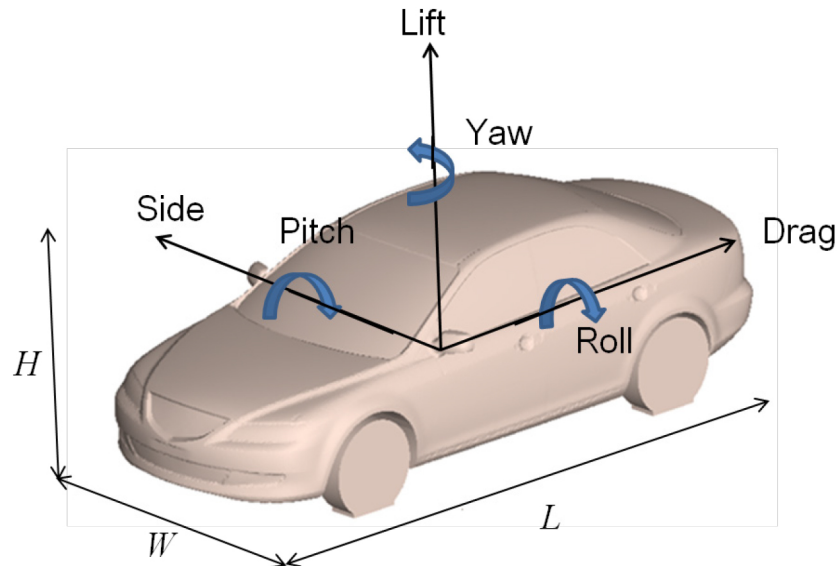


**Figure 3. Definition of forces and moments.**

by van Leer [20] was also adopted for the convective term at the region away from the vehicle where upwind dissipation does not affect the wake or near-wall turbulence of the target vehicle.

The time marching was based on the fractional step method by Kim and Moin [21], in which the third-order Adams-Moulton scheme or the implicit Euler scheme was adopted for the velocity prediction step. The coupling of the velocity and pressure fields to obtain the pressure and to correct the velocity was based on the simplified marker and cell (SMAC) method by Amsden and Harlow [22]. The flow rate on the control-volume surface was estimated following the method proposed by Rhie and Chow [23] to keep the checker-board type pressure oscillation off. The pressure Poisson equation was solved using the incomplete Cholesky conjugate gradient (ICCG) method.

## MOVING BOUNDARY METHODS

We used two techniques to recreate the vehicle's meandering body motion during the flow simulation. One was the arbitrary Lagrangean-Eulerian (ALE) method [24]. In this method, numerical grids move arbitrarily to capture a vehicle's yaw motion without changing the nodes' topological connectivity. On the contrary, the vehicle's lateral motion is relatively large compared with the yaw motion, and is difficult to trace by deforming the original numerical elements. Thus, we adopted the Navier-Stokes equations in non-inertial frame reference as the second method. In the fluid simulation, the coordinate for calculating the fluid motion is fixed on the gravitational center of the vehicle with its axis parallel to the coordinate fixed on the ground during the simulation. The lateral acceleration of the vehicle is reproduced by the external force in the Navier-Stokes equations for the lateral direction, and corresponding lateral

velocity is given as crosswind acting on the vehicle. Accordingly, we need blow and suction boundary conditions on the side walls of the numerical domain.

## NUMERICAL CONDITIONS

Figure 4 shows the computational domain and grid allocation on the floor of the numerical box. A constant velocity of 33.3m/s is imposed at the inlet and floor, while blow and suction boundary conditions with variable fluid velocity corresponding to the vehicle's lateral motion are given on the sides of the domain. Simple gradient-free and slip boundary conditions are imposed at the outlet and ceiling of the domain, respectively. The Reynolds number of the flow with respect to the vehicle length and the incoming flow is  $1.04 \times 10^7$ . At this higher Reynolds number, it is impossible to allocate a sufficiently fine grid to resolve the boundary layer on the surface of the vehicle. Thereby, a universal velocity profile of the fully developed turbulent boundary layer is assumed on the vehicle's surface. In the assumed velocity boundary, the fluid velocity at the first nearest grid point from the wall is supposed to fit the universal logarithmic law of the mean velocity, and the surface friction  $T_w$  is estimated. Then the obtained surface friction is imposed as a Neumann-type boundary condition of the velocity on the first nearest control-volume surface.

Figure 5 illustrates a numerical grid of triangle shapes allocated on the vehicle surface with the sizes of triangles ranging from 5 to 25 mm. Three prism layers were inserted on the surface of the vehicle body to reproduce the rapid velocity change around this region and to obtain surface friction with higher accuracy. The other spatial domain was divided into tetrahedral elements, with a total of 18 million elements utilized.

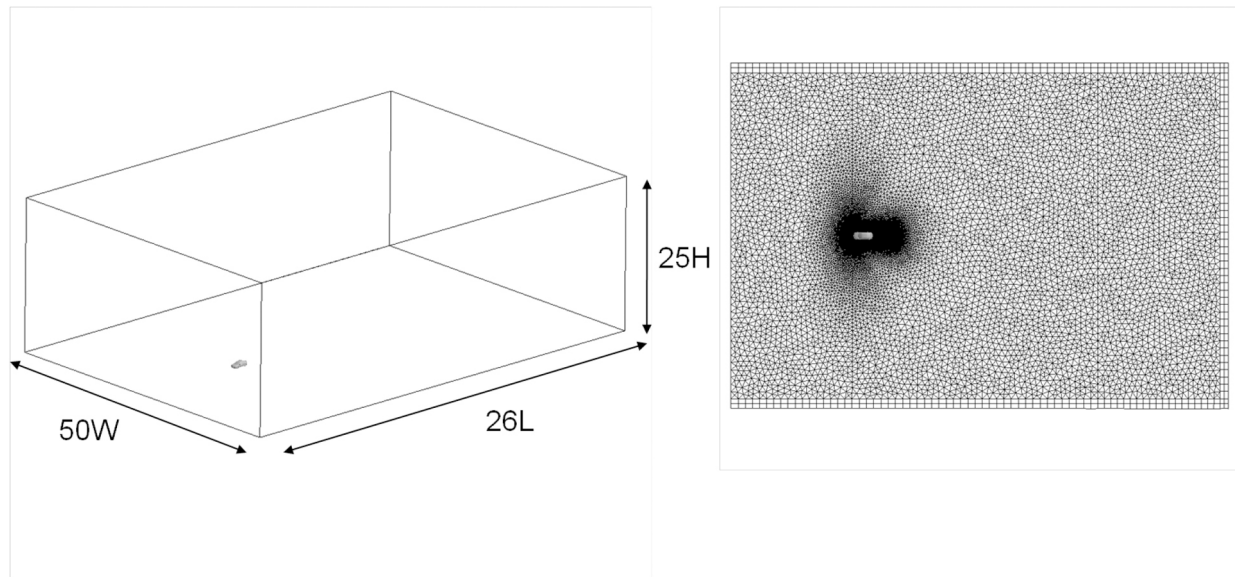


**Figure 4. Computational domain (left) and grid allocation on the ground (right).**

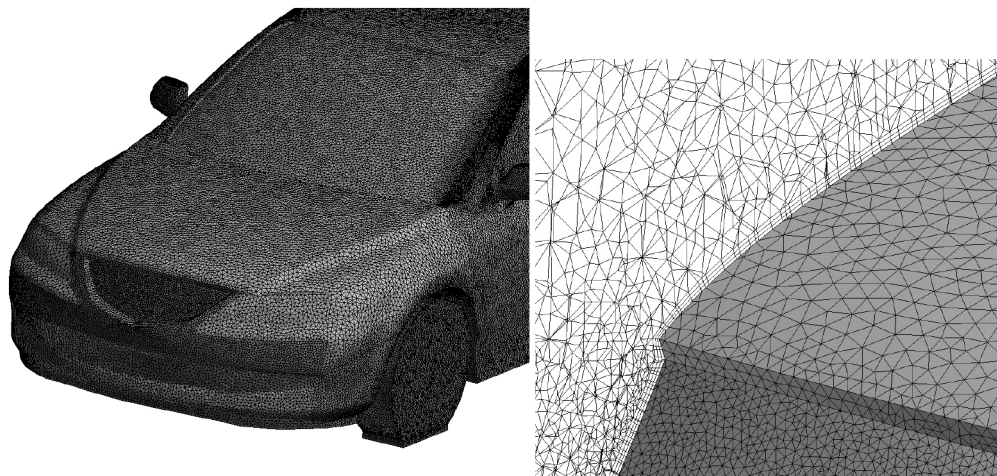


**Figure 5. Grid resolution on vehicle body (left) and near-wall prism-layer (right).**

## SOFTWARE AND HARDWARE

The computational code adopted in the present study was originally developed in the “Frontier Simulation Software for Industrial Science” project [25]. The project started in 2002 as an IT research program sponsored by the Ministry of Education, Culture, Sport, Science and Technology of Japan; and was successfully concluded in 2005. The authors intensively optimized the code for LES applications under the subsequent IT project “Revolutionary Simulation Software (RSS21)” [26], and parallel and vector efficiency as high as 96% and 99%, respectively, were achieved in simulations performed on 100 nodes/800 CPU. This allowed completion of LES calculations of flow around a formula car with complex geometry in about 120 hours [27]. In the simulation, the computational meshes that were adopted consisted of up

to 120 million cells with a typical memory usage of more than 500 GB, including data storage for the calculation of time-averaging turbulence statistics. The code was then optimized for unsteady vehicle aerodynamics simulations [4] under a project supported by an Industrial Technology Research Grant Program in 2007 from the New Energy and Industrial Technology Development Organization (NEDO) of Japan. We are now improving this code for the very large-scale simulation of up to some billion numerical elements to be executed on “K,” the world’s fastest supercomputer, in Japan [28].

The present simulation was conducted on the SR11000 supercomputer developed by Hitachi. A total of eight nodes/128 CPU was used. It took about 10 hours in wall-clock-time to simulate 1 sec of vehicle motion and the flow

**Table 1. Comparison of  $C_D$  and  $C_L$  between wind-tunnel measurements and present numerical simulation.**

|                          | $C_D$       |                      | $C_L$       |                      |
|--------------------------|-------------|----------------------|-------------|----------------------|
|                          | Wind tunnel | Numerical simulation | Wind tunnel | Numerical simulation |
| Vehicle-A                | 0.32        | 0.323                | 0.20        | 0.207                |
| Modified Vehicle-A       | 0.30        | 0.301                | 0.16        | 0.142                |
| $\Delta C_D, \Delta C_L$ | -0.02       | -0.022               | -0.04       | -0.065               |

around it, and a total of about 400 hours was required to complete one meandering simulation.

## VALIDATION ON STEADY-STATE ANALYSIS

The numerical method was validated in the steady state under the condition that the vehicles were fixed on the floor without any motion. Table 1 summarizes the drag and lift coefficients measured in the Mazda full-scale wind tunnel and obtained by the present numerical simulation. It should be noted that comparison of the absolute values of  $C_D$  and  $C_L$  between the wind tunnel data and numerical simulation is only meant to be a reference, because the numerical models shown in Fig. 1 are simplified from the real vehicle in the wind tunnel and are not exactly the same. On the other hand, the geometrical differences between these two models are relatively well reproduced in the numerical model. In that context, validation of the simulation was carried out based on the improvement of drag  $\Delta C_D$  and lift  $\Delta C_L$  observed in the wind tunnel. In the wind tunnel, the drag was improved by  $\Delta C_D = -0.02$  in Modified Vehicle-A compared with Vehicle-A, and our numerical simulation of  $\Delta C_D = -0.022$  shows excellent agreement with that. Whereas a slightly larger discrepancy is identified in the improvement of lift  $\Delta C_L$  ( $-0.04$  by the wind tunnel and  $-0.065$  by the numerical simulation), it is still acceptable because the qualitative improvement is well reproduced by the numerical simulation.

Figure 6 compares the total pressure distribution behind the side mirror observed in the wind tunnel with that predicted by the present simulation. The typical flow structures around this region are the interaction of two eddies generated by the A-pillar and the side mirror, which are observed on the planes at  $x = 2.26 \sim 2.86\text{m}$  in the figure. The simulation results show good agreement with the wind-tunnel measurements. Figure 7 shows wake structures extracted from the simulation results (left) by expressing the streamlines and velocity magnitude above the trunk deck, and their comparison with the hypothetical image suggested by the wind-tunnel visualization [1]. The important flow structures above the trunk deck are the two couples of eddies coming from the A and C pillars. In our previous studies on pitching stability [1][2][3][5][6], we found that the interactions of these two kinds of eddies profoundly contribute to the aerodynamic damping mechanisms of a vehicle's pitching motion. We can observe that our present

simulation generates the expected flow structures above the vehicle. We have also compared other flow structures such as beside or under the body obtained in our simulation with those visualized in the wind-tunnel measurements and confirmed that dominant flow features were properly reproduced in the simulation (figures are not shown here). These results clearly indicate that our simulation properly reproduced the difference in flow structures between Vehicle-A and Modified Vehicle-A generated by the slight geometrical differences shown in Figs. 1 and 2.

It should be noted here that, even though our simulation resolution was not high enough to reproduce the fine turbulence structures in the boundary layer on the vehicle surface as mentioned in the previous section, we understand that such fine-scale eddies and transition to turbulence have only a minor effect on the unsteady aerodynamics during the dynamic vehicle motion. More important than the fine-scale eddies is the reproduction of the dominant and larger trailing vortex structures such as those shown in Fig. 7 that characterize the unsteady aerodynamics of vehicles. Based on these simulation results and an intensive comparison of them with the full-scale wind-tunnel measurements, we can say that our simulation is sufficiently correct to discuss the vehicle's aerodynamic stability during the steering action.

## UNSTEADY MEANDERING MOTION ANALYSIS

The meandering motion of vehicles considered in the present study is modeled based on the on-road measurements conducted at Mazda's Miyoshi Proving Ground [17]. The definition of the yaw and slip angles, and the two coordinates used in the study are given in Fig. 8. The yaw angle is defined on the coordinate fixed on the ground, whereas the relative slip angle is given as the difference between the vehicle's direction of motion and its center line and is thus defined on the coordinate fixed on the vehicle.

We assume that the vehicles travel at a constant velocity of 33.3 m/s while their transverse displacement changes sinusoidally. Then the gravity-center motion of the vehicles is expressed as follows:

$$X = -V_x t, \quad (7)$$