

Coupled 6DoF Motion and Aerodynamic Crosswind Simulation Incorporating Driver Model

Kosuke Nakasato
 Nissan Motor Co., Ltd.

Makoto Tsubokura and Jun Ikeda
 Kobe University

Keiji Onishi
 RIKEN Advanced Institute

Shoya Ota, Hiroki Takase, Kei Akasaka, Hisashi Ihara, Munehiko Oshima, and Toshihiro Araki
 Nissan Motor Co., Ltd.

ABSTRACT

Because of rising demands to improve aerodynamic performance owing to its impact on vehicle dynamics, efforts were previously made to reduce aerodynamic lift and yawing moment based on steady-state measurements of aerodynamic forces. In recent years, increased research on dynamic aerodynamics has partially explained the impact of aerodynamic forces on vehicle dynamics. However, it is difficult to measure aerodynamic forces while a vehicle is in motion, and also analyzing the effect on vehicle dynamics requires measurement of vehicle behavior, amount of steering and other quantities noiselessly, as well as an explanation of the mutual influence with aerodynamic forces. Consequently, the related phenomena occurring in the real world are still not fully understood. Reproducing vehicle motion in the real world requires the ability to recreate disturbances from the driving environment, including natural wind and road surface undulations, and to predict the dynamic aerodynamics resulting from the driver's operations such as steering, braking and other actions. This study, therefore, focused on a crosswind situation which involves relatively large changes in vehicle behavior induced by air pressure inputs to the vehicle. A two-way coupled numerical simulation method was developed for performing aerodynamic calculations using a vehicle motion model with six degrees of freedom and incorporating a driver model for keeping the vehicle in its driving lane.

The results obtained with this calculation system revealed that aerodynamic parts reduced the amount of corrective steering needed, which coincided with the experimental data. It led to the visualization of the aerodynamic phenomena assumed to be the causal factors.

CITATION: Nakasato, K., Tsubokura, M., Ikeda, J., Onishi, K. et al., "Coupled 6DoF Motion and Aerodynamic Crosswind Simulation Incorporating Driver Model," *SAE Int. J. Passeng. Cars - Mech. Syst.* 10(2):2017, doi:10.4271/2017-01-1525.

INTRODUCTION

In developing the aerodynamic performance of passenger vehicles, automakers have adopted measures to reduce aerodynamic drag mainly for the purpose of addressing environmental concerns. Meanwhile, along with the improvement of vehicle dynamic performance, aerodynamic performance engineering for obtaining better vehicle dynamics has become increasingly important. This is true not only for high-performance vehicles that require outstanding vehicle dynamics, but also for small passenger cars that usually emphasize fuel economy. Previously, for the sake of improving handling and stability, efforts were made to reduce the time-averaged lift and yawing moment measured under uniform flow conditions. However, there is a case in which the improvement effects assumed for aerodynamic measures were not obtained in driving test

evaluations conducted under real-world conditions. [1, 2] In addition, lower lift generally involves a trade-off with lower drag. For many passenger vehicles, it is necessary to give priority to lower drag in order to comply with fuel economy.

For the reasons noted here, research conducted in recent years has focused on dynamic aerodynamic phenomena, and the results have made clear the importance of time-related changes in the aerodynamic forces acting on vehicles. [3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14] The factors causing time-related changes in aerodynamic forces are not just the vortices generated by separation, but also include natural winds in the driving environment and vehicle attitude changes induced by driving operations executed by the driver. Because such aerodynamic forces are very difficult to measure

experimentally, the phenomena involved have still not been fully explained. [15, 16] Simulating these phenomena requires a coupling between aerodynamics and vehicle motion. As a method executed in the past, one-way coupling which predict the aerodynamic loads in advance and calculates the vehicle motion based on it, tow-way coupling which calculates aerodynamic loads and vehicle motion at each time step was used. [10] In the tow-way coupling, real-time position of the vehicle used in CFD, calculated based on the preceding aerodynamic loads, so it's a closer system to predict the real-world phenomena. Therefore, a two-way coupled calculation method combining a vehicle motion model and an aerodynamics simulation system was selected and developed in this study with the aim of reproducing and making clear real-world aerodynamic phenomena. Large-scale aerodynamic calculations were run on a high performance computing (HPC) to obtain the aerodynamic forces acting on a vehicle in each axis and around each axis. Vehicle motion calculations were performed on a personal computer (PC) to obtain information on vehicle attitude changes. Coupled calculations were performed each time-steps on both sets of data through communication supported by the Transmission Control Protocol (TCP)/Internet Protocol (IP), taking into account the mutual influence of aerodynamic forces and vehicle motions. Moreover, a driver model was incorporated in the simulation system to reproduce dynamic aerodynamic phenomena that occur during driving more faithfully, and the amount of steering needed to keep the vehicle in the target driving lane was estimated.

The driving situation modeled in the simulation assumed that an ordinary passenger car traveling straight ahead was buffeted by a crosswind gust. The validity of the calculated results was verified in a comparison with experimental data. The calculation system was also used to investigate the effectiveness of aerodynamic parts that were developed empirically for the purpose of improving vehicle handling and stability. The simulation results agreed with the experimental data concerning the tendency for aerodynamic parts to reduce the amount of corrective steering done by the driver. In an analysis of the data, physical phenomena were visualized that supported the aerodynamic measures developed empirically.

Many automakers treat vehicle dynamic performance and aerodynamic performance as two separate specialized fields, and often the people working in these fields do not fully understand each other's way of thinking. The simulation technology proposed here is expected to be an effective approach for obtaining higher vehicle performance by promoting a shared understanding between specialists in different fields of expertise.

NUMERICAL METHODS

In this study, we used "FrontFlow/red-Aero (FFR)" [17, 18] being developed in Hokkaido Univ. as a CFD code. To predict the aerodynamics acting on a vehicle in motion, unsteady CFD method is indispensable. Then, we used large eddy simulation (LES) as a turbulence model. However, from the point of view of the computational costs, it is impossible to allocate the sufficiently fine meshes which can fully resolve the boundary layer on the wall and

small eddies. In addition, to insert the layer meshes on the wall to resolve the boundary layer is difficult in our method due to the complexity of the geometry of the vehicles. Thus, we used the governing equations of LES with the coarse meshes, and the logarithmic wall law to model the flow near the wall. That is, the method used in this study can be called a kind of wall modelled LES or very large eddy simulation. To reproduce vehicle motions, we used Arbitrary Lagrangian and Eulerian (ALE) method as the moving boundary method.

Governing Equations

The fundamental equations of LES are given by the spatially filtered equation of continuity and the Navier-Stokes equation under the assumption of incompressible and Newtonian flow:

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

$$\begin{aligned} \frac{\partial \bar{u}_i}{\partial t} + \frac{\partial}{\partial x_j} \bar{u}_i (\bar{u}_j - v) \\ = -\frac{\partial \bar{P}}{\partial x_i} + 2 \frac{\partial}{\partial x_j} (v + v_{SGS}) \bar{S}_{ij}, \end{aligned} \quad (2)$$

where variables that have a bar above them had been spatially filtered. The strain rate tensor $\bar{S}_{ij}$ and the pressure $\bar{P}$ are defined as:

$$\bar{S}_{ij} = \frac{1}{2} \left(\frac{\partial \bar{u}_j}{\partial x_i} + \frac{\partial \bar{u}_i}{\partial x_j} \right). \quad (4)$$

$$\bar{P} = \frac{\bar{p}}{\rho} + \frac{1}{3} (\bar{u}_k \bar{u}_k - \bar{u}_k \bar{u}_k). \quad (3)$$

The sub-grid scale (SGS) eddy viscosity v_{SGS} in Eq. (2) is modeled with the standard Smagorinsky model as:

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

where C_s is called the Smagorinsky constant and was set to 0.15. Here, Δ is the spatial filter length and is defined as the cube root of each element volume, and f_s is the damping function of the SGS eddy viscosity near the wall created by van Driest.

$$f_s = 1 - \exp \frac{-y^+}{25}, \quad (6)$$

where y^+ represents the non-dimensionalized distance from the wall.

Discretization

The governing equations were discretized in space with the vertex-centered unstructured volume method. The simplified marker and cell (SMAC) method was used in order to couple pressure and velocity. To solve the Poisson equation, the incomplete Cholesky conjugate gradient (ICCG) method was used. The 2nd order central difference scheme was used to discretize the spatial term. However, only the convective term contained 5% of the 1st order upwind scheme to stabilize the simulation, and the 3rd order upwind scheme was used for the distance from the vehicle to suppress numerical oscillation and to maintain the step-like profile of the crosswind because a coarse grid far from the vehicle was used. The first order Euler implicit method was used as the time integration method, and the increment of each time step was 4×10^{-5} s.

NUMERICAL SETUP

Vehicle Model

The vehicle that was the subject of this study was a small hatchback having the neutral steer point (NSP) positioned behind its center of gravity (CG). Generally, passenger cars were positioned the NSP behind the CG, resulting in understeer characteristics. In crosswind stability, if the aerodynamic force application point is far from the NSP, the vehicle will be rotated and becomes unstable. Therefore, it is one of the important factors in discussing the crosswind stability. As shown in Fig. 1, the vehicle dimensions were an overall length of 4.25 m, wheelbase of 2.70 m, overall width of 1.70 m, overall of 1.55 m and a frontal projected area of 2.23 m². The vehicle's center of gravity was used as the origin of the coordinate system.

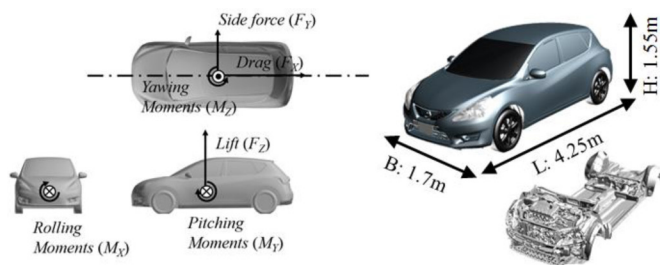


Figure 1. Vehicle model and surface mesh

Computational Domain and Boundary Condition

The computational domain and grid are shown in Figs. 2, 3, and 4. The surface resolution ranged from 4 mm to 12 mm and was composed of tetrahedral elements. Almost all of the computational domain was divided by the tetrahedral elements, but hexahedral elements were placed near the inlet and outlet to prevent numerical oscillations. The total cell number was about 40 million. As shown in Fig. 4, the inlet and side walls were set to the inflow boundary

condition, and crosswind was applied from the side walls. The inlet velocity condition was set to 0 m/sec, and the whole computational domain was moved at 33.3 m/sec in the direction of travel. The outlet was set to the outflow boundary condition (pressure was fixed and velocity gradient was zero), and the ceiling was set to the free slip condition. Regarding the boundary condition on the floor, a logarithmic wall law was implemented near the vehicle, while the free-slip boundary condition was implemented far from the vehicle to maintain the crosswind profile. In this study, tire rotation was not considered.

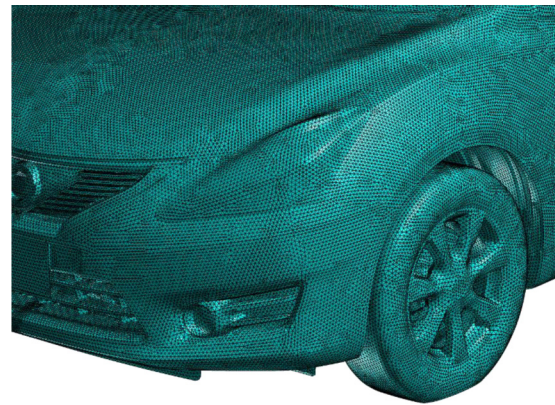


Figure 2. Surface mesh of vehicle

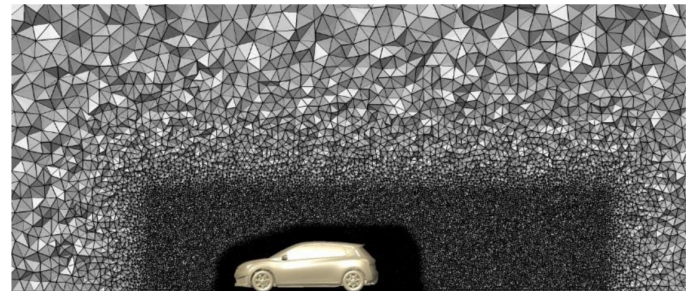


Figure 3. Grid around vehicle

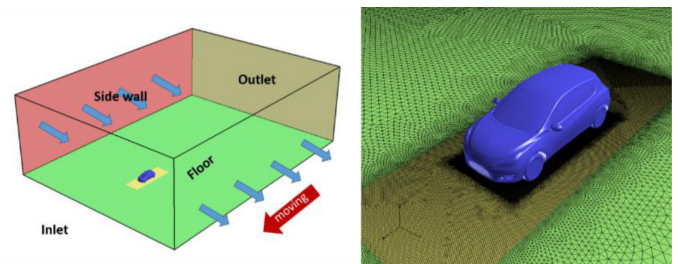


Figure 4. Computational domain and boundary condition.