

Development of an Unsteady Aerodynamic Simulator Using Large-Eddy Simulation Based on High-Performance Computing Technique

Makoto Tsubokura
Hokkaido University

Takuji Nakashima
Hiroshima University

Kozo Kitoh
Kozo Kitoh Technology, Inc.

Yoshihiro Sasaki
AdvanceSoft Co.,Ltd.

Nobuyuki Oshima
Hokkaido University

Toshio Kobayashi
Japan Automobile Research Institute

Copyright © 2009 SAE International

ABSTRACT

A numerical method specially designed to predict unsteady aerodynamics of road vehicle was developed based on unstructured Large-Eddy Simulation (LES) technique. The code was intensively optimized for the Earth Simulator in Japan to deal with the excessive computational resources required for LES, and could treat numerical meshes of up to around 120 million elements. Moving boundary methods such as the Arbitrary Lagrangian-Eulerian (ALE) or the sliding method were implemented to handle dynamic motion of a vehicle body during aerodynamic assessment. The method can also model a gusty crosswind condition. The method was applied to three cases in which unsteady aerodynamics are expected to be crucial.

INTRODUCTION

While most of aerodynamic testing on road vehicles are conducted in wind tunnels with vehicle body fixed and low turbulence intensities, recently, more attention is paid to unsteady aerodynamic forces[1][2], which act in the condition of such as strong gusty crosswind[3][4], sudden steering, driving over bumpy road, or overtaking, to achieve higher vehicle driveability or safety. The most conventional method to assess the stability in a cross wind condition at the moment is based on the quasi-steady method. However, limitation of the quasi-steady method has also been pointed out such as by Charuvisit et al. (2004)[5] in a strong cross wind condition.

Computational Fluid Dynamics (CFD) is an attractive approach as it can provide a large amount of data and detailed information on the flow field, that can help to understand the mechanisms by which unsteady aerodynamic forces act on vehicles. In fact, over the

past decade, attempts have been made to employ both the overset grid approach and the Arbitrary Lagrangian Eulerian (ALE) method [6, 7] to the study of flow in crosswind gusts with limited success. Previous investigations mostly adopted Reynolds-averaged Navier-Stokes (RANS) turbulence models, the accuracy of which is questionable for the transient aerodynamic response caused by the higher wave-number turbulence typical of flow in crosswind gusts.

Since precise estimation of the unsteady forces is difficult for wind tunnel[8][9][10], or conventional RANS based CFD, as well as on-road testing, an alternative method is strongly desired. Large-Eddy Simulation (LES) is a promising candidate for the solution, because it can reproduce unsteady three-dimensional turbulence structures around a vehicle or in an incoming flow. On the contrary, the disadvantage of LES is its high computational load. To overcome the problem, in the previous study, we optimized the unstructured finite-volume LES code FrontFlow/Red (FFR), which had been developed under the national project "Frontier Simulation Software for Industrial Science"[11], and the subsequent project "Revolutionary Simulation Software (RSS21)" [12], and successfully conducted one of the world's largest LES of flow around a formula car with 120 million numerical elements[13].

Thereby, the objective of this study is to develop an aerodynamic simulator based on FFR, which can predict transient aerodynamic forces. Three targets in which unsteady aerodynamics are expected to be crucial are considered here: a straight-ahead vehicle in pitching or yawing motion, and in a gusty crosswind condition. To track the vehicle in motion, moving mesh techniques such as a sliding-mesh and an Arbitrary-Lagrangian-Eulerian (ALE) method[14] were installed. To reproduce the gusty crosswind, an innovative boundary condition for the side-walls of computational domain was introduced.

NUMERICAL METHOD

GOVERNING EQUATIONS – In Large Eddy Simulation the equations of motion (continuity and momentum) are spatially filtered. These read, in tensor notation:

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

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

in which over-bar denotes the spatially filtered quantity; u_i , ν , and ρ are the i -th velocity component, the kinetic viscosity, and fluid density, respectively. The strain rate tensor and the filtered pressure in eq. (2) are:

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

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

The effect of subgrid-scale (SGS) turbulence on the grid-scale turbulence motion is represented by the SGS eddy viscosity, which is modeled following Smagorinsky [15]:

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

where Δ is the volume of the generic numerical element, and 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 solid using a Van-Driest-like function:

$$f = 1 - \exp \frac{-l^+}{25} \quad (6)$$

Where l^+ is the distance from the wall in wall coordinates.

DISCRETIZATION – The governing equations are discretized in space by a vertex-centered unstructured finite volume method. Second-order central differences are applied for the spatial derivative, blended with a first-order upwind scheme for the convective term in the Navier-Stokes to avoid the excessive numerical oscillation appearing at coarse tetrahedral elements. It should be noted here that use of the first-order upwind should be avoided, whenever possible, in LES due to the excessive amount of numerical dissipation introduced. On the other hand the dissipation properties of upwind schemes may be desirable to a certain extent for engineering applications of LES on unstructured meshes such as those adopted here. As a compromise, the contribution of the upwind discretisation to the convective fluxes is set to be as low as 5%. In LES, first-order explicit Euler scheme should not be used for the time marching method, because of its absolute instability. On the other hand, fully implicit schemes often used for conventional engineering LES hopefully should be avoided because longer time increment permitted by the fully implicit scheme happens to damp higher wave-number turbulence. Thus, the second-order Adams-Bashforth scheme is adopted for time discretisation in this study. The SMAC (Simplified Marker and Cell) method is employed to maintain coupling between the pressure and the velocity.

SOFTWARE – The computational code adopted in the present study was originally developed in the context of the "Frontier Simulation Software for Industrial Science" project [11]. The project started in 2002 as an IT research program sponsored by Ministry of Education, Culture, Sport, Science and Technology; and was successfully concluded in 2005. The code was intensively optimized by the authors for simulations

under the subsequent IT project “Revolutionary Simulation Software (RSS21)” [12], and parallel and vector efficiency as high as 96% and 99% respectively were achieved in simulations performed on 100PNs/800CPUs. This allowed completion of LES calculations of flow around a formula car with complex geometry in about 120 hours [13]. The computational meshes adopted counted up to 120 million cells with a typical memory usage of more than 500 GB, also including data storage for the calculation of time-averaging turbulence statistics. The code has now been optimized for unsteady vehicle aerodynamics simulations under the project supported by an Industrial Technology Research Grant Program in 2007 from New Energy and Industrial Technology Development Organization (NEDO) [16] of Japan.

DYNAMIC PITCH MOTION

Contrary to the common aerodynamic practice that a vehicle with lower aerodynamic lift coefficient shows better straight driving stability at high speed, it has been found by Okada et al.[17] that two geometrically different sedan-type vehicles with almost identical low aerodynamic lift show remarkable difference of high-speed stability through on-road measurements. They further investigated the relationships between flow structures around the vehicles and the high-speed stability, and suggested that transient vortex structures above the trunk deck contribute to the aerodynamic pitching stability. In the successive study[18], they developed simplified car models, which represent the two sedan-type vehicles studied in the previous study in terms of the flow structures above the trunk deck, and tried to clarify the physical mechanisms of causing the aerodynamic pitching instability by means of the unsteady aerodynamic simulator developed in this study.

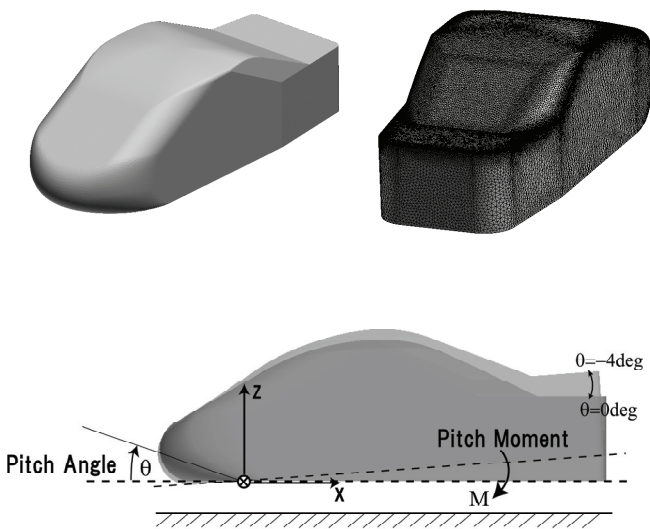


Fig. 1 View of the simplified model for the aerodynamic stability test and surface grid allocation(above), and the coordinate system (below).

The ALE method[14] was adopted to capture the dynamic pitch motion of the vehicle, and sinusoidal oscillation was imposed for its pitch angle, which is given as follows:

$$\theta = \theta_0 + \theta_1 \sin \phi(t), \quad \phi(t) = 2\pi f t \quad (7)$$

Both coefficients θ_0 and θ_1 are set to be -2.0 degrees in the present study, then the pitch angle oscillates between 0 and -4 degrees. The target simplified vehicle is shown in Fig. 1, together with the surface grid resolution adopted and the coordinate system. The amplitude of the oscillation is the same as the wind tunnel experiment, while a frequency f is set to be 10 Hz for the Strouhal number $St = 0.13$ considering the previous on-road test of the real passenger sedans, in which St is around 0.15.

The vehicle is about 1/20 scale of the real passenger sedan, with 210mm length (L), 80mm width (W), and 65mm height (H). The numerical domain is the rectangular duct with the size of $11.0L \times 9.0W \times 7.2H$, for streamwise, spanwise, and normal-wall directions, respectively. The vehicle is mounted $3.14L$ downstream from the inlet plane with the clearance of $0.23H$ from the floor.

At the inlet, uniform and constant flow velocity $U_0=16.7\text{m/s}$ was imposed. Reynolds number, based on the vehicle length L and an inlet velocity, is about 2.3×10^5 . The vehicle body and most of the floor ($3.0L$ to $11.0L$ from the inlet) were treated by the wall-model assuming a fully developed turbulent boundary layer. On the other hand, the free-slip wall condition was imposed on the upstream floor (from the inlet to $3.0L$), which represents a boundary-layer suction device. The side and top walls are also treated as free-slip boundary. The computational domain was covered by about two million unstructured (tetra and prism) elements with 1.5 million nodes. The computational grids on the central cut-plane are shown in Fig. 2. The minimum distance between the nodes on the surface is 0.5mm on the edges. Nine layers of prism mesh are inserted in the vicinity of the vehicle body and thickness of the first layer is about 0.1 mm. The other region is divided by tetrahedral meshes except for the prism-layer on the outlet boundary. In the grid deformation process of the ALE method, only the tetrahedral elements are deformed.

Phase-averaged pitch moment acting on the vehicle during the dynamic pitch motion is illustrated as a non-stationary case in Fig. 3. The results were averaged over eight periods of the pitch oscillation. For reference, the pitch moment when the vehicle motion was fixed to corresponding pitch angles ($\theta=0, -2, -4$ degrees) is also indicated as a quasi-stationary case. In both cases, the moment changes in a sinusoidal manner with respect to the phase angle ϕ and the mean pitch moment over the

phase ϕ is about -0.013 Nm, however, its dependence on the phase is quite different. In the quasi-stationary case, the pitch moment becomes larger at $\theta=0$ degrees ($\phi=270$), and smaller at $\theta=-4$ ($\phi=90$), while their difference is only about 0.002Nm.

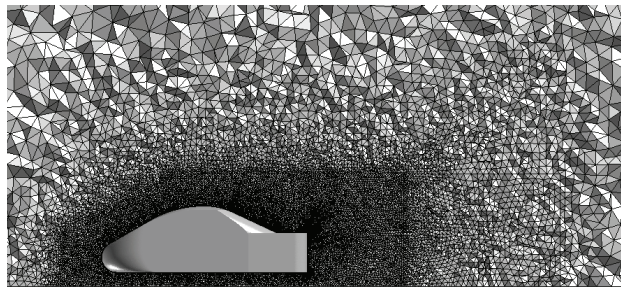


Fig. 2 The computational grids on the symmetry plane.

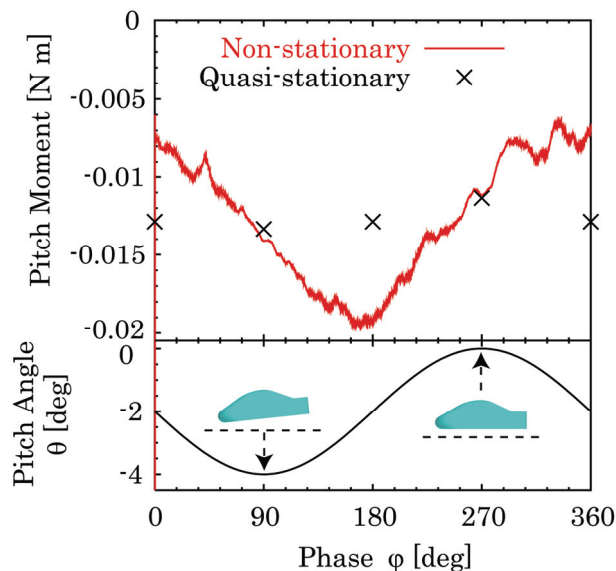


Fig. 3 The aerodynamic pitching moment during the dynamic pitch motion (non-stationary) and the quasi-stationary cases.

On the other hand, drastic unsteady aerodynamic force appears in the non-stationary case when the pitch is dynamically oscillated. The moment reaches maximum of -0.008 Nm at $\theta=-2$ and $\phi=0$ degrees, while it reaches bottom of -0.019 Nm at the same $\theta=-2$ but $\phi=180$ degrees. The important feature to be noted is the phase difference between the quasi- and non-stationary cases. In the non-stationary cases, the pitch moment reaches maximum when the rear end moves upward ($\theta=-2$, $\phi=0$ degrees), and becomes minimum when the rear end moves downward ($\theta=-2$, $\phi=180$ degrees). This fact leads to the important result that the unsteady pitch moment caused by the forced pitching oscillation acts to stabilize the vehicle pitch motion. Such an aerodynamic stability

can not be assessed by the conventional quasi-steady analysis, and the unsteady analysis is indispensable.

DYNAMIC YAW MOTION

Dynamic yaw-angle change of approaching flow with respect to the vehicle center line occurs when a driver suddenly turns the steering wheel to avoid risk or a vehicle is subjected to gusty crosswind. Unsteady aerodynamic forces are supposed to act on a vehicle in such situations, and a reliable prediction of unsteady or transient aerodynamic effects is important to achieve safety and comfort of vehicles. For this purpose, we have installed a sliding-mesh technique to our LES to capture the dynamic yawing motion of a vehicle.

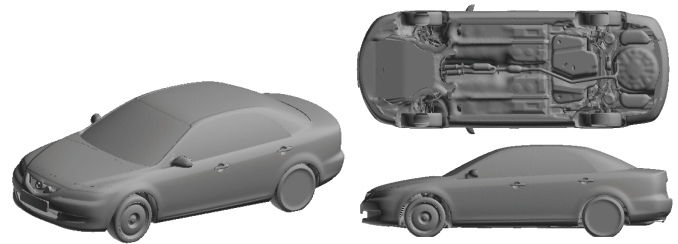


Fig. 4 Views of the full-scale sedan.

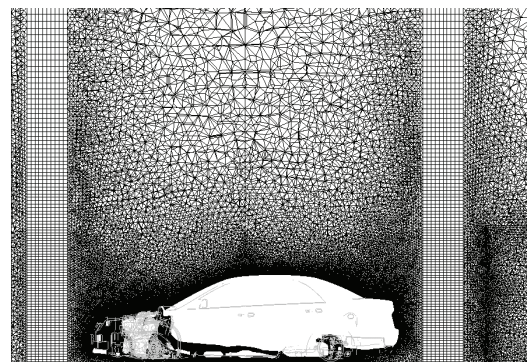
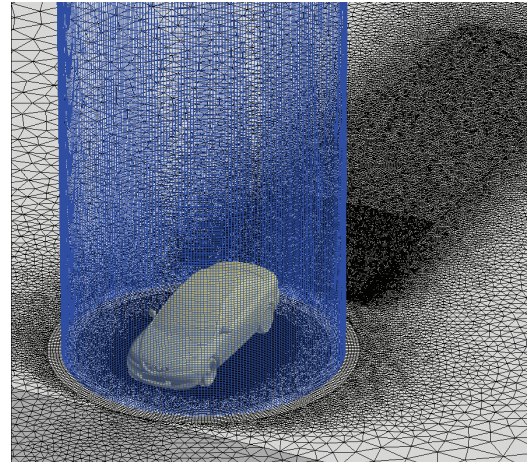


Fig. 5 Grid configuration around the sedan.