

Response of a Prototype Truck Hood to Transient Aerodynamic Loading

Anurag Gupta, Joaquin I. Gargoloff and Bradley D. Duncan

Exa Corporation

Copyright © 2009 SAE International

ABSTRACT

A study was performed to determine the fluid structure interaction (FSI) for a prototype truck hood for transient aerodynamic loads. The growing need to make vehicle panels lighter to enhance the fuel economy of vehicles has made hood panels more prone to deformation and vibration from aerodynamic loads. Moreover, as global pedestrian crash standards become more stringent to provide safer front end designs to minimize injuries to head and leg, automotive manufacturers are being required to design flexible hoods that crush significantly more than the present designs to absorb the crash energy better. These flexible designs lead to potentially undesirable deformations and/or vibration behavior of the hood at typical highway speeds. This paper presents a methodology for performing fluid structure interaction computations for a typical hood by detailing the process of computing dynamic (time varying) aerodynamic loads using CFD and related deformations of the hood using commercial structural solvers. The Lattice Boltzmann Method (LBM) was employed for performing transient Computational Fluid Dynamics analysis, and commercial finite element analysis (FEA) software was used for structural calculations. The coupling between the two codes, which is described in the paper, was achieved using proprietary in-house tools. The paper identifies the wind loading conditions that have significant potential to induce hood vibration by studying "disturbed" wind profiles such as those caused by high speed wakes, high speed travel at substantial yaw angles, and wind gusts. This work demonstrates the benefits of the inherent unsteady flow field computational capability of LBM in determining the dynamic structural response to aerodynamic excitation.

INTRODUCTION

Fluid structure interaction, or FSI, remains important to many disciplines of engineering, automotive being one of the important ones. Automotive designs are constantly in a state of flux, driven by ever changing market needs and government regulations. However, the challenges faced by the automotive industry and the need to innovate have never been greater. Rapid increase in fuel prices, stricter government oversight on pollution standards, continuously evolving pedestrian and occupant safety standards, and an increasingly demanding consumer has led to the need for innovation at an accelerated pace. The structural design of automobiles is at the forefront of this much needed design revolution to address these various challenges. For example, automotive panels need to be lighter and/or made of other non-traditional materials to lower the weight and gas consumption of vehicles. These panels need to reduce the impact to pedestrians and occupants during a crash. Automotive structures need to address crash safety, without sacrificing comfort and aesthetics. New designs to satisfy these rapidly evolving needs have increased problems in the domain of FSI due to increased sensitivity of lighter-weight structural panels to aerodynamic loads.

The vehicle hood is an important example. As automobile manufacturers have explored unconventional materials and reduced panel thickness to make the hood lighter to meet the new fuel economy and legislative standards, the problem of hood deformation and vibration under various highway driving conditions has worsened. Automotive manufacturers are exploring

solutions that can be applied early in the design phase of the vehicle so that these problems can be captured before they occur. Transient CFD methods coupled with FEA techniques can conveniently predict the impact of disturbed wind profiles on hood deformation and vibration. This early design evaluation helps manufacturers gain confidence in their design and reduce the cost of vehicle production. This paper discusses the application of coupled CFD-FEA techniques for the computation of transient wind profile with significant turbulence that is responsible for causing undesirable hood vibration. It investigates the effect of time varying gradients of pressure and the periodicity of load as the hood excitation parameters.

Because multidisciplinary test and simulation work performed by the manufacturers is often proprietary in nature, the literature on coupled physics simulation pertaining to automotive applications, such as FSI, is rather sparse. Duncan *et al.* conducted one such study [1] involving aerodynamics, thermal and acoustics. Ramsay *et al.* conducted a study [2] on the static deformation of the hood and compared the results to the experimental tests.

Other studies have been reported which examine the effect of transient cross-wind. Demuth and Buck conducted numerical investigations on the unsteady aerodynamics of road vehicles under gusty weather conditions [3]. Dominy and Ryan [4] conducted an experimental investigation into modeling of transient cross wind gust influences on passenger vehicles. They also looked at the aerodynamic forces in response to a transient crosswind at a relative incidence angle of 30° [5]

Examinations of transient aerodynamics on vehicles, in particular using LBM, have been reported by other authors. Albukrek *et al.* [6] showed the influence of transient flow structure on the performance of a detailed open-wheel race car, including the effect of steering angle. Duncan and Golsch [7] used statistical and spectral analysis to evaluate the unsteady flow features of various complex-flow regions on the exterior of a stock-car race car. The results showed that turbulent eddies were captured in both the low frequency range up to 100 Hz, which is dominated by fundamental and secondary instability mechanisms such as vortex shedding and wake oscillations, and in the mid-frequency range up to 1000 Hz and higher, in which the fluctuation spectra collapse to a theoretical cascade representative of isotropic turbulence. The low-frequency behavior of the various flow regions were used to characterize the nature of the turbulent flow structure in those regions, while the mid-frequency range for each collapsed to the more universal behavior when the simulation grid resolution was adequate. This study shows that flow structure details needed for understanding vibration of vehicle panels can be obtained

with such a simulation, for which accurate pressure loading must be provided in the low-frequency range.

The domain of computational aeroacoustics of vehicles is closely related to FSI, in that the transient pressure field on the surface produces a response of the structure and the interior of the vehicle. The requirements for transient CFD simulation are similar, and LBM has been previously validated for aeroacoustics applications, including the case of cross-wind.

In Senthoooran *et al.* [8] a detailed sedan was simulated at both zero and ten degree yaw angle for both aerodynamics and aeroacoustics analysis, and wall-pressure fluctuation data were compared between simulation and experiment. The geometric details such as complex grille geometry, glass offsets, A-pillar moldings, side-mirror shape, and door handle geometry are evident in the model. Detailed flow structures were visualized in the flow field and on the surface side-mirror surface, to gain understanding into the pressure fluctuation spectra. In Senthoooran *et al.* [9] the pressure fluctuation spectra on the side glass were evaluated on a vehicle with and without a side-mirror.

COMPUTATIONAL APPROACH

AERODYNAMICS

The commercial software, PowerFLOW was used for performing time-accurate, turbulent fluid dynamics simulations in this study. This software uses a special discretization of the Lattice Boltzmann method (LBM) on a variable resolution Cartesian volumetric grid, which is automatically generated by the software. In the LBM method, the fundamental equation of motion for fluid particles, the Boltzmann equation, is solved on a cubic lattice using a discrete set of fluid velocity states. The discrete Boltzmann equation is coupled with the ideal gas equation of state to represent realistic transfer of mass and momentum sufficient to describe small-scale fluid motions with very low numerical dissipation. The surface geometry is provided as an input in the form of a triangular surface mesh. The geometry of the surface enters the LBM discretization as boundary facets immersed in the lattice, and they provide boundary conditions for the discrete velocity states by enforcing zero-flux through the wall. The LBM thus provides a mass and momentum solver that is capable of computing the evolution of shear-layer instabilities, vortices, and pressure waves with high-order accuracy, in the presence of complex geometric boundaries – aspects of disturbed fluid flow critical to accurately predicting the wind induced vibration behavior of structures. A feature of the solver is that the method is inherently time-dependent. A detailed account of the LBM method and the treatment of unsteady flow and turbulence can be found in references [10] through [14].

STRUCTURAL MECHANICS

The commercial finite element code, MSC.NASTRAN was used for structural computations. The structural mesh consisted primarily of shell elements. However, limited number of beam and solid elements were also used in the model formulation. The fluid flow calculations from PowerFLOW and the structural computations from NASTRAN were one-way coupled in this study. In other words, the structural excitation was explicitly defined in the time domain and consisted of the time varying pressure field over each element in the structure model that was computed by the fluid solver. The solution process concluded upon the calculation of the dynamic structural deformation (vibration) from the unsteady pressure field. One-way coupling is an accurate and efficient FSI formulation for scenarios in which the structural deformation caused by the unsteady fluid flow is relatively small as compared to the physical dimensions of the structure such that an iterative exchange of boundary conditions between the fluid and the structural solver is not needed. The transient structural calculations were performed using the modal transient response method. This analysis methodology is especially advantageous for models with a large number of degrees of freedom and for the computation of a large number of time steps. This is a common analysis technique, the details of which can be found in [15].

FLUID-STRUCTURAL COUPLING

The coupling between the fluid and the structural disciplines was achieved by using an in-house tool that maps the pressures from the fluid mesh (used in PowerFLOW calculations) on to the structural mesh. The tool allows the mapping of either the averaged pressure; the pressure transient history; or the pressure spectra that is obtained by performing Fast Fourier Transform of the pressure time history. This pressure export tool makes possible the computation of static, transient or harmonic calculations using NASTRAN. A separate pressure map is created on every element of the structural mesh that allows for an accurate representation of the spatial variation of pressure. It must be noted that in the most general case, the fluid and structural meshes are substantially different, not only because of the mesh fidelity which is much finer in the fluid model, but also because the type of elements used in the two meshes can be different. The pressure export tool used in this study is capable of accommodating various differences between the fluid and the structural mesh. A summary of the sequence of steps in the coupling loop for hood vibration calculation is as follows:

- 1) Evaluate the unsteady flow field using PowerFLOW.
- 2) Transfer the transient pressure field from the CFD solution to the NASTRAN model, using the pressure export tool.

- 3) Compute the dynamic structural solution using NASTRAN.

PHYSICAL MODEL

GEOMETRY AND TEST CONDITIONS

The fluid simulations were performed on a prototype pickup to provide a complete description of the aerodynamics under different gust conditions, including the interaction of the unsteady flow with the vehicle and the unsteady loads exerted on different structural components (hood and door). The geometry of the model was complete in terms of detail of trim, underhood and underbody components. The test conditions for the aerodynamic baseline were a free-stream velocity of 41.1 m/s (92 mph) at 101325 Pa pressure and 20°C temperature. The conditions were matched to emulate a physical wind-tunnel test in a static floor wind-tunnel. The cooling fans were removed (as an approximation to a free-wheeling fan), and open grille with porous media models for the radiator and condenser were employed. The computational domain was represented as a low-blockage static wind-tunnel condition with velocity inlet and pressure outlet, both at a distance from the vehicle which would not influence the air flow. The inlet was at 1.5 vehicle-lengths upstream, and the outlet at 3.5 vehicle-lengths downstream. The floor condition was matched to the wind-tunnel by providing boundary layer growth using friction on the floor surface in the vicinity of the vehicle. The ceiling was modeled as frictionless wall and the side-walls were modeled as periodic boundary conditions. Details of these boundary conditions follow in the subsection titled “transient aerodynamic simulation”. The setup of the physical model is shown in Figure 1.

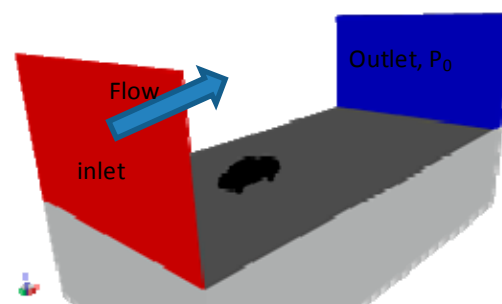


Figure 1. Computational domain to simulate an open-road condition

Grid Resolution - The simulation parameters and geometry specify the simulated test environment. To complete the specification of the simulation setup, the fluid domain is divided into nested regions surrounding the vehicle, and these regions are used to specify the grid spacing for the cubic lattice. The Lattice-Boltzmann Method in PowerFLOW allows the grid spacing to change by a factor of 2 between adjacent regions, and

so these regions are called Variable Resolution (VR) regions. For the aerodynamics baseline the finest cell size was set to 4.0 mm, and 5 levels of refinement were used between the edges of the flow domain and the finest region on the vehicle. The grid spacing and location of each region is summarized in Figure 2.

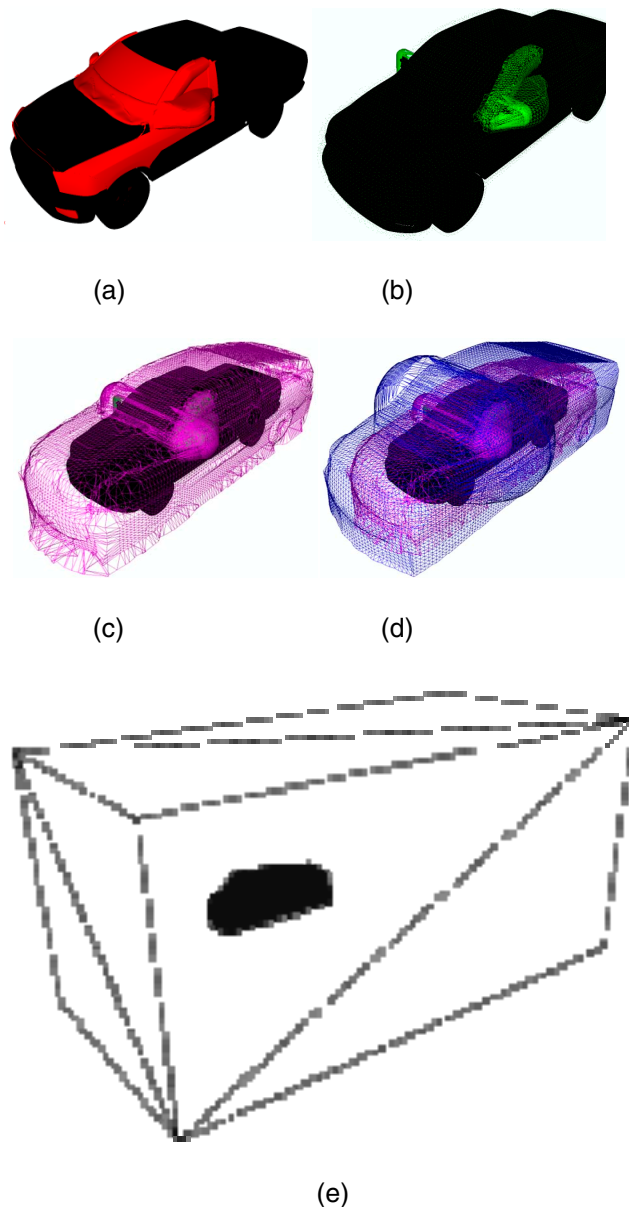


Figure 2. Variable resolution regions specifying the grid spacing for the cubic lattice grid: (a) VR9, 4 mm on key areas on driver-side A-pillar vortex, mirror wake, hood, windshield, cowl, and lateral of the vehicle, (b) VR8, 8 mm on key areas on driver-side A-pillar vortex, mirror wake, in region offset from upper body and enclosing the underbody, (c) VR7, 16 mm in near-field flow volume down to floor, (d) VR6, 32 mm large offset from VR7 and (e) VR5, 64 mm cells in box around vehicle and extending through the computational domain.

Figure 3 shows the location of the VR regions for each level from VR level 9 (4 mm cells) located at several critical locations on the vehicle surface such as the grille bars, front of the hood, cowl region including the windshield wipers, and in the cooling flow path from the grille to behind the radiator and condenser (not shown here), to VR level 8 (8 mm cells) at key locations mentioned above, and out to VR level 5 (64 mm cells) used to cover the entire computational domain. This particular resolution strategy has been standardized for production vehicles, with some flexibility left to individual judgment on the location of the finest regions, and the exact cell size used. This standardized approach has been validated to regularly produce drag predictions that match very well with wind tunnel and road tests. This approach results in models that take 2000-3000 CPU-hours to solve on a 128-CPU cluster with 3.0 GHz Pentium Xeon processors.

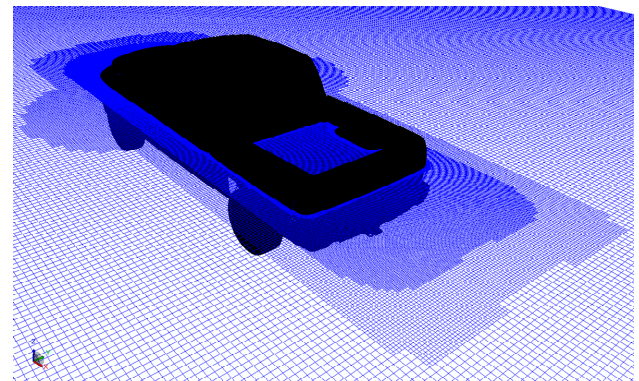


Figure 3. Planar slices of the cubic lattice grid.

TRANSIENT AERODYNAMICS SIMULATION

With the specified grid resolution setup, the cubic lattice grid is automatically generated during a pre-processing step. Then the fluid domain is initialized by running a no-gust straight flow simulation to produce a starting condition for future runs. The set of gust cases were then run starting from this initial condition, for an overall time window of 100,000 time steps, or 1.769 seconds of physical real time.

Gust setup - Four different fluid configurations were considered, in order to encompass a broad range of possible gust scenarios. These fluid configurations were:

1. Step at 20 degree yaw: This configuration simulates the flow over a vehicle plunging into a crosswind.
2. Harmonic yaw at 5 Hz: This configuration simulates the flow over a vehicle in a pulsating crosswind, such as the wake of a large vehicle with 5 Hz frequency content.