

CFD-based Modelling of Flow Conditions Capable of Inducing Hood Flutter

2010-01-1011

Published
04/12/2010

Adrian Gaylard and Martin Beckett
Jaguar Land Rover

Joaquin Ivan Gargoloff and Bradley D. Duncan
EXA Corporation

Copyright © 2010 SAE International

ABSTRACT

This paper presents a methodology for simulating Fluid Structure Interaction (FSI) for a typical vehicle bonnet (hood) under a range of onset flow conditions. The hood was chosen for this study, as it is one of the panels most prone to vibration; particularly given the trend to make vehicle panels lighter. Among the worst-case scenarios for inducing vibration is a panel being subjected to turbulent flow from vehicle wakes, and the sudden peak loads caused by emerging from a vehicle wake. This last case is typical of a passing manoeuvre, with the vehicle suddenly transitioning from being immersed in the wake of the leading vehicle, to being fully exposed to the free-stream flow.

The transient flowfield was simulated for a range of onset flow conditions that *could* potentially be experienced on the open road, which may cause substantial vibration of susceptible vehicle panels. As these conditions cannot generally be replicated in the wind tunnel, a comprehensive numerical simulation methodology is required if this issue is to be addressed before vehicle prototypes are built.

Transient aerodynamic simulations were performed using a Lattice-Boltzmann Method (LBM) and several different approaches were used to create dynamic onset flow conditions approaching the vehicle, such as simulating a:

- instantaneous transition to a twenty-degree yaw crosswind;
- twenty-degree harmonic yaw at 10 Hz;
- flow behind a 'bluff body' turbulence generating grid;
- 'passing disturbance';
- vehicle in the wake of another vehicle, at a range of separation distances.

A structural solver was used to predict the amplitude of vibration of the hood under these flow conditions. The amplitude response was largest when the frequency content of the oncoming flow matched the natural frequencies of the hood; the turbulence grid was found to produce the largest vibration for this reason. This study shows that transient aerodynamics simulation coupled with a structural solver can be used to predict the onset of vibration for different flow conditions, which could be encountered on the road but cannot be easily reproduced in the wind tunnel.

INTRODUCTION

The ability to simulate Fluid Structure Interaction (FSI) is becoming increasingly important in automotive development. Designs are constantly being adapted to meet changing market needs and government regulations. Increasing fuel prices, stricter government oversight of pollution standards and evolving pedestrian safety standards are shaping the structural design of vehicles. This trend will only continue.

In particular, as automotive panels become lighter, perhaps being made from non-traditional materials to lower the weight and fuel consumption of vehicles, they will still need to reduce the forces experienced by pedestrians and occupants during collisions. Customers will require that this is not at the expense of either comfort or aesthetics. This trend towards lighter-weight structural panels raises the possibility of increased problems resulting from FSI, due to increased sensitivity to aerodynamic loads. The vehicle bonnet (hood) is an important example. Reducing panel thickness or making use of unconventional materials to make the hood lighter, to meet the new fuel economy and legislative standards, may lead to deformation and vibration under highway driving conditions. Given this, it is important

that automotive manufacturers have simulation methods available that can be applied early in the design phase of the vehicle so that any problems can be eliminated before committing to tooling.

This requires the coupling of transient CFD methods with structural FEA techniques, as hood vibration is a function of both the unsteady aerodynamic loading and structural response. It is also vital that the CFD model is able to simulate particular instances of disturbed onset flow, as these are likely to represent the design case.

This paper discusses the application of a one-way coupled CFD-FEA technique for the simulation of hood (structural) response to transient wind profiles with turbulence levels representative of environmental conditions. It investigates the effect of time varying gradients of pressure and the periodicity of load as the hood excitation parameters.

PREVIOUS WORK

Because multidisciplinary test and simulation work performed by manufacturers is often proprietary, the literature on coupled physics simulation relevant to this application is rather sparse. For instance, Ramsay *et al.* reported a study [1] on the static deformation of the hood under an aerodynamic load, comparing the CFD results to experimental tests. However, this used classic wind tunnel onset flow conditions: steady, relatively uniform and low-turbulence; though the yaw angle was varied.

Moving away from traditional wind-tunnel simulations, other studies have been reported which examine the effect of transient crosswind. Demuth and Buck conducted numerical investigations on the unsteady aerodynamics of road vehicles under gusty weather conditions [2]. Dominy and Ryan [3] conducted an experimental investigation into modelling of transient crosswind gust influences on passenger vehicles. They also looked at the response of aerodynamic forces to a transient crosswind at a yaw angle of 30° [4].

Any computational approach needs to be able to represent the *intrinsic* unsteadiness in the flow around the vehicle. This kind of transient aerodynamic analysis, using LBM, has been reported by other authors. Albukrek *et al.* [5] showed the influence of transient flow structure on the performance of a detailed open-wheel racecar, including the effect of steering angle. Duncan and Golsch [6] used statistical and spectral analysis to evaluate the unsteady flow features of various complex-flow regions on the exterior of a stock 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 behaviour 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 behaviour 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 simulation of FSI has to meet requirements similar to those for computational aeroacoustics, 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 yawed onset flows.

For instance, Gaylard [7] reported a simulation performed on an SUV that captured the flow structures and wall pressure fluctuations over the front side glass. These compared well to data measured in the wind tunnel. In Senthoooran *et al.* [8] a detailed saloon (sedan) was simulated at both zero and ten degree yaw angles for both aerodynamics and aeroacoustics analysis, with wall-pressure fluctuation data obtained from CFD and experiment being compared. Geometric details such as complex grille geometry, glass offsets, A-pillar finishers, door mirror and door handle geometry were included in the model. Detailed flow structures were visualized in the flow field and on the surface of the door-mirror, to gain understanding of the pressure fluctuation spectra. Many of the same authors [9] also evaluated the side glass pressure-fluctuation spectra for a vehicle with and without a door-mirror. Finally, the ability to capture flow effects relevant to aerodynamics, thermal management, and aeroacoustics in a single calculation has previously been demonstrated by Duncan *et al.* [10].

The foregoing demonstrates that the LBM methodology is capable of resolving both the *intrinsic* and *extrinsic* flow unsteadiness required for investigating hood flutter. The coupling of the aerodynamic and structural simulations to investigate this issue has been reported previously by Gupta *et al.* [11]. This paper builds on that work, investigating the simulation of a range of onset flow conditions - which cannot be generated by the vast majority of wind tunnel facilities - capable of generating hood flutter.

COMPUTATIONAL APPROACH

AERODYNAMICS

Commercial CFD software (PowerFLOW) was used to perform the time-accurate, turbulent fluid dynamics simulations reported in this study. This software uses a

special discretization of the Lattice Boltzmann method (LBM) on a variable resolution Cartesian volumetric grid (lattice), 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 behaviour 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 the work of Gupta *et al.* [11], Lietz *et al.* [12], Chen *et al.* [13], Fares [14], Chen, Teixeira and Molvig [15] along with Teixeira [16])

STRUCTURAL MECHANICS

The commercial finite element code MSC NASTRAN was used for the structural computations. The structural mesh consisted primarily of shell elements. However, a 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 with 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 the NASTRAN User's Guide [17].

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. This tool allows either the mapping of the averaged pressure; the pressure transient history; or the pressure spectra that are obtained by performing a Fast Fourier Transform of the pressure time history. This pressure export tool allows 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. This tool is also capable of accommodating various differences between the fluid and the structural mesh. The sequence of steps in the coupling loop for hood vibration calculation can be summarised 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.

COMPUTATIONAL FLUID DYNAMICS MODEL

Geometry and Test Conditions

The fluid dynamics simulations were performed on a Land Rover SUV (Range Rover Sport), which included detailed upper body geometry, as well as all under body and under hood components. Unsteady flow simulations were carried out to provide a complete description of the aerodynamics under different onset flow conditions, including the interaction of the unsteady flow with the vehicle and the unsteady loads exerted on different structural components (the hood in this case). The baseline conditions for the simulations were a free-stream velocity of 44.7 m/sec (100 mi/h) with a pressure of 101,325 Pa and fluid properties consistent with a temperature of 20°C. These conditions were matched to emulate a physical test in a moving ground wind tunnel. The cooling fans were removed (as an approximation to a freewheeling fan). The simulation allowed air to flow into open cooling apertures, with porous media models representing the radiator, condenser, and charge-air cooler. The computational domain replicated an open-road condition by using a very low solid blockage, with a velocity inlet and pressure outlet both set at a distance from the vehicle that would not influence the airflow. The inlet was at 11 vehicle-lengths upstream, and the outlet at 14 vehicle-lengths

downstream. The ceiling was modelled as frictionless wall and the sidewalls were modelled as periodic boundary conditions. The setup of the aerodynamics model is shown in [Figure 1](#).

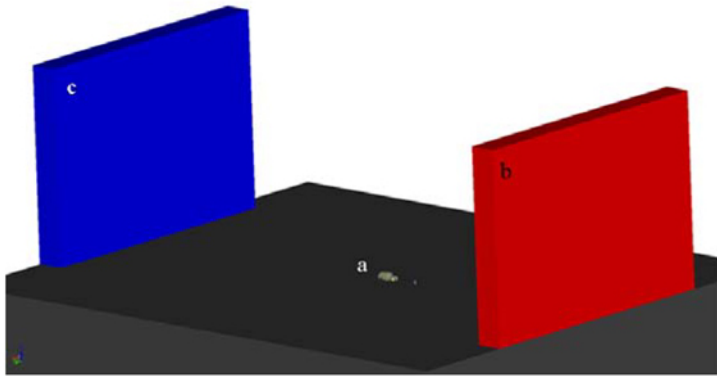


Figure 1. Low blockage computational domain, showing the vehicle (a), inlet (b) and outlet (c).

Grid Resolution

For the flow simulation, 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 two between adjacent (Variable Resolution - VR) regions. For the aerodynamics baseline, the finest cell (voxel) size was set to 1.5 mm, and 10 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](#). This shows (a) 1.5 mm voxels on the walls of the front intakes, (b) 3 mm voxels on key areas of the vehicle, including A-pillar vortex, mirror wake, cowl region, roof leading and trailing edges, front grilles, lower bumper, and edges of the rear face. Then, (c) 6 mm voxels offset from external surfaces and enclosing the under body/under hood, (d) 12 mm voxels in the near-field flow volume down to floor and enclosing the wake and finally (e) 24 mm voxels offset from the VR7 boundary.

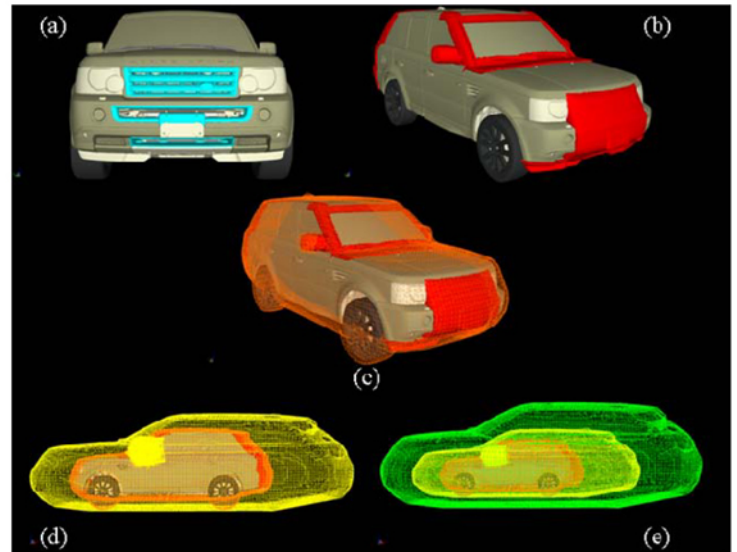


Figure 2. Variable resolution (VR) regions specifying the grid spacing for the cubic lattice: (a) VR10, (b) VR9, (c) VR8, (d) VR7, (e) VR6..

TRANSIENT AERODYNAMICS SIMULATION

Immediately prior to simulation, the domain is spatially discretized using the specified VR regions, producing a cubic lattice. Then the fluid domain is initialized by running a zero-yaw flow simulation to produce a starting flow field for future runs. The set of onset flow cases were then run starting from this initial condition.

Onset flow setup

Five different conditions were considered, in order to encompass a broad range of *possible* excitation scenarios. These configurations were:

1. Instantaneous transition to a twenty-degree yaw crosswind.
2. Harmonic yaw at 10 Hz.
3. Disturbance from a 'bluff body' grid.
4. Flow through a 'passing disturbance', created by suddenly removing the 'bluff body' grid.
5. Unsteady wake generated by an upstream vehicle.

[Figure 3](#) shows the yaw profile for the two gust models, instantaneous transition or step function, and harmonic yaw at 10 Hz. The peak yaw angle for both cases is twenty degrees.

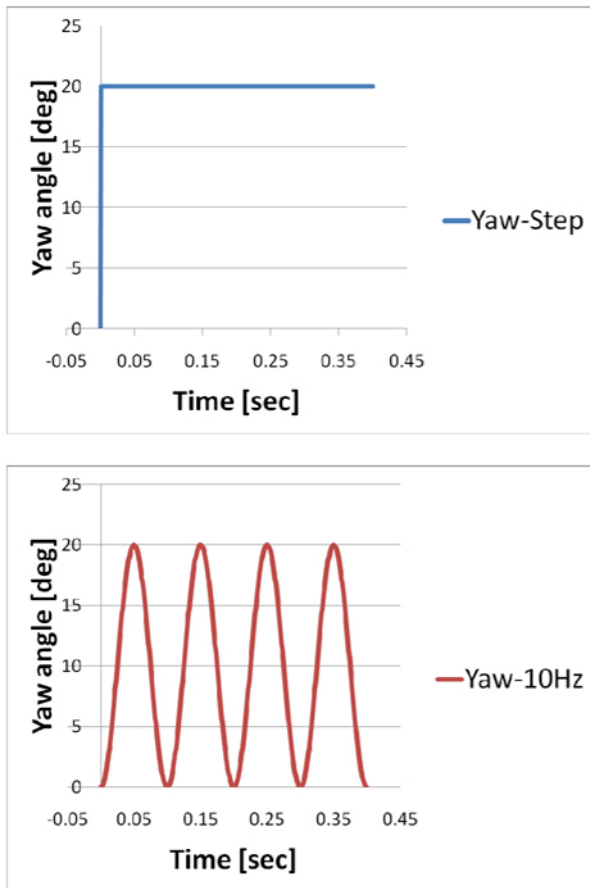


Figure 3. Yaw profiles for different gust models used in the present study (twenty degree yaw instantaneous transition, and 20 degree harmonic yaw with a 10 Hz periodic frequency).

The following sections will present the details of each of these five onset flow configurations. The figures show the resultant velocity magnitude non-dimensionalized by the free-stream velocity.

Instantaneous step to twenty-degree yaw crosswind

Stepping to twenty degrees yaw, models a vehicle that plunges into a crosswind. This condition can be experienced by vehicles emerging from tunnels, cuttings and between buildings in the presence of strong ambient wind. The simulation is initialized with a zero-yaw flow until the forces on the vehicle converge to a stable time-average, and then a lateral velocity component is added to the incoming flow such that the resulting flow impinges on the vehicle at a 20-degree yaw angle. The yaw front is convected downstream with the bulk flow.

<figure 4 here>

Figure 4 shows the flow structure as the yaw front passes over the vehicle. After the yaw front passes over the vehicle

and the transient stage is over, the flow settles to a standard 20-degree yaw onset flow. Vortex shedding due to the yawed flow is quite evident in the last image.

Harmonic yaw at 10 Hz

This represents the flow over a vehicle submerged in an onset flow that varies harmonically between zero and twenty degrees yaw, with a frequency of 10 Hz. Though this condition is idealized compared to natural wind, it could be thought of as emulating a gusty condition in which crosswind is varies over time. It could also be experienced by a vehicle passing a static array of obstacles on a windy day, or submerged in a wake flow dominated by 10 Hz frequency content. Again, the simulation is initialized with a zero-yaw flow condition until the forces on the vehicle ‘converge’; then a time-dependent lateral velocity component is added to the onset flow such that the resulting flow impinges on the vehicle with a time-dependent yaw angle. The oscillating yaw front is convected downstream with the free-stream flow. Figure 5 illustrates the flow structure as the simulated gusts are convected over the vehicle.

<figure 5 here>

‘Bluff body’ grid disturbance

The grid, consisting of five horizontal slats, represents the turbulence generated by the wake of a leading vehicle. According to Borg and Vevang [18], it was designed to provide a velocity deficit and turbulent kinetic energy level comparable to that seen 35m downstream of a truck and trailer combination, 1m above the ground. Nominal onset flow velocities of 24.6 m/s (55 mi/h), 40.2 m/s (90 mi/h) and 55.9 m/s (125 mi/h) were used.

The turbulence pattern generated by this grid has a dominant fluctuation peak that increases with velocity.

The simulation is initialized with a lower (spatial) resolution version of the simulation, generated by removing the three finest VR regions. The turbulent wake of the grid is convected over the vehicle, generating a very unsteady pressure distribution over its surfaces.

Figure 6 shows snapshots of flow velocity at two different time intervals as the flow passes through the grid and impinges on the vehicle. Substantial change in the velocity profile with time is an indication of strong unsteadiness in the flow.

<figure 6 here>

‘Passing disturbance’

This simulates the transition from immersion in an unsteady wake to the free-stream condition. This transition is simulated by initializing the flow with the flow through the ‘bluff body’

grid, and then suddenly removing the grid, allowing the remaining disturbance to be convected over the vehicle, and the flowfield to quickly transition to a free-stream condition. This situation can be encountered when passing a truck on the highway, where after travelling in the wake of the truck the vehicle moves to the side and is suddenly faced with free-stream flow. This sudden transition generates unsteady loads that are much higher than loads generated either in a low-speed turbulent wake, or in a high-speed steady flow condition.

Figure 7 shows the flow structure caused by the grid wake being convected onto the vehicle by the bulk flow. It provides snapshots of flow velocity at four time intervals as the grid wake is carried by the flow and impinges on the vehicle. High transient loads on the hood are observed at the instant that the grid gust wake disappears, and is replaced with free-stream flow (last image).

<figure 7 here>

Unsteady wake generated by an upstream vehicle
This is one of the most typical unsteady onset flow environments. For this study, another Range Rover Sport was positioned in front of the vehicle being studied. The position of the upstream vehicle was varied relative to the test vehicle. The configurations simulated are detailed in Table 1.

Table 1. Relative offset of the upstream vehicle relative to the test vehicle (L is the vehicle length and W the vehicle width)

Case:	$\Delta x/L$	$\Delta y/W$
1	0.25	0.0
2	0.25	0.5
3	0.5	0.0
4	0.5	0.5
5	1.0	0.0
6	2.0	0.0
7	4.0	0.0

Figure 8 shows snapshots of the unsteady flowfield for a selection of configurations.

<figure 8 here>

DYNAMIC STRUCTURAL SIMULATION

The construction of the hood and the boundary constraints are shown in Figure 9. The hood geometry considered in this study consisted of four main components: the top panel, the inner stiffening panel, two sets of hinges to facilitate the

opening and the closing of the hood, and two latch locks at the front. The sheet panels are aluminium with self-pierce rivet and adhesive connections, whilst hinges and latch-lock strikers are made from steel.

All panels were meshed primarily with four ‘noded’ quadrilateral shell elements.

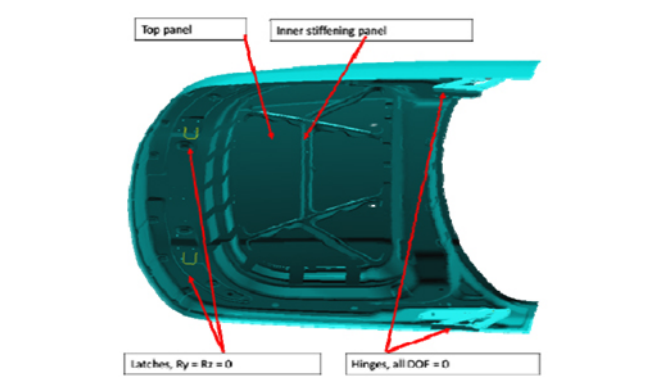


Figure 9. Structural model of the hood - construction and boundary constraints.

There are two primary constraint locations on the hood - the hinges and the latches. All degrees of freedom were constrained from motion at the hinges, as shown in Figure 9, to simulate a rigid connection to the vehicle body. The latches were constrained from rotation in the Y and Z directions.

An evaluation of the structural model and the accuracy of the boundary conditions was conducted by computing the natural frequencies and the mode shapes of the hood structure. The first six computed natural frequencies of the hood are shown in Table 2.

Table 2. The first six natural frequencies of the hood.

Mode	Frequency (Hz)
1	23.9
2	26.3
3	31.3
4	33.8
5	53.7
6	57.9

The first mode at 23.9 Hz exhibits “oil canning” like bending, partly aided by the fact that the levers connecting the hinge and the hood are allowed to rotate about the Y-axis. The second mode at 26.3 Hz is primarily a torsional mode. The third mode at 31.3 Hz is also a torsional deformation, with the hinges rotating about the x-axis, instead of deforming upwards as in mode 2. The fourth mode at 33.8 Hz is similar to