

Numerical Simulations and Measurements of Mirror-Induced Wind Noise

David Lepley

Nissan Technical Center North America

Sivapalan Senthooran and Dena Hendriana

Exa Corporation

Tony Frazer

Bruel & Kjaer

Copyright © 2009 SAE International

ABSTRACT

The high cost and competitive nature of automotive product development necessitates the search for less expensive and faster methods of predicting vehicle performance. Continual improvements in High Performance Computing (HPC) and new computational schemes allow for the digital evaluation of vehicle comfort parameters including wind noise. Recently, the commercially available Computational Fluid Dynamics (CFD) code PowerFlow, was evaluated for its accuracy in predicting wind noise generated by an external automotive tow mirror. This was accomplished by running simulations of several mirror configurations, choosing the quietest mirror based on the predicted performance, prototyping it, and finally, confirming the prediction with noise measurements taken in an aeroacoustic wind tunnel. Two testing methods, beam-forming and direct noise measurements, were employed to correlate the physical data with itself before correlating with simulation. Results of the correlations build confidence in both simulation and experiment, and confirm the use of exterior noise measurements as a substitute characteristic for interior noise.

INTRODUCTION

The ultimate goal in the simulation of automotive interior noise is to predict the noise that will be experienced by the customer. This noise is generated primarily by

engine and road forces and wind pressures loading the structure thereby exciting the vehicle acoustic cavity or there may be a direct path for exterior noise to leak into the cavity. With so many sources and paths for noise and different simulation technologies required to predict each one, the development of a fully integrated noise model is still in its early stages. In the mean time, it remains practical to develop substitute characteristics that predict overall interior noise based on component behavior. In the case of wind noise generated by an external mirror, the measurement of exterior noise on the surface of the side glass is an emerging standard. This is due to several factors: 1) the side glass is considered to be the primary path for mirror wind noise; 2) the physical glass is convenient to instrument during correlation experiments; and 3) there have been several papers documenting its successful use as a component level metric for interior noise [1,2]. The purpose of this paper is to confirm the accuracy of predicted exterior wind noise spectra on the outer surface of the side glass and determine whether it can be used as a substitute characteristic for interior noise. The results and conclusions supplement previous CFD studies [3,4]. In this case, the Lattice Boltzmann Method is used to predict the exterior wind noise. Countermeasures to improve the exterior noise are then developed based on analysis of the simulation results. The improved mirror is then prototyped and wind tunnel tests undertaken to confirm the improvement.

NUMERICAL SCHEME

THE LATTICE BOLTZMANN METHOD - The CFD code used for this study is based on the Lattice Boltzmann Method (LBM). Lattice based methods were proposed two decades ago [5] as an alternative numerical method to traditional Computational Fluid Dynamics (CFD). Unlike conventional methods based on solving the macroscopic continuum Navier-Stokes equations as Partial Differential Equations (PDEs), LBM starts from a “mesoscopic” kinetic equation based on the discrete Boltzmann equation for the particle distribution function, where the correct macroscopic fluid dynamics is obtained as a result of evolving the underlying particle distributions.

Use of a kinetic description makes the physics modeling simpler and more general than what is captured in the Navier-Stokes (N-S) equations. The physics is simpler since it is restricted to capturing the kinetic behavior of particles or collections of particles as opposed to attempting to solve non-linear PDEs, which is very difficult. This mesoscopic description is also more general since by augmenting the particle interactions at this level, more complex fluid physics, valid for a much wider range of spatial and time scales, can be modeled more accurately.

The lattice Boltzmann equation has the following form:

$$f_i(\vec{x} + \vec{c}_i \Delta t, t + \Delta t) - f_i(\vec{x}, t) = C_i(\vec{x}, t) \quad (1)$$

where f_i is the particle distribution function moving in the i th direction, according to a finite set of the discrete velocity vectors $\{\vec{c}_i : i = 0, \dots, b\}$. $\vec{c}_i \Delta t$ and Δt are space and time increments respectively. For convenience, we choose the convention $\Delta t = 1$ in the subsequent discussions. The collision term on the right hand side of Eq. (1) adopts the simplest and most popular form known as the Bhatnagar-Gross-Krook (BGK) form [6,7,8,9,10,11]:

$$C_i(\vec{x}, t) = -\frac{1}{\tau} [f_i(\vec{x}, t) - f_i^{eq}(\vec{x}, t)] \quad (2)$$

Here τ is the relaxation time parameter, and f_i^{eq} is the local equilibrium distribution function, which depends on local hydrodynamic properties. The basic hydrodynamic quantities, such as fluid density ρ and velocity $\vec{u}$, are obtained through moment summations:

$$\rho(\vec{x}, t) = \sum_i f_i(\vec{x}, t), \quad \rho \vec{u}(\vec{x}, t) = \sum_i \vec{c}_i f_i(\vec{x}, t) \quad (3)$$

In the low frequency and long-wave-length limit, for a suitable choice of the set of discrete velocity vectors, the transient compressible N-S equations are recovered through Chapman-Enskog expansion, in the limit of low

Mach number (~ 0.4). The resulting equation of state obeys the ideal gas law, and the kinematic viscosity of the fluid is related to the relaxation time parameter, τ , by [5,7,11].

$$\nu = (\tau - 1/2)T \quad (4)$$

The combination of Eq.s (1)-(4) forms the usual LBM scheme for fluid dynamics. It is solved on a grid composed of cubic volumetric elements called voxels, and Variable Resolution (VR) is allowed, where the grid size changes by a factor of two for adjacent resolution regions [12].

FLUID TURBULENCE MODEL - For higher Reynolds number flows it is not computationally practical to perform direct simulations by resolving all of the scales, thus it becomes necessary to incorporate turbulence models to account for the unresolved turbulent flow structures.

There are three basic categories of turbulent scales of motion: the dissipative range, the inertial range, and the anisotropic range. The dissipative and inertial ranges of turbulence are universal in nature lending themselves to a possible theoretical description. Turbulence theory is based on describing these universal aspects. The anisotropic turbulence contains the largest scales of turbulent motion and is not universal in nature, therefore turbulence theory does not apply to this range. The numerical scheme implemented in this LBM-based CFD code directly resolves the anisotropic turbulent scales and uses turbulence theory to model only what it applies to: the universal scales of turbulence in the dissipative and inertial ranges.

In order to model the effects of unresolved small scale turbulent fluctuations, the lattice Boltzmann equation is extended by replacing its molecular relaxation time scale with an effective turbulent relaxation time scale; i.e., $\tau \rightarrow \tau_{eff}$, where τ_{eff} can be derived from a systematic Renormalization Group (RNG) procedure [5,6] as

$$\tau_{eff} = \tau + C_\mu \frac{k^2 / \varepsilon}{T(1 + \tilde{\eta}^2)^{1/2}} \quad (5)$$

where $\tilde{\eta}$ is a combination of a local strain parameter $\eta = k|S|/\varepsilon$, local vorticity parameter $\eta_\omega = k|\Omega|/\varepsilon$, and local helicity parameters [13].

A modified k - ε two-equation model based on the original RNG formulation describes the subgrid turbulence contributions [14], and is given by:

$$\rho \frac{Dk}{Dt} = \frac{\partial}{\partial x_j} \left[\left(\frac{\rho v_0}{\sigma_{k_0}} + \frac{\rho v_T}{\sigma_{k_T}} \right) \frac{\partial k}{\partial x_j} \right] + \tau_{ij} S_{ij} - \rho \varepsilon \quad (6)$$

$$\rho \frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_j} \left[\left(\frac{\rho v_0}{\sigma_{\varepsilon_0}} + \frac{\rho v_T}{\sigma_{\varepsilon_T}} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{\varepsilon_1} \frac{\varepsilon}{k} \tau_{ij} S_{ij} - \left[C_{\varepsilon_2} + C_{\mu} \frac{\tilde{\eta}^3 (1 - \tilde{\eta} / \eta_0)}{1 + \beta \tilde{\eta}^3} \right] \rho \frac{\varepsilon^2}{k} \quad (7)$$

The parameter $v_T = C_{\mu} k^2 / \varepsilon$ is the eddy viscosity in the RNG formulation. All dimensionless coefficients are the same as in the original models [15,16,17]. The above equations are solved on the same lattice using a modified Lax-Wendroff explicit time marching finite difference scheme [24]. This description of turbulent fluctuation effects carries flow history and upstream information, and contains high order terms to account for the nonlinearity of the Reynolds stress [6,22]. This is contrasted with typical Navier-Stokes solvers, which tend to use the conventional linear eddy viscosity based Reynolds stress closure models.

WALL BOUNDARY CONDITION - In a fully resolved wall boundary layer flow, the no-slip boundary condition can be realized by a particle bounce back process on the solid surface. The resulting momentum flux across the fluid-solid interface corresponds to the pressure and wall-shear stress acting on the flow [3,7]. In the software used here, a generalized volumetric boundary scheme on arbitrary geometries has been implemented to precisely control the momentum flux across the boundary [4,8].

For high-Reynolds-number turbulent flows, the flow structures become tiny when approaching the wall; fine resolution in the near wall region is impractical. Hence a turbulent wall model is used to provide approximate boundary conditions for the near wall particles. In the software used in the current study, the following wall-shear stress wall model based on the extension of the generalized law of wall is used [8,9]:

$$U^+ = f\left(\frac{y^+}{A}\right) = \frac{1}{\kappa} \ln\left(\frac{y^+}{A}\right) + B \quad (8)$$

$$\text{with } A = 1 + f\left(\frac{\partial p}{\partial x}\right)$$

This equation is iteratively solved to provide an estimated wall-shear stress for wall boundary conditions in the LBM calculation. A slip algorithm [8,9] which is a generalization of bounce-back and specular reflection, is

then used for the boundary process. The estimated friction forces from the wall shear stress model are supplied to alter the momentum of scattered near wall particles. Compared to the standard wall function approach used in the former LBM framework, the extension to include pressure gradient information has provided better predictions for flows under adverse pressure gradient [9,13].

The following empirical boundary condition for turbulent kinetic energy and dissipation is imposed at the near wall lattice positions [5,23]:

$$k^+ = \frac{k}{u_{\tau}^2} = \frac{1}{\sqrt{C_{\mu}}} - e^{-0.1y^+} \left(\frac{1}{\sqrt{C_{\mu}}} + 0.29y^+ \right)$$

$$\varepsilon^+ = \frac{\varepsilon v_0}{u_{\tau}^4} = 0.04y^+ - 0.0033y^{+2} + 1.04y^{+3}/10^4 - 1.14y^{+4}/10^6 \quad (9)$$

EVALUATION PROCEDURES

COMPUTATIONS - The full geometry of the test vehicle, including detailed underhood and underbody geometry, was first tessellated into 1.5 million triangular facets, as shown here in Figure 1.

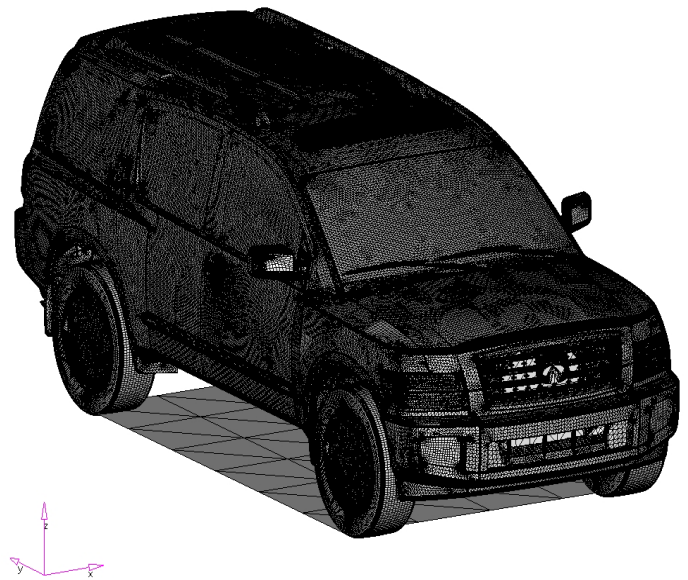


Figure 1. Tessellated vehicle geometry.

The vehicle ride height and static load radii for the front and rear tires were adjusted based on measurements of the test vehicle. The vehicle is simulated in a Virtual Wind Tunnel (VWT) with a very low blockage ratio of 0.1%, resulting in a fluid computational domain size of 136.6 m x 63.6 m x 52.0 m (LxWxH). This very low blockage ratio is close to the conditions of a vehicle running on the open road, however in a wind tunnel, the

blockage ratio is usually much higher. The tessellated geometry and surrounding air volume was then resolved in to approximately 5 million surfels and 62 million voxels, respectively. The voxels were organized into variable resolution regions ranging from 1280 mm in the outer most region to 1.25 mm in the inner most region near the mirror. (Fig. 2)

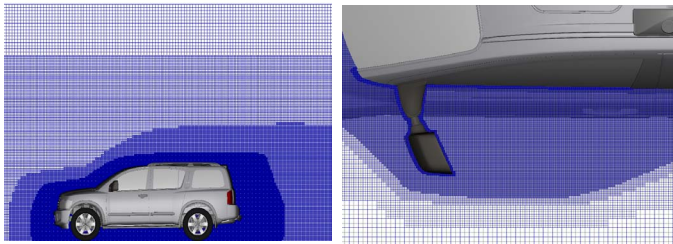


Figure 2. Cross-section of variable resolution regions.

The shape of the variable resolution regions are based on past experience and balance solution accuracy with computational efficiency. To ensure a proper flow distribution through the underhood passage ways and around the vehicle, the flow resistance for the radiator and condenser were modeled using porous media calibrated from corresponding pressure-velocity curves. A velocity inlet and constant pressure outlet condition were applied to the front and rear domain boundaries respectively. A slip-wall condition was applied to the top and on both sides of the domain boundaries. Fixed tires and ground were used in the simulation to represent the physical wind tunnel. A no-slip wall condition was applied to the ground upstream of the vehicle to obtain a floor boundary layer thickness similar to that in the physical wind tunnel test. To shorten the time to compute fully developed flow around the vehicle, the following coarse-to-fine seeding strategy was used: a coarse model was simulated with uniform velocity and pressure initial condition and then its result was used to initiate the solution of the fine model. The simulation time for the fine model was 1.0 second with a time step of 6.8 microseconds. Probe measurements were started at $t = 0.1$ s and fluid measurement from $t = 0.4$ s to avoid an initial unsteadiness.

To further economize on the computational time required to complete a simulation, a half-symmetric model of the vehicle was used. Because the windshield wipers on the vehicle that was evaluated are non-symmetric, they were removed from the model to preserve the assumption of symmetry. For the baseline run, a flow velocity of 120 km/h, yaw angle of 0° and production tow mirror geometry were simulated. A non-mirror configuration was also simulated in order to test the solver's ability to predict changes in noise due to changes in geometry. Models of both the Baseline and Non-Mirror configurations are shown in Figure 3.

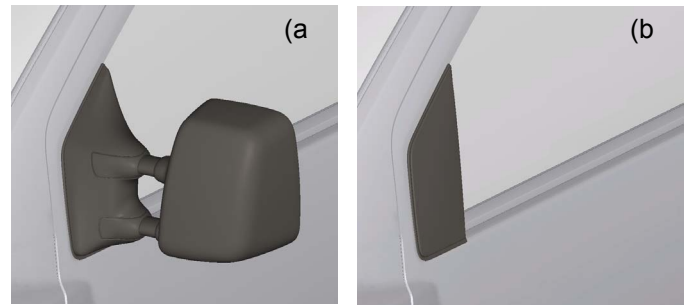


Figure 3. Models of the (a) Baseline and (b) Non-Mirror configurations.

EXPERIMENTS – Physical wind noise experiments were conducted in the Jacobs Sverdrup Aeroacoustic Wind Tunnel in Allen Park, Michigan. (Fig. 4) The baseline condition of 120 km/h flow velocity, 0° yaw angle and no wipers used in the simulations were replicated in the tunnel. To gain confidence in the experiments, two separate testing methods were employed simultaneously during separate rounds of testing spaced several weeks apart.

Phased Microphone Array – To capture the predicted spatial distribution of noise on the side glass, the first method used a Phased Microphone Array (PMA), also known as a beam-forming array, to acquire data simultaneously with a Flush Microphone Array (FMA). The PMA apparatus used in the test was a B&K model WA-0890-F circular array consisting of 90 microphones spaced over a 2.4 m diameter framework. The array was placed outside the open jet, 3.5 m away from the side glass as can be seen on the right hand side of Figure 4.

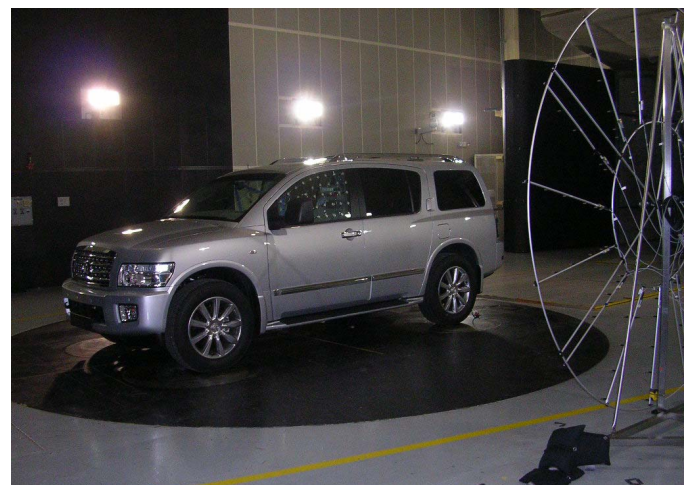


Figure 4. Wind tunnel, test vehicle and instrumentation.

Flush Microphone Array – The second test method used an array of B&K 4942-A 12.7 mm diameter condenser microphones, evenly spaced to cover nearly all the surface area of the production vehicle glass. (Fig. 5) Each microphone was carefully flush-mounted to the outer surface of the glass to avoid perturbing the flow. This method directly measures noise and turbulent