

Simulation of Underbody Contribution of Wind Noise in a Passenger Automobile

Robert Powell, Philippe Moron, Ganapathy Balasubramanian, Barbara Neuhierl, Sivapalan Senthoooran, Bernd Crouse and David Freed
Exa Corporation

Cornelia Kain and Frank Ullrich
BMW Group

ABSTRACT

Wind noise is a significant source of interior noise in automobiles at cruising conditions, potentially creating dissatisfaction with vehicle quality. While wind noise contributions at higher frequencies usually originate with transmission through greenhouse panels and sealing, the contribution coming from the underbody area often dominates the interior noise spectrum at lower frequencies. Continued pressure to reduce fuel consumption in new designs is causing more emphasis on aerodynamic performance, to reduce drag by careful management of underbody airflow at cruise. Simulation of this airflow by Computational Fluid Dynamics (CFD) tools allows early optimization of underbody shapes before expensive hardware prototypes are feasible. By combining unsteady CFD-predicted loads on the underbody panels with a structural acoustic model of the vehicle, underbody wind noise transmission could be considered in the early design phases. This paper describes a numerical process capable of simulating underbody contributions to interior wind noise. A CFD model based on the Lattice Boltzmann Method is applied to generate fluctuating pressure loads on the underbody. Structural and acoustic loads are then calculated and applied to a vehicle noise model based on Statistical Energy Analysis, with certain parameter updates from a finite element model to predict accurately the noise transmission through the underbody panels to the driver's ear location. Interior noise validation has been achieved on a series production sedan in an aeroacoustic wind tunnel, making use of acoustic insulation materials to block greenhouse transmission. Variation in flow speed and microphone location is included to verify prediction capability of this completely numerical process.

CITATION: Powell, R., Moron, P., Balasubramanian, G., Neuhierl, B. et al., "Simulation of Underbody Contribution of Wind Noise in a Passenger Automobile," *SAE Int. J. Passeng. Cars - Mech. Syst.* 6(2):2013, doi:10.4271/2013-01-1932.

INTRODUCTION

Competitive pressures on the cost and speed of product development processes are driving many industries to move from design development based on hardware prototypes to alternatives based on digital simulation. A particularly challenging problem is the simulation of automotive interior noise from exterior air flow at cruising speeds. Wind noise in automobiles is caused by aeroacoustic loads on the exterior, transmitting through many paths into the cabin acoustic field. Multiple technologies are required in order to efficiently simulate the physics of this process. A method has been reported on successful digital simulation of wind noise from side glass panels on a sedan by Moron et al [1] and on an SUV by Lepley et al [2] using a combination of CFD for the air flow and SEA for the structural acoustics. In these two

studies, the frequency range reported was 250-5000 Hz in third octave bands and both also showed proper design prediction with alternate side mirrors. Another recent model of vehicle wind noise transmission has been reported using deterministic finite element representations of the structural acoustics behavior of multiple glass panels and the enclosed cabin by Van Herpe et al [3], also using CFD to obtain transient exterior pressure loadings. Results up to the 1600 Hz third octave band frequency were reported in [3].

Application of the CFD/SEA side glass method to a number of different vehicles and configurations by several authors of the present paper provided a suite of test cases to validate the simulation. Statistical analysis of the errors between simulation and aeroacoustic wind tunnel experiment over 30 such cases suggested that the underbody air flow was the most significant factor contributing to differences at lower

frequencies [4]. The results published in [4] were for octave band levels from 500 Hz to 4000 Hz center frequencies.

Underbody wind noise contributions are relatively simple to measure with production vehicles in an aeroacoustic wind tunnel. Attachment of a skirt around the front and sides of the vehicle, with taped sealing to the ground, serves to redirect underbody air flow to the sides and roof, usually with only a minor effect on greenhouse noise contributions. Interior noise spectra with and without this underbody skirt can be used in a straightforward calculation of the underbody contribution to wind noise. A more direct underbody contribution method, also used in this paper, attenuates greenhouse panel contributions by interior acoustic insulators.

A previous CFD validation study by Crouse et al [5] has shown that good aeroacoustic accuracy is possible up to 400 Hz for underbody exterior pressure fluctuations. The basic approach of applying SEA/FEM hybrid floor modeling for interior transmission of underbody wind noise was proposed with a numerical-only comparison in a recent paper [6]. In the present paper, the methods used to extend the side glass CFD/SEA approach to handle vehicle underbody panels are presented with updates, now including experimental validation on a production vehicle. The side glass method itself is additionally validated over flow changes, including wind speed and yaw angle. For the separate underbody CFD setup, careful representation of the acoustic cavities under the vehicle was found to be quite important in the CFD model. The technique employed to represent panel stiffness effects due to curvature, beading and damping treatments involves modification of the SEA panel properties based on drive point conductance data from measurements or numerically from a Finite Element Method (FEM) model, as described originally by Manning [7, 8]. Automotive applications of this method have been described by Moeller et al [9] and also by Manning [10]. With such refinements from a trimmed body FE model, the CFD/SEA/FEM hybrid method for underbody wind noise was found to provide good simulation accuracy in the frequency range 63-500 Hz for both front and rear interior microphone positions.

Combining both side glass and floor panel simulations leads to an analytical representation of interior wind noise for a fully taped vehicle over two decades of frequency. Such simulation results for the vehicle under study are compared to aeroacoustic wind tunnel experiments over the frequency range 50 - 5000 Hz. Good absolute accuracy is obtained. The extension of the simulation methods to include other paths, such as sealing and additional panels, is suggested in the Conclusions section below.

SIMULATION APPROACH

Analytical simulation of interior wind noise is performed in the three steps shown in [Figure 1](#). The initial CFD step simulates the fluctuating pressure loads on the exterior panels of the vehicle at speed. These transient pressures are analyzed

in the frequency domain to develop load cases for a structural acoustics vehicle model.

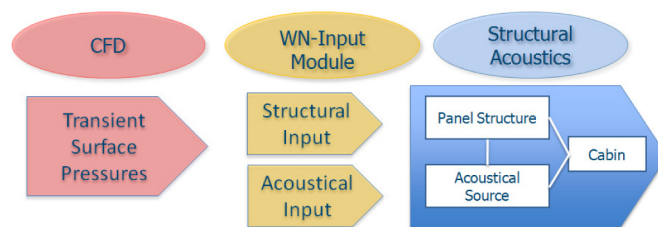


Figure 1. Interior wind noise simulation process

For each active panel in the vehicle model, structural and acoustic loads are calculated to provide inputs for the vehicle model of panel vibration and interior cabin noise.

Flow Simulation (CFD)

Transient pressures were obtained from the flow simulations performed using the commercial solver PowerFLOW. The simulation kernel of this software is based on the numerical scheme known as the Lattice Boltzmann Method (LBM), combined with a turbulence model. Simulations were performed by placing the fully detailed vehicle geometries in the digital wind tunnel, which has a very low blockage (0.1%). Flow conditions were set to match the test conditions. Variable resolution regions were used with fine resolution in critical regions to predict the boundary layer development and flow separation accurately and coarse resolution in non-critical regions to keep the computational effort efficient. Greenhouse simulations ran for 1.0 s in physical time with 1 mm finest fluid cell size, using existing best practice setup parameters. Pressure time histories were recorded on a very fine mesh over the greenhouse panels and sampled at 23 kHz. Underbody simulations were run separately from the greenhouse, to concentrate regions of fine resolution (2 mm finest cells) to front fascia components, tires and floor panels, as well as to allow longer simulation time (2.0 s) and lower effective sampling rates on the panel pressure measurements of 3.4 kHz.

Lattice-Boltzmann Method (LBM)

The CFD code used for this study is based on the Lattice Boltzmann Method (LBM). Lattice based methods were proposed almost three decades ago [11] as an alternative numerical method to traditional Computational Fluid Dynamics (CFD). Unlike conventional methods based on solving the macroscopic continuum Navier-Stokes (N-S) equations as partial differential equations, 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 kinetic description makes the fluid modeling simpler and more general than what is captured in the 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 [12,13,14,15,16,17]:

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

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 and absolute temperature T by [11, 13, 17]

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

The combination of Eqs (1,2,3,4) 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 [18].

Fluid turbulence model

For higher Reynolds number flows, it is not computationally practical to perform direct simulation 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 them 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., $\tau \rightarrow \tau_{\text{eff}}$, where τ_{eff} can be derived from a systematic Renormalization Group (RNG) procedure [11,12].

A modified $k-\varepsilon$ two-equation model based on the original RNG formulation describes the subgrid turbulence contributions [19]. All dimensionless coefficients are the same as in the original models [20,21,22]. In the CFD solver used for the present simulations, turbulence equations are solved on the same CFD lattice using a modified Lax-Wendroff-like explicit time marching finite difference scheme [19, 23]. 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 [14]. 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 [15, 24, 25]. 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 [18, 24, 25].

For high-Reynolds-number turbulent flows, the flow structures become tiny when approaching the wall; yet fine resolution in the near wall region can be 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, a wall-shear stress wall model based on the extension of the generalized law of wall is used [19]: The wall model equation is iteratively solved to provide an estimated wall-shear stress for wall boundary conditions in the LBM calculation. A slip algorithm [24], 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.

It should be noted that bypassing the viscous sublayer also has two other advantages. The first is that the empirical low Reynolds number formulations for the viscous dominated regions are avoided - RG theory cannot be extended to viscosity dominated flow regions [20]. The second is that the stability for the extended LBM is improved - the effective turbulent relaxation time scale in the fully turbulent region is much larger than the molecular one.

Coupling to Structural Acoustics

The computational procedure used here to provide suitable excitation for the SEA model of interior wind noise was detailed by Moron et al [1]. Using only one-way coupling is justified for wind noise because the very small amplitude of panel vibration in automotive structures at audio frequencies is considered negligible for any feedback on the flow. With this assumption, excitation of a structure by turbulent flow is described by a turbulent wall pressure fluctuation, providing stochastic distributed force on the side glass. Turbulence also generates a radiated acoustic field, even when acting on a rigid structure or in a free shear layer. This acoustic field is separated from turbulence in the excitation formulation using a low-pass wavenumber filter.

In the CFD solver described above, acoustics and turbulence are simulated simultaneously, and measured by thousands of surface elements on the panel surface. The contribution of an exterior acoustic field to interior sound is particularly important in the frequency range near the structural/acoustic coincidence frequency, where bending wavelengths match acoustic wavelengths. The long-wavelength acoustic field radiated from the turbulent pressure field couples more strongly to the bending wavelengths on typical glass panels than does the short-wavelength turbulence. More significant for the lower frequency range of the underbody contribution is the so-called mass law coupling of acoustics across long-wavelength structural modes. All of these effects are considered and included in the SEA modeling.

Statistical Energy Analysis (SEA)

Statistical Energy Analysis is a framework of methods that simulate mid and high frequency dynamics by considering statistical ensembles of mode groups and the dynamical energy exchange between them [26]. In the present approach, wind noise sources are calculated from the CFD transient surface pressures and transformed into power inputs for the SEA model. For an SEA model of automotive panels and interior acoustics, it is necessary to separately calculate direct power input to the panel modes and the external acoustic field, because of the different coupling calculations related to spatial wavelengths. The general

expression of the input power to a panel from a surface pressure field in a given frequency band can be written as

$$W_{in}(\omega) = \int_{\Omega} \frac{1}{2} Re(P^*(x, \omega) V(x, \omega)) dx \quad (5)$$

where the real part is taken over complex harmonic amplitudes representing pressure P and normal velocity V , and the integral is taken over the panel surface area Ω , with location x and angular frequency ω . By expanding the velocity response V into a separate area integral over surface pressure and structural mobility Y , input power can be expressed with pressure cross power spectrum S_p , and transfer mobility Y , in a spatial double integral

$$W_{in}(\omega) = \int_{\Omega} \int_{\Omega} Re(S_p(x, y, \omega) Y(x, y, \omega)) dy dx \quad (6)$$

These two parameters can be assumed independent for vehicle structures in air, since pressure cross power spectrum derives from the fluid dynamics while the transfer mobility is a structural parameter, and can be determined in vacuo. Full numerical calculation of equation (6) remains prohibitively expensive to perform directly, because the surface CFD model is comprised of hundreds of thousands of surface elements (surfels). In the current process, simplifications are adopted in the form of models to represent the pressure cross spectrum in terms of directly predicted auto power spectra and local coherence decay rates in streamwise and spanwise flow directions. These models of surface pressure take care of the convective turbulence scales, but cannot generalize easily to the three-dimensional acoustic field outside the panel. To estimate the exterior acoustic field, a low-pass wavenumber filter is applied over the surface.

Hybrid SEA/FEM panel modeling

In a similar level of approximation for the structure, the general expression of transfer mobility is first simplified to that of an infinite panel, a very common assumption in SEA. For the lower frequencies important in underbody contributions, the panel stiffness is modified by the ratio of infinite flat panel conductance to a spatial and frequency average of panel conductance from an FEM model.

Bending stiffness D of a uniform thin flat panel, with thickness h , Young's Modulus E and Poisson's Ratio ν is defined by

$$D = \frac{Eh^3}{12(1-\nu^2)} \quad (7)$$

SEA models using this bending stiffness seem to provide reasonable dynamic behavior for uniform panels with small curvature, such as vehicle glazing panels, above about 500 Hz. At lower frequencies, curvature and boundary conditions, as well as stiffening beads, generally lead to higher effective stiffness than that of a uniform flat panel of the same section.