

From Exterior Wind Noise Loads to Interior Cabin Noise: A Validation Study of a Generic Automotive Vehicle

Barbara Neuhierl and Sivapalan Senthooran
Exa Corporation

Reinier Toppinga and Anke Jäger
Daimler AG

Maarten Brink and Timo Lemke
Porsche AG

Philippe Moron and Raghu Mutnuri
Exa Corporation

ABSTRACT

The object of the validation study presented in this paper is a generic vehicle, the so-called *SAE body*, developed by a consortium of German car manufacturers (Audi, Daimler, Porsche, Volkswagen). Many experiments have been performed by the abovementioned consortium on this object in the past to investigate its behavior when exposed to fluid flow. Some of these experiments were used to validate the simulation results discussed in the present paper.

It is demonstrated that the simulation of the exterior flow is able to represent the transient hydrodynamic structures and at the same time both the generation of the acoustic sources and the propagation of the acoustic waves. Performing wave number filtering allows to identify the acoustic phenomena and separate them from the hydrodynamic effects.

In a next step, the noise transferred to the interior of the cabin through the glass panel was calculated, using a Statistical Energy Analysis approach. It is shown that the simulated results correspond well with experiments.

CITATION: Neuhierl, B., Senthooran, S., Toppinga, R., Jäger, A. et al., "From Exterior Wind Noise Loads to Interior Cabin Noise: A Validation Study of a Generic Automotive Vehicle," *SAE Int. J. Passeng. Cars - Mech. Syst.* 8(3):2015, doi:10.4271/2015-01-2328.

INTRODUCTION

The generic vehicle or *SAE Body* consists of a simplified cabin whose interior can be equipped with either reverberant or partly absorbing walls. To obtain more realistic flow and pressure excitation typical for production vehicles, an A-pillar and a series rear view mirror were attached to the exterior surface. Furthermore it has a glass side window, allowing noise transmission to the interior. The other panels are isolated to prevent sound excitation and transmission. Some details are described below.

A detailed description of numerous experiments performed on this model is given in [1].

Two different configurations of the generic vehicle body that have been examined in the experiments were simulated, focusing on a geometrical variation of the body that results in the generation of strong acoustic sources in the flow. This was achieved by offsetting the sideglass

inwards. In the experiments, comparisons with a "baseline" geometry where the sideglass fitted evenly to the body walls had shown that the backward facing step had indeed the desired effect.

For this study a combined CFD (Computational Fluid Dynamics) and SEA (Statistical Energy Analysis) based simulation method was applied which allows the prediction of interior windnoise - i.e. the sound transferred to the interior and recorded e.g. at the driver's head location - and shows in general good correlation with experiments (see e.g. [2], [3], [4]).

A computational aeroacoustics method able to both predict the transient hydrodynamic flow structures as well as the generation and at the same time propagation of the acoustic waves within the fluid requires an accurate, highly resolved simulation of the time dependent and compressible flow. Thus, a Lattice-Boltzmann approach, a kinetic scheme modeling the dynamics of particles

distributions, was used. The inherently transient and compressible nature of this approach combined with very little numeric dissipation makes it extremely suitable for efficient aeroacoustic simulation.

The noise transmission through the glass panel to the interior was then modeled by Statistical Energy Analysis, representing both the vehicle cabin and the panel excited by the flow. As material and dependent SEA input parameters, typical standard values usually appropriate for production vehicle glass panels were used.

The approach discussed and demonstrated here has been used intensively in the past for interior noise prediction for productive passenger vehicles. It was basically developed to provide a fast and efficient method to predict acoustics.

As such an analysis is very valuable in early phases of the product development process, where not all details and properties of a vehicle may be available yet, the fact that properties in the SEA are described via probabilistic parameters can be considered to be an advantage, allowing a statistical estimation of e.g. interior noise results.

COMPUTATIONAL METHODS

Lattice-Boltzmann Method

The CFD simulation is based on the Lattice-Boltzmann method, a kinetic scheme modeling the dynamics of particle distributions (see [5], [6], [7]).

As a result of the simulation; pressure acting onto the greenhouse glass panels is written out as transient, highly time resolved so-called "surface measurement files". It has been shown in earlier studies (see e.g. [8]) that the pressure distributions on the panels are captured very well.

Most of the conventional CFD methods solve the *macroscopic* Navier-Stokes equations - a system of partial differential equations describing the continuum. The Lattice-Boltzmann method hence is based on a *mesoscopic* kinetic equation for the particle distribution function. The respective macroscopic transient fluid behavior evolves from the particle distributions in time.

The Lattice-Boltzmann equation has the following form:

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

f_i is the particle distribution function which moves according to a finite set of discrete velocity vectors $\{c_i; i=0, \dots, b\}$ in the direction of i . $c_i \Delta t$ and Δt are the respective spatial and time increments. Here $\Delta t = 1$ is assumed.

The right side of equation (1) consists of the so-called collision operator C_i . It determines whether the fluid behavior is physical, e.g. if the necessary conservation laws are satisfied.

The most popular formulation is the Bhatnagar-Gross-Krook (BGK)-form (see [9], [10], [11], [12]):

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

τ represents a time relaxation parameter and f_i^{eq} a local equilibrium distribution function dependent of local hydrodynamic properties.

Macroscopic direct and derived fluid variables (density, pressure, velocity, momentum) are obtained by summing up the particle distributions:

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

In the case of low frequencies and wave numbers and for suitable discrete velocity vectors and small mach numbers ($\text{Mach} < 0.5$), the Navier-Stokes equations can be derived from the Lattice-Boltzmann equation by performing the so-called Chapman-Enskog expansion.

The resulting equation of state obeys the ideal gas law, and the kinematic viscosity is related to the relaxation parameter τ and temperature T (see [13], [14]):

$$\nu = (\tau - \frac{1}{2})T \quad (4)$$

Equations (1), (2), (3), (4) represent a typical Lattice-Boltzmann scheme. It is resolved on a regular cartesian grid. Local grid refinement is possible, so the resolution can be adapted to the simulated problem.

In order to model also small fluctuations in the flow field not resolved by the grid, the Lattice-Boltzmann equation is extended by replacing the molecular by a turbulent relaxation time scale: $\tau \rightarrow \tau_{\text{eff}}$. τ_{eff} is formulated by a RNG (Renormalization Group) procedure ([15], [16]). Such a simulation is called *Very-Large-Eddy-Simulation* (VLES), see [17], [18].

This typical Lattice-Boltzmann scheme is resolved on a regular cartesian grid. To adapt resolution to the simulated problem, local grid refinement is possible.

To be able to take into account the typically high-Reynolds number small boundary layers, a turbulent wall model is implemented, approximating the respective boundary conditions (see [19], [20], [21], [22]).

The Lattice-Boltzmann method thus simulates very efficiently transient, compressible fluid flows, meeting the requirement of representing both the flow and the acoustic field.

Statistical Energy Analysis

A detailed description of a Statistical Energy Analysis approach has been given e.g. by Moron et al, see e.g. [2]. The method has been tried and tested many times on production passenger vehicles, as mentioned above already. It was thus used to predict interior noise for the generic vehicle described in the present study as well.

The term “Statistical Energy Analysis” (SEA) describes methods where dynamic systems are represented by statistical distributions of properties rather than deterministic values, with energy or mean quantities (averaged in space and time) derived from it being the primary parameter of interest [23]. Such approach is particularly appropriate to simulate the behavior of complex structures in the mid and high frequency range, where the examined systems usually show a high mode density, making deterministic approaches extremely expensive.

A system simulated via SEA typically is divided up into several coupled subsystems which represent the statistical ensembles of mode groups. Such subsystems can e.g. be structural parts or fluid-filled regions. To simulate the dynamic behavior, the energy flows between the subsystems are modeled.

In order to use this technique to predict interior noise for passenger vehicles, the vehicle interior is described by fluid subcavities which are connected to the respective window glass panels. To take into account only the so-called “greenhouse windnoise” - i.e. the sound transferred to the interior through the windows - only the glass panels are modelled, while the contributions from other assembly group panels are neglected.

The excitation of a vibrating structure by a turbulent flow is caused by turbulent wall pressure fluctuations. It is considered to be a random process depending on both time and space variables. The influence of structure vibrations onto the flow is assumed to be insignificant, only a one-way coupling is performed.

The turbulent flow leads not only to an excitation of stochastically distributed force acting onto a panel, but it also generates an acoustic field. This cannot be neglected, as the long-wavelength acoustic field radiated from the turbulence field couples more strongly to the bending wavelengths on the sideglass and plays a particularly important role close to the so-called “coincidence frequency” where the bending and acoustic wavenumber match. Compared to that, the more short-wavelength (high wavenumber) turbulent pressures meet a higher impedance of the panel structures.

As different coupling calculations related to spatial wavelengths are used in the SEA approach to describe the panel excitation, it is necessary to separately calculate the direct power input to the panel modes and the external acoustic field. Thus the transient pressure simulated by the previous CFD are transformed into hydrodynamic and acoustic loads in the frequency domain. This is achieved by separating the pressure fluctuations propagating with speed of sound, using a low-pass wavenumber filter over the panel surfaces.

This process is schematically illustrated in Figure 1:

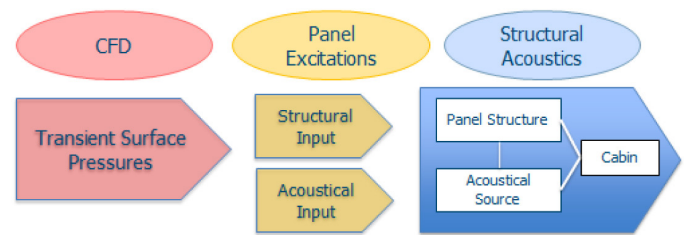


Figure 1. Scheme of Load Treatment for the SEA Approach

EXPERIMENT

Figure 2 shows the exterior of the generic vehicle model, placed inside an aeroacoustic windtunnel. The flow speed examined here was 140 kph, and the vehicle had no yaw angle.



Figure 2. SAE Body in Aeroacoustic Windtunnel, for more details see also [1]

The interior of the model can be seen below (Figure 3). To measure interior noise, 5 microphones have been placed in the interior.



Figure 3. SAE Body Interior, with Microphones, for more details see also [1]