

Numerical Simulations of Noise Induced by Flow in HVAC Ventilation Ducts

2011-01-0505

Published
04/12/2011Jun Chen
Purdue Univ.Zhengqi Gu
Hunan Univ.Yiping Wang
Hunan Univ. and Purdue Univ.

Copyright © 2011 SAE International

doi:10.4271/2011-01-0505

ABSTRACT

Numerical simulations are performed to investigate noise generated by flow in automotive HVAC ducts. A hybrid computational method for analyzing flow noise is applied: Large Eddy Simulation (LES) for predicting flow fields and Multi-domain boundary element method for predicting acoustic propagation. LES gives time-resolved solutions of flow velocity and pressure fields. By applying the acoustic analogy theory, the unsteady flow parameters are translated into sound source in evaluating the acoustic propagation. The computational result shows the noise caused by the HVAC ducts is strong. The noise is of broadband with a peak value at 370Hz. A major contribution of the noise generation is from the center ducts. Two design modifications of the center ducts are explored to regulate the flow structures with the ducts for reducing noise generation. Test results demonstrate the effectiveness of the modifications.

INTRODUCTION

With the demanding requirement for higher levels of passenger comfort, the interior acoustic environment of automobile draws more attention. Automotive manufacturers and system suppliers are making efforts to reduce sound generation and emission. The interior acoustic characteristic is dominated by noise generated by engine, transmission system, tyre-road, and HVAC (Heating, Ventilating and Air Conditioning) system, as well as aerodynamic and exhaust noise. Among them, the HVAC-induced noise is rarely

concerned in the past. However, as the development of automotive technology, especially in the era of hybrid or fuel cell vehicles, noise from power train can be highly diminished [1], and noise generated by the HVAC system becomes perceptible to passengers. If untreated, this factor can cause serious passenger discomfort and thus influences vehicle marketing in a negative way. Thus, quieter HVAC systems are pursued by car manufacturers and can serve as distinguishing design characteristics for marketing [2].

Prediction of the acoustic behavior generated by HVAC system in early design phase is desirable to reduce the development cost and to shorten the development cycle in automobile industry. Researches on flow noise induced by HVAC systems have been performed in laboratory experiments, and empirical formula relating the generated noise to pressure drop as well as flow rate is derived [3]. While detailed experimental study is very costly and empirical formula can only predicate the overall sound pressure level, the frequency spectrum of the sound pressure level cannot be obtained. With the development of computational fluid dynamics (CFD), statistical turbulence quantities extracted from steady state solutions of Reynolds-Averaged Navier-Stokes (RANS) equations, together with semi-empirical correlations, can provide a measure of the broadband source noise [4]. This method provides an approximate measure of the radiated noise at the source, but cannot provide tonal noise information or noise spectra. With the continued improvement of numerical simulation techniques, the prediction of sound generation and

propagation in complex flows and acoustic analogy becomes feasible [2].

The main noise source in HVAC systems is circulating fan and duct flow. It has been shown that different noise sources dominate over different frequency ranges [5]. At low frequency ($f < 500$ Hz), the noise generated by flow in the ventilation ducts dominates, and in the high frequency range, $f > 500$ Hz, the duct flow noise and the fan noise are about equally important. In most previous research, to predict aeroacoustic induced by turbulence inside duct is based on CFD coupling with Ffowcs-Williams and Hawkins [6] acoustic model [1] or Lattice Boltzmann Method [7,8]. However, the former can only be legitimately used to predict far-field noise due to external aerodynamic flows, e.g., flows around ground vehicles and aircraft. It can not be used for predicting the noise propagating inside ducts or wall-enclosed space [9], and the latter requires that the noise reception point must locate within the computational domain.

In this paper we describe a numerical procedure to study the noise induced by flow in HVAC ducts. Flow noise from ventilation ducts is predicted and analyzed using commercial CFD software Fluent 6.3 and acoustic software Sysnoise. Particularly, the time varying flow field is simulated using large eddy simulation (LES), through which the pressure fluctuations are identified as the source term for acoustics analysis. Then, the aeroacoustic pressure is calculated using BEM solver. This method has been applied to different aerodynamic noise problems, such as noise induced by side mirror [10, 15] and fan [11], etc. The results have shown good agreements with the experimental data.

The paper is organized as following: a general HVAC system is briefly depicted first; details of the methodology are discussed; the results of flow analysis and acoustic analysis are presented; in the discussion part, according to the flow and acoustic field result, two proposal are posed to reduce the noise, and the effect of noise reduction is tested by experiment measurements; conclusion is given at the end.

GENERAL HVAC SYSTEM

HVAC system is an important subsystem of an automotive, ensuring thermal comfort (cooling/heating the cabin air) and safety (clearing the windows for a good visibility) [1]. It is composed of instrument panel, ventilation ducts, defrosts ducts and evaporator assembly (Figure 1).

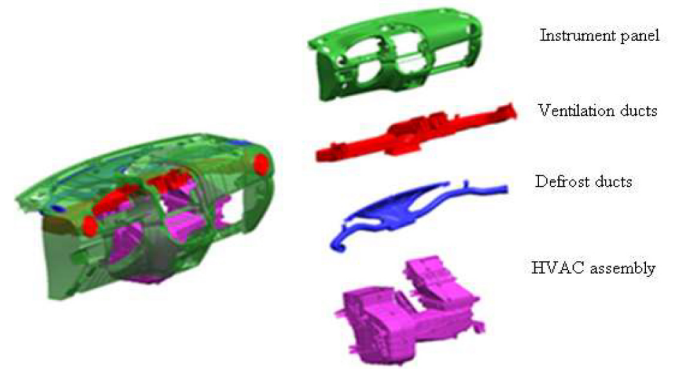


Figure 1. Description of a general HVAC system of automobile.

Because of the limited space between the HVAC and the outlet vents, the ventilation ducts must be designed to have a complicated 3D geometry that results in a highly complex duct flow. Particularly, the duct is too short to allow fully development and expansion of air flow through plenums and ducts, and sudden changes in duct dimensions and flow directions trigger flow separation, vortex shedding, unwanted flow circulation, as well as other complicated turbulent flow structure inside the air ducts. These features contribute to increasing noise generation [5]. A schematic of HVAC ventilation ducts used in this study is shown in figure 2. The ventilation ducts include two center ducts, one left and one right duct. The two center ducts are not left-right symmetric.

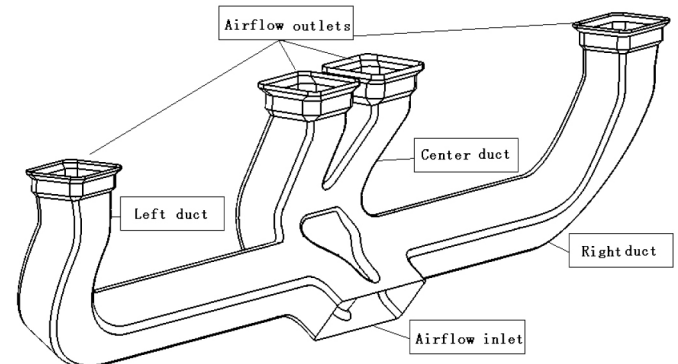


Figure 2. Schematics of HVAC ventilation ducts.

METHODOLOGY

AEROACOUSTIC MODELING

Aeroacoustic problem can be completely characterized by solving the compressible Navier-Stokes equations. For simple model problems, Direct Numerical Simulation (DNS) is able to solve for both the aerodynamic flow field and the acoustic field [12]. However, for high Reynolds number problems encountered in industrial applications, this approach is impossible due to the fact that DNS requires tremendous

amount of computational resources. It is also because that the acoustic energy and acoustic pressure perturbations are several orders of magnitude smaller than the aerodynamic energy and aerodynamic pressure, respectively. Furthermore, length and time scales of the flow and acoustic fields are not compatible [4]. Because of all these differences, it is necessary and feasible to decouple the flow field simulation and acoustic analysis.

A hybrid approach is first proposed by Lighthill [13], which lays down the foundation of aeroacoustics theory. Curle [14] extends Lighthill's theory of aeroacoustics from unbounded flow to bounded flows. Ffowcs-Williams and Hawkins [6] further extend this theory to flows with moving boundaries, in which the Navier-Stokes equations can be rearranged:

$$\frac{\partial^2 \rho}{\partial t^2} - c^2 \frac{\partial^2 \rho}{\partial x_i^2} = \frac{\partial}{\partial t} \{ \rho_0 v_n \delta(f) \} - \frac{\partial}{\partial x_i} \{ p_{ij} \delta(f) \} + \frac{\partial^2 T_{ij}}{\partial x_i \partial x_j} \quad (1)$$

where the three terms on the right-hand side describe contributions of monopole, dipole and quadrupole terms, respectively. These terms can be obtained from the flow parameters predicted by unsteady CFD simulations. ρ is air density, c sound velocity, ρ_0 stationary density, v_n the normal velocity, $\delta(f)$ the Dirac delta function, p_{ij} the compressive stress tensor, and T_{ij} the Lighthill stress tensor. If the solid surface is stationary, the monopole term can be neglected. The intensity of dipole term is proportional to the flow velocity [15], i.e.,

$$I_D \approx \rho u^6 c^{-3} l^2 \quad (2)$$

where u is flow velocity and l is the distance between two monopoles.

The intensity of the quadrupole term is [15]

$$I_Q \approx \rho u^8 c^{-5} l^2. \quad (3)$$

Thus, the ratio of these two terms is

$$\frac{I_Q}{I_D} \propto \left(\frac{u}{c} \right)^2 = Ma^2$$

where Ma is Mach number. For air flow of a characteristic velocity $u = 16$ m/s ($Ma \sim 0.047$ for standard air), the intensity of the quadrupole term is

$$I_Q \propto 0.002 I_D,$$

i.e., less than one percent of the dipole intensity. Thus, the dipole source is the most relevant noise generation term, and

the quadrupole sources in the turbulent flows can be ignored. Then the FW-H equation can be simplified to.

$$p' = - \frac{\partial}{\partial x_i} \int \frac{\{ \rho v_i v_j + P_{ij} \}}{4\pi r(1 - M_r)} \frac{\partial f}{\partial y_j} \delta(f) dS \quad (4)$$

where, r is distance between observer and sources. f is solid surface and ∇f gives its outward normal direction. M_r is the component of Ma (Mach number) in the direction of the radiation vector $\mathbf{r}$.

COMPUTATION OF UNSTEADY FLOW FIELD

Flow Simulation and Solver Settings

In order to obtain the aeroacoustic noise sources (in equations 1, 2, 3, 4), the unsteady flow structures of the flow field must be predicted first, and a suitable CFD technique should be able to resolve time-dependent small-scale fluctuations. This always involves effective combination of simulation tools, numerical model, mesh generation, spatial resolution, and time step.

Extensive efforts show that Unsteady Reynolds-Averaged-Navier-Stokes (U-RANS) simulations with a turbulence closure models (e.g., $k - \varepsilon$ model, $k - \omega$ model and SST model) are not enough to resolve the unsteady flow characteristics at all relevant scales, and thus not suitable to aero acoustic computation. In this paper LES is applied with Smagorinsky-Lilly [16] sub-grid scale (SGS) model employed to express the unresolved scales using resolved parameters. The eddy viscosity in the model is determined by

$$\mu_t = \rho L_s^2 \left| \overline{S} \right| \quad (5)$$

Where L_s is the mixing length for subgrid scales and $\left| \overline{S} \right| \equiv \sqrt{2 \overline{S_{ij}} \cdot \overline{S_{ij}}}$. In present study, L_s is computed using

$$L_s = \min(\kappa d, C_s \Delta) \quad (6)$$

where κ is the von Karman constant, d the distance from the closest wall, $\Delta = \nu^{1/3}$ the local grid scale, $\overline{S_{ij}}$ the rate of strain tensor, and C_s model constant with a value 0.1.

The LES model uses the following laminar stress-strain relationship to compute wall shear stress if the mesh resolution is found to be fine enough to resolve the laminar sub-layer [17]

$$\frac{\bar{u}}{u_\tau} = \frac{\rho u_\tau y}{\mu} \quad (7)$$

where, $\bar{u}$ is the mean velocity, u_τ the friction velocity, y distance from the wall, and μ dynamic viscosity.

However, the restriction that the near-wall meshes must be sufficiently fine imposes a very strict computational requirement. In present study, it is assumed that the centroid of the wall-adjacent cell falls into the logarithmic region of the boundary layer, and a law-of-the-wall is employed

$$\frac{\bar{u}}{u_\tau} = \frac{1}{\kappa} \ln E \left(\frac{\rho \mu_\tau y}{\mu} \right) \quad (8)$$

with a constant $E=9.973$.

In this paper the flow simulation is performed using commercial CFD software Fluent, and the solver is set with details listed in [Table 1](#).

Table 1. Solver Settings

Function	Setting
Solver	Segregated implicit Double precision
Time-stepping	2nd order implicit
Pressure discretization	2nd order
Momentum discretization	2nd order upwind
Pressure-velocity coupling	PISO
Fluid	Standard Air (20°C)

Mesh and Boundary Conditions

The computation mesh is generated using software ICEM. An unstructured tetrahedron mesh is created in the entire domain ([Figure 3](#)). In order to assure the quality of element, elaborate prism mesh is used on the duct surface, and a total of three prism layers are generated. Their heights increase successively by a factor of 1.2 and the height of the first layer is 0.1 mm ([Figure 4](#)). The boundary conditions applied to the different surfaces bounding the computational domain are listed in [Table 2](#).

Table 2. Boundary conditions.

Boundary	Boundary Condition	Value
Inlet	Mass Flow Rate (Turbulent intensity: 10%)	0.146 kg/s (Provided by manufacturer)
Outlet	Constant Pressure	0 Pa (gage)
Duct wall	No-slip	—

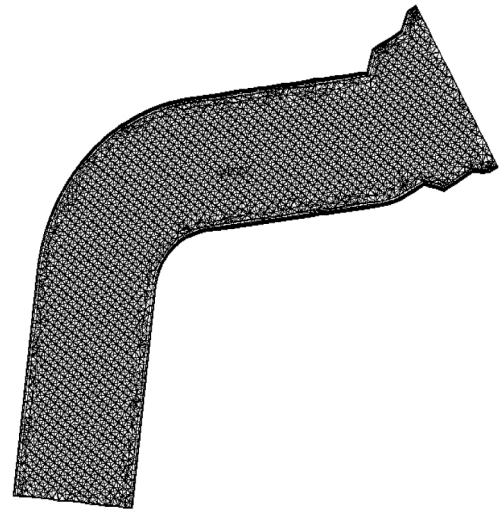


Figure 3. Cross-sectional view of the LES grid (center duct).

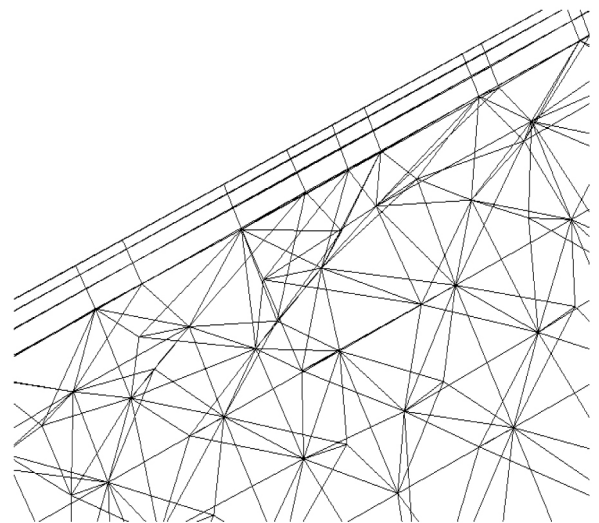


Figure 4. Three layers prism mesh (center duct).

Computation Procedure

A steady state solution is first obtained using RANS with RNG $k - \varepsilon$ closure model and a 2nd order discretization scheme. The steady state data is then used to initialize LES unsteady simulation. During the transient simulation a time-step of 0.003 s is employed, while amplitude and frequency