Applied Acoustics 79 (2014) 23–34

Contents lists available at ScienceDirect

Applied Acoustics

journal homepage: www.elsevier.com/locate/apacoust

Numerical analysis and passive control of a car side window buffeting noise based on Scale-Adaptive Simulation

Yang Zhendong [a,b,*], Gu Zhengqi [a,c], Tu Jiyuan [b], Dong Guangping [a], Wang Yiping [a,d]

[a] State Key Laboratory of Advanced Design and Manufacture for Vehicle Body, Hunan University, Changsha, Hunan 410082, China
[b] School of Aerospace, Mechanical and Manufacturing Engineering, RMIT University, Melbourne, Victoria 3083, Australia
[c] School of Mechanical Engineering, Hunan University of Technology, Zhuzhou, Hunan 412008, China
[d] School of Automobile Engineering, Wuhan University of Technology, Wuhan, Hubei 430070, China

ARTICLE INFO

Article history:
Received 5 January 2013
Received in revised form 19 August 2013
Accepted 5 December 2013

Keywords:
Scale-Adaptive Simulation
Cavity
Side window buffeting noise
Mechanism
Characteristics
Passive control

ABSTRACT

Flow over an open side window in a car exhibits similar characteristics as the flow over an open cavity. Computational Fluid Dynamics (CFD) simulation over a cavity was done as a benchmark. The unsteady flow simulation was carried out using Scale Adaptive Simulation (SAS) turbulence model. The benchmark results, frequency and sound pressure levels of feedback and resonance modes, all well matched with the experimental data. Then, with the right rear window, for example, the mechanism of the side window buffeting was investigated. The simulation results show that side window buffeting noise is generated by large scale vortices and in low frequency. Furthermore, buffeting noise characteristics under several patterns of side windows opening were also numerically investigated. As a result, rear window buffeting noise is more severe than that of front window when one window open, and combination pattern of side windows open can reduce buffeting noise. To decrease the interior noise and improve car ride comfort, four suppression measures through adding a side window weather deflector at the A-pillars, constructing a cavity at the B-pillars, combination of the front and rear windows and installing a row of square cylinder deflector at the B-pillars were also studied, respectively. In conclusion, certain noise reduction can be achieved through four passive control methods.

1. Introduction

Wind buffeting, the noise and pulsating forces that are experienced when driving a car with the side windows open, has received a great deal of attention in recent years.

Since the entire passenger compartment acts as a cavity when the sunroof or one of the windows is open, buffeting can be classified as cavity noise. Most early research work on flow over a cavity was done in aeronautical industry [1–4]. Rossiter and Britain [5] measured the time average and unsteady pressures acting on the roof and behind a series of rectangular cavities found that the unsteady pressures contain both random and periodic components. In the sixties last century, the wind buffeting named as “aerodynamic wind throb” was studied analytically and experimentally by Bodger and Jones [6]. They point out that “aerodynamic wind throb” is noticeable over a range of speeds. They also shown that the buffeting noise can be weaken by changing the natural frequency, minimizing or eliminating the excitation, or increasing damping of the entire passenger compartment. Since 1990s, cavity noise as a major source of automobile aerodynamic noise has received an attention. Henderson [7,8] presented benchmark experimental data of a cavity flow at the door gap of a road vehicle with a slot and flow of 45–60 m/s in the 3rd and 4th computational aeroacoustics (CAA) workshop of NASA, respectively. Cavity noise is caused by an unstable shear layer established at the upstream edge of the cavity [9]. Disturbances are shed from the front edge of the opening and are converted along the flow. When they impinge on the rear edge of the opening, a pressure wave is generated that propagates inside as well as outside the cavity (passenger compartment). When the wave reaches the front edge of the opening, it triggers another disturbance shedding. This process occurs periodically and causes the shear layer to generate a specific buffeting frequency. This frequency depends on the speed of the vehicle and the geometry of the opening [10]. For automobiles, this frequency usually is very low (<20 Hz). Human ears cannot detect such a low frequency, but it is felt by the passengers as a pulsating wind force which can be very fatiguing. Therefore, it is important to consider aerodynamic buffeting in automobile design from the point of view of passenger comfort [11]. Zhu and Gleason [12] introduced a process of prediction of side window buffeting at a very early program stage. The process includes CFD simulations, a full scale

* Corresponding author at: State Key Laboratory of Advanced Design and Manufacture for Vehicle Body, Hunan University, Changsha, Hunan 410082, China. Tel./fax: +86 73188823055.
E-mail address: yzdrly@hnu.edu.cn (Z. Yang).

0003-682X/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.apacoust.2013.12.006

aero-acoustic buck and pilot vehicle wind tunnel tests. The compared results show that CFD and the aeroacoustic buck can be used for side window buffeting predictions at early stages of the program. Extensive researches on the wind buffeting noise of the side windows of a passenger car were done [13,14]. The analysis result indicates that there is only 3 Hz difference between the peak frequency obtained by the simulation of the CFD software FLUENT and the test of the wind tunnel, and in a similar way, the difference of the sound pressure level (SPL) is 4 dB.

In recent years, although a lot of research findings on the wind buffeting noise of the vehicle were gained [15–21], but still many crucial problems are imperative to be solved. For example, how to obtain automobile aerodynamic noise sources using CFD technology is a crucial problem. Now, the methods for a prediction of an unsteady flow field, such as unsteady Reynold Averaged Navier–Stokes (URANS), the Large Eddy Simulation (LES) or even the Direct Numerical Simulation (DNS), play a dominant role in CFD modeling. However, automotive applications imply generally high-Reynolds number, low Mach number flows in very complex geometries. The use of DNS is by far the most sophisticated computational methodology whereby all the temporal and length scales of turbulence are resolved. Due to its immense computational cost, it is still currently restricted to simple geometries at low to moderate Reynolds numbers [22,23]. LES approach, proposed by Smagorinsky [24] and refined by many researchers, has been the most widely used to solve unsteady flow problem over the last decades. It is based on the concept of resolving only the large scales of turbulence by filtering the Navier–Stokes equations over a finite spatial region (typically the grid volume) and to model the small scales by a simple eddy viscosity model. However, LES computations are usually performed on numerical grids that are too coarse to resolve the smallest scales. The eddy viscosity is often calibrated to provide the correct amount of dissipation at the LES grid limit. LES is not modeling the influence of unresolved small scale turbulence onto the larger, resolved scales, but the dissipation of turbulence into heat. With LES, if you coarsen your grid, you will get worse and worse results. LES is a fairly simple technology, which does not provide a reliable backbone of modeling [25]. Wall-resolved LES is therefore prohibitively expensive for moderate to high Reynolds numbers [25]. Large-eddy simulations (LES) still suffer from extremely large resources required for the resolution of the near-wall region, especially for high-Re flows [26]. As point out by Iaccarino et al. [27], LES still requires significantly fine spatial and temporal resolution, thus placing a high demand on both computing resources and time, despite the emergence of parallel computing. This is the main reason why LES is not suitable for most engineering flows.

There are some encouraging results using RANS models for unsteady flow predictions [27–29], Iaccarino et al. [27] claims that unsteady RANS (URANS) is highly capable of predicting flows with gross unsteadiness, given that the unsteadiness is deterministic and that the frequency spectrum shows a spike at the vortex shedding frequency. However, URANS simulations only give information on the low-frequency content, and are moreover often ill-posed for separated flows [30]. The Scale-Adaptive Simulation (SAS) is an improved URANS formulation, which allows the resolution of the turbulent spectrum in unstable flow conditions. The SAS turbulence model was first proposed by Menter et al. [31] in 2003. The SAS model has also been used to solve industrial problems, including internal flows, aerodynamics with massive flow separation, and aero-acoustic problems [32]. Belamri et al. [33] used the SAS concept to predict the sound generated by a generic side view car mirror. Younsi et al. [34] applied SAS turbulence model to predict the unsteady flow field behavior in a HVAC forward centrifugal fan. The agreement between simulation values and the experimental data is very satisfactory showing the ability of the SAS model to predict the noise levels. Applications to flow over shallow cavities can be found, for example, Menter et al. [32] used the Scale-Adaptive Simulation model to calculate air flow past a 3-D rectangular shallow cavity. The main acoustic modes are predicted in good agreement with the experiment. However, shallow cavity is a little simple compared to the cavity mentioned in this study. Actually, the vehicle interior passenger compartment should be considered as a deep cavity with a big volume and small opening. According to the authors' best knowledge, there is almost no SAS turbulence modeling studies have been published in vehicle buffeting noise research area.

In order to shorten the period of automobile development in industry, aerodynamic noise should be evaluated in the early stage of concept design. One promising approach is to apply high-fidelity numerical simulation to predict aerodynamic noise of various design options. The aim of this study is to apply SAS modeling approach to vehicle buffeting noise analysis induced by opening windows. In this study, a new computational method based on Scale-Adaptive Simulation was proposed. Firstly, the computational schemes and methodology used for CFD simulation is described including meshing, solver set-up and solution procedure. The reliability of CFD simulations were validated by using a benchmark problem. Then, the mechanism of side window buffeting noise was revealed. Buffeting noise characteristics of several opening patterns were discussed. As a solution, adding side window weather deflector at the A-pillar, was proposed to reduce front side window buffeting noise. In addition, three solutions were used for rear window buffeting reduction: constructing a cavity at the B-pillar, combination of the front and rear window opening and stalling a row of square cylinder spoiler at the B-pillar. This paper can provide valuable contributions for the research on this subject.

2. Computational schemes

The SAS concept is based on the introduction of the von Karman length-scale into the turbulence scale equation. The information provided by the von Karman length-scale allows SAS model to dynamically adjust to resolved structures in a URANS simulation, which results in a LES-like behavior in unsteady regions of the flow field. At the same time, the model provides standard RANS capabilities in stable flow regions. SAS method provides "LES" level of flow-field capture in unsteady regions at less than half the cost of LES model.

The difference between standard RANS and SAS models lies in the treatment of the scale-defining equation (typically ε-, ω, L_t – equation) [35,36]. In classic RANS models, the scale equation is modeled based on an analogy with the k-equation using simple dimensional arguments. The scale equation of SAS models is based on an exact transport equation for the turbulence length scale as proposed by Rotta [37]. This method was revisited by Menter and Egorov [35] and avoids some limitations of the original Rotta mode. As a result of this re-formulation, it was shown that the second derivative of the velocity field needs to be included in the source terms of the scale equation. The original SAS model was formulated as a two-equation model, with the variable $\Phi = \sqrt{k}L_t$ for the scale equation:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_j}(\rho \overline{U}_j k) = P_k - \rho c_u^{3/4}\frac{k}{\Phi^2} + \frac{\partial}{\partial x_j}\left(\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right) \tag{1}$$

$$\frac{\partial \rho\Phi}{\partial t} + \frac{\partial}{\partial x_j}(\rho \overline{U}_j \Phi) = \frac{\Phi}{k}P_k\left(\zeta_1 - \zeta_2\left(\frac{L_t}{L_{\upsilon K}}\right)^2\right) + \frac{\partial}{\partial x_j}\left(\frac{\mu_t}{\sigma_\omega}\frac{\partial \Phi}{\partial x_j}\right) - \zeta_3 \rho k \tag{2}$$

where $u_t = c_u^{1/4}\rho\Phi$ is turbulence (or eddy) viscosity, c_u is a constant, $P_k = u_t S^2$ is turbulence production term, $L_{\upsilon K} = \kappa\left|\frac{\overline{U'}}{\overline{U''}}\right|$ is von Karman