

Evaluation of Aerodynamic Noise Generated in a Miniature Car Using Numerical Simulation

Gu Zhengqi, Wang Yiping and Li Weiping
Hunan University

Yangxue
National University of Defense Technology

Copyright © 2009 SAE International

ABSTRACT

Aerodynamic noise generated in a miniature car had been evaluated using numerical simulation. Large Eddy Simulation (LES) was applied to analyze the transient flow field and the Ffowcs Williams-Hawkings (FW-H) acoustic analogy was employed to conduct acoustic analysis. The time accurate flow data was obtained using a finite volume flow solver on an unstructured grid. The flow field around the rear view mirror was obtained by numerical for two cases with different side view mirrors. Moreover, the distribution of acoustic source was predicted on side windows, and the aerodynamic noise was lowered through optimizing the shape of the rear view mirror and some experiments were done to validate the effect. Present study ascertained the feasibility and applicability of finite volume method (FVM) with SGS model towards prediction of aerodynamic noise generated in production vehicle.

INTRODUCTION

The noise emission from cruising vehicles essentially consists of tire/road noise, drivetrains noise (engine with in take and exhaust system, transmission and driving axle) and aerodynamic noise due to the flow around bodywork, chassis wheels and cooling air flow (fan). Engines and drivetrains had become quieter due to many man-years of engineer attention and tire noise had also been reduced - at least the noise reaching the passenger compartment. Consequently, the

aerodynamic noise of ground vehicles has become dominant at driving speed above 100 km/h both in interior and exterior noise.

Many researchers had revealed that the primary source of aerodynamic noise is due to the flow around front-pillar and mirror region. The flow is separated at the region and reattaches on the side window surface with strong rotating motion. As a result, the high fluctuating pressure occurs around side window, which causes the in-cabin aerodynamic noise. So far, the ordinary methods to evaluate aerodynamic noise are wind tunnel experiment and road test by using prototype.

However, to shorten the development period of vehicle, it is requested that the aerodynamic noise can be evaluated in the stage of prototype not existing. The most efficiency tool to realize this aim is applying numerical simulation to predict aerodynamic noise, and Ogawa [1] presented a review of the use of CFD in simulating road vehicle acoustics.

Kumarasamy et al. [2] had conducted the computational and experimental study to investigate the aeroacoustics of an automobile front-pillar rain gutter. Reynolds Averaged Navier-Stokes (RANS) equations had been applied to obtain the flow solution, and the far-field sound was computed using Lighthill-Curle equation. The computed results including pressure and sound intensity were in good agreement with the experimental values

YeLi et al. [3] had applied Finite difference method (FDM) and finite element method (FEM) to analyze the flow field, and Lighthill's analogy was employed to conduct acoustic analysis. The flow fields around front-pillar obtained by numerical simulations agreed with those by experiment for two cases with different front-pillar shape. Moreover, the distribution of acoustic source predicted by FEM was consistent with that obtained by experiment.

Kenneth J. Karbon et al. [4] presented a study of roof rack wind noise using commercial Computational Fluid Dynamics (CFD) software. The focus was to predict the noise generated from the roof rack cross bars mounted on a realistic vehicle geometry. Results from the CFD simulations include frequency spectra of Sound Pressure Level (SPL) for comparison to typical wind tunnel measurements. The CFD and physical experiments compare very well with respect to tonal noise generation, tonal frequency content, and relative magnitudes.

Hold et al. [5] and Siegert et al. [6] had conducted computational and experimental study to investigate the aeroacoustics of a Generic Side Mirror. At first, in order to yield the unsteady surface pressure distribution, the overall numerical simulation of the Navier-Stokes solver was used to calculate a time accurate solution of the flow field around the mirror, and in a second step this surface pressure source strength was used as input for two different acoustic tools predicting the far-field radiated sound signals. The computed results including pressure and sound intensity were in good agreement with the experimental values. However, in their study, the test model was quite simple and placed on a plate, which was different from the production vehicle.

In this study, numerical simulation using production vehicle had been performed to investigate the mechanism of aerodynamic noise, and road experiment was used to test the aerodynamic noise. At the same time, effects of mirror shape on aerodynamic noise will be investigated. In the last, by comparing the simulation and experiment results, we will investigate the possibility of applying numerical simulation toward prediction of aerodynamic noise generated from production vehicle.

MAIN SECTION

METHODOLOGY

A general purpose CFD solver can be used to conduct aeroacoustics simulations in two ways. The first approach is to simply conduct a spatially and temporally well resolved transient simulation of the fluid domain that encompasses the acoustic sources as well as the receivers. The acoustic wave equation is a special case of the Navier-Stokes and continuity equations which are

solved by a general purpose CFD code. Hence, sound generation and propagation are automatically contained in the transient flow simulation conducted by a CFD solver. However, the sound frequency and amplitude range that can be predicted by this approach is severely restricted by spatial and temporal resolution (mesh resolution and time-step size) used in such simulations. The reasons for such restrictions are well-known and well documented in a variety of publications, e.g. [7]. Never-the-less, this approach is practically viable for simulating acoustics in situations where the geometric dimensions of the domain are relative small and frequencies of interest are low. However in this paper the geometric is quite complexity and computing frequency is 5000Hz, this approach is almost impossible.

The second approach is to conduct a spatially and temporally well-resolved CFD simulation only around the acoustic sources and to use an acoustic analogy to calculate sound at the receivers from the time-varying stresses recorded in the simulation at the sources. This approach is particularly useful for predicting noise in the far-field as it avoids the need to extend a fine mesh all the way up to the receiver as required in the first approach. In the subsequent discussion we will refer to this approach at the Acoustic Analogy (AA) approach. In the present study we have tested this approach using the acoustic analogy of Ffowcs-Williams and Hawkins [11].

In the current work the commercial CFD codes Fluent6.3 has been used for the second approach listed above. Fluent solves transient Navier-Stokes, continuity and energy equations using the finite volume method [9, 10, 11]. In the rest of this section we discuss the meshing considerations, turbulence models, boundary conditions, implementation of the Ffowcs-Williams-Hawkins acoustic analogy and the solution procedure for the CFD simulations.

VEHICLE GEOMETRY

The vehicle model used in the present study resembles a modern miniature car, and the geometry of the vehicle was created in a computer aided design (CAD) software package and was exported to ICEM, a commercial geometry and mesh building tool, in the IGES format. The model geometry had been purposefully made generic by discarding custom details such as tyres, rain wiper, doorknob, antenna and underbody components. Fig.1 shows the car model, and the length of the model is 4018mm, the width is 800mm, the height is 1860mm. In order to investigate the effects of side view mirror geometry on aerodynamic noise, tow models with different bottom curvature were used in this study, which were showed in Fig.1. As shown in figure, the bottom curvature in Modified Model is larger than that in Original Model.

Fig.2 shows the placement of the vehicle in the virtual wind tunnel .The computational domain comprises of the car kept in a rectangle box, the inlet of the domain is roughly 3 vehicle lengths upstream of the front tip of the model while the outlet is roughly 7 vehicle lengths downstream of the rear end of the model, and the height of the domain is roughly 5 vehicle heights, and the width of the domain is roughly 5 vehicle widths. In order to analysis the noise, the received point behind the wake of the left mirror and specific coordinate is (-200mm,-760mm, 847mm) (Fig.3).

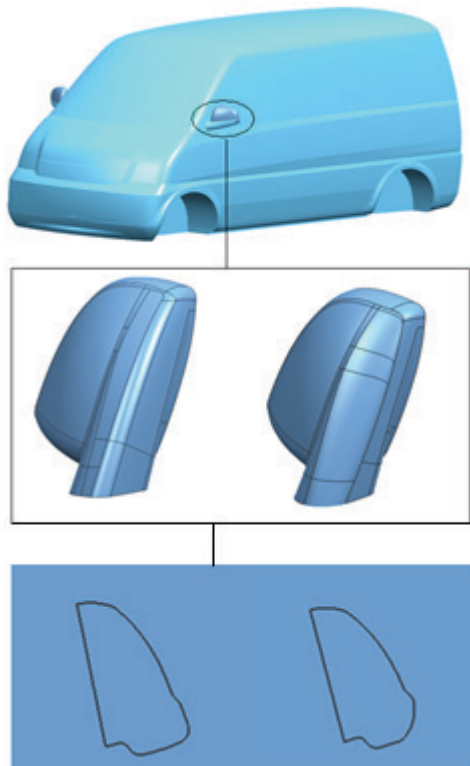


Fig. 1 Research objects and definition of two mirror models used in this study

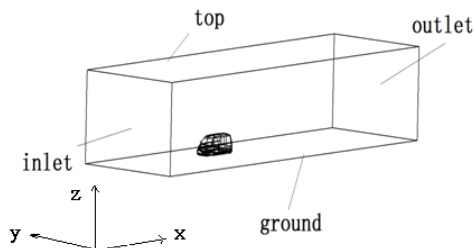


Fig. 2 Computational domain

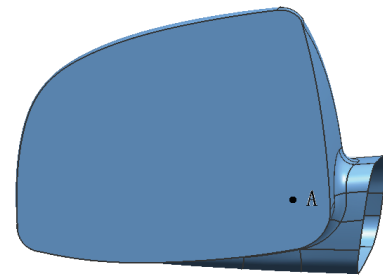


Fig.3 The position of received point

MESHING

Triangular surface meshes replicate complex three dimensional shapes more accurately than meshes with quadrilaterals or multi-sided polygons, hence a triangular surface mesh was used in the present work, the flow separation around Pillar-A and side view mirror is quite strong, and the side windows is the main noise source region, so very elaborate mesh was used on these regions. Delaunay method was used to generate volume mesh. In order to assure the quality of element, elaborate prism mesh was used on the automotive surface, and a total of three prism layers are grown. Their heights increase successively by a factor of 1.2 and the aspect ratio of all cells in the first prism layer is maintained below 4.

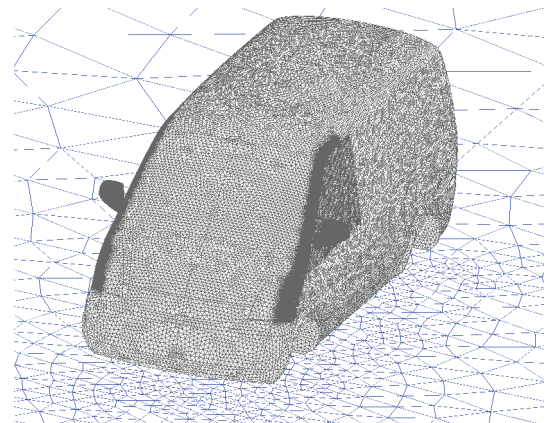


Fig.4 Surface meshes on the vehicle.

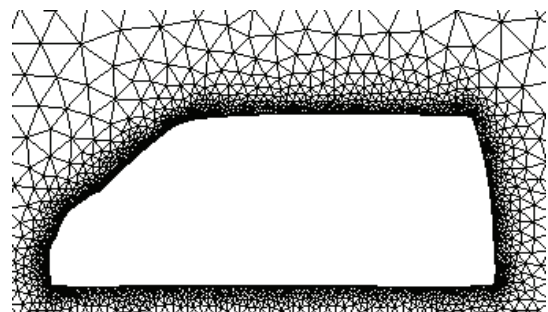


Fig.5 Cross-sectional views of the mesh in the vicinity of vehicle, along the symmetry plane