A COMPUTATIONAL STUDY OF AERODYNAMIC NOISE ALLEVIATION ON A GENERIC CAR MODEL

Dung Hoang Thi Kim[1] and Khanh Nguyen Phu[2]

[1] Department of Aeronautical and Space Engineering, School of Transportation Engineering, Hanoi University of Science and Technology, Hanoi, Vietnam, e-mail: dunghtk.dase@gmail.com, dung.hoangthikim@hust.edu.vn

[2] Department of Aeronautical and Space Engineering, School of Transportation Engineering, Hanoi University of Science and Technology, Hanoi, Vietnam, e-mail: khanh.nguyenphu@hust.edu.vn

Received Date: January 30, 2015

Abstract

The aerodynamic noise is referred to the noise generated by unsteady flow, which the turbulent flow is connected to the noise sources. In this paper, the aerodynamic noise on a generic car was carried out by using numerical method. First, the turbulent flow on and around the car, when the car moved at velocity 100 km/h, was found out by using CFD method in ANSSYS FLUENT software. Then, Fast Fourier Transform tools in ANSYS FLUENT software was applied to estimate the sound pressure level from the fluctuation of pressure on and around the car. With the aim of alleviating the aerodynamic noise on the cars, four different car shapes were studied. The results remarked that the worse car shape had, the more complex turbulent pressure was found and the higher aerodynamic noise was found as well.

Keywords: Aerodynamic car, Aerodynamic noise, ANSYS FLUENT, Fast fourier transform

Introduction

Movement of cars generated many different types of noise, such as engine noise, aerodynamic noise, noise generated by the friction between wheel and road. The aerodynamic noise was considered here only in this paper. The aerodynamic noise refers to the noise generated by unsteady flow. It is also called aeroacoustics or flow noise. The noise sources can be connected with a turbulent flow [1-5].

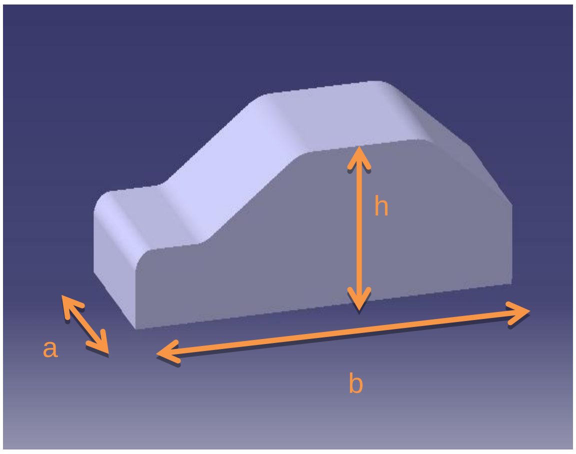
In the present paper, the aerodynamic noise on a basic car model was first employed at velocity of 100km/h (27.8 m/s), then three next car models were studied with purpose of alleviating aerodynamic noise. To carry out the aerodynamic noise, two main problems needed to solve: one was aerodynamic problem and other was acoustic problem. The aerodynamic problem estimated the distribution of pressure, velocity … on and around the car and also estimated the aerodynamic characteristics of car such as lift force and drag force. This problem was solved by using CFD tool in ANSYS FLUENT software. After the aerodynamic problem was solved, Fast Fourier Transform (FFT) tool in ANSYS FLUENT software was applied to calculate the sound pressure level (SPL) of acoustic problem.

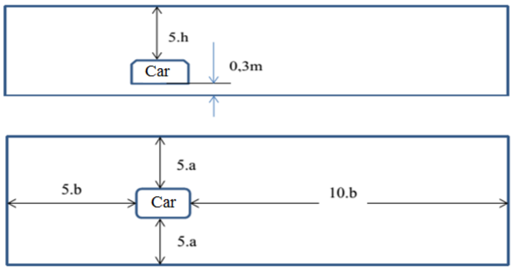
In this study, the car was studied at velocity of 100 km/h (27.8 m/s). With the aim of alleviating the aerodynamic noise on the cars, four models of cars were put into consideration. The main idea of changing the car shape was found out in Hucho and Sovran [5].

Numerical Setup

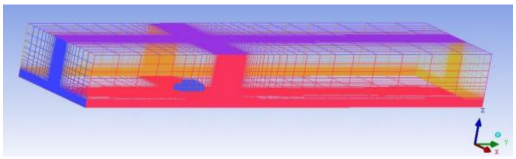
Basic Car Model

The basic car had a simple model as shown in Figure 1a. Dimensions of the car were: height h = 1.4 m, width a = 1.6 m and length b = 4.4 m.





b. Computational domain



a. Basic car model

c. Computational grid

Figure 1. Numerical setup

Numerical Setup

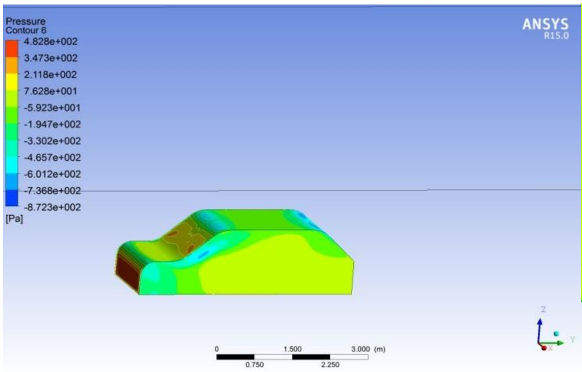
ANSYS FLUENT software was used to simulate the aerodynamic problem and noise problem. The proceeds were included five basic steps below:

- **Identify Computational Domain*:*** From the car model (Figure 1a), computational domain was identified as shown in Figure 1b [1, 2],
- **Mesh Computational Domain*:*** Computational mesh was composed of $0.823*10^6$ structured nodes (Figure 1c).
- **Set Up Numerical Conditions*:***
  - Turbulent model: Shear Stress Transport (SST) k-w turbulent model was chosen due to its good behavior in adverse pressure gradients and separating flow [1].
  - Boundary conditions:
    - Velocity inlet: 27.8 m/s,
    - Outlet pressure: ambient pressure.
  - Transient problem: Due to the aim of research for the aerodynamic noise, transient problem was chosen with:
    - Time step: 5e-04 s [2],
    - Number of time step: 2000.
- **Solve Occurring Problems*:*** There were two problems needed to solve in parallel:
  - Aerodynamic problem: Distribution of pressure, velocity and streamline were carried out using CFD tool in ANSYS FLUENT software.

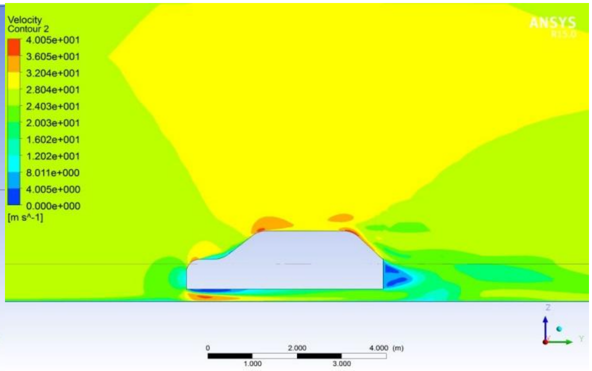
- Aerodynamic noise problem: From the results of aerodynamic problems, special transient pressure, FFT tools in ANSYS FLUENT were used to transform the aerodynamic results to aerodynamic noise results.
- **Analyze the Results*:*** The results were presented in the next part.

Results

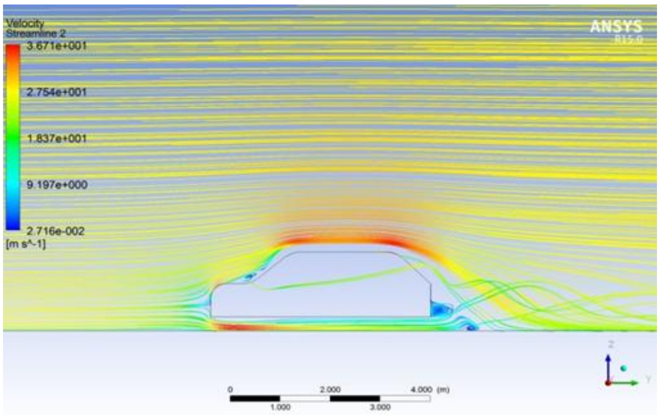
Aero-Acoustic Properties



a. 3D pressure



b. 2D pressure



c. Streamline

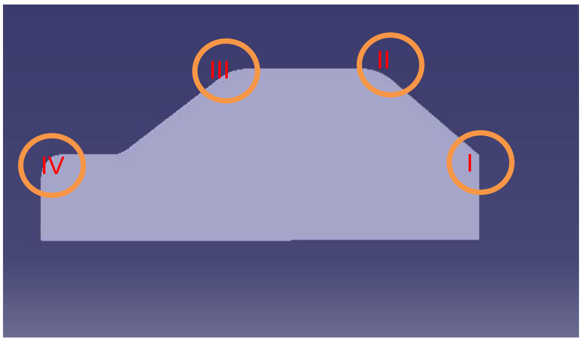
Figure 2. Aerodynamic analysis

The aerodynamic flow of basic car was found in Figure 2. At the nose, inflection points or two sides of the car, the pressure was high and suddenly changed (Figure 2a, b). This sudden change of pressure was remarked as the main cause of bad aerodynamic quality on the body [2, 5].

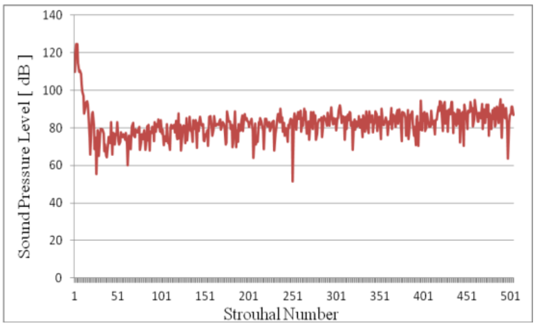
Behind the car, at the trailing car, high upper-pressure combined with low lower-pressure caused the apparition of vortices (Figure 2c). The aerodynamic characteristics of car were 0.71 for CD – drag coefficient and 0.18 for CL – lift coefficient. Both the lift and drag coefficients of basic model were high. However, following [5], high lift coefficient at high moving speed would cause a loss of traction between the wheels and road surface. This phenomenon would be very dangerous. It would lead to overturn the car. So, this model needed to be improved.

The acoustic analysis was estimated at four different positions where the pressure was suddenly changed (Figure 2b). These four positions were found as the positions which could improve the shape of the car [5]. The coordinates of these points were: Point I (0.8; 2.2; 0), Point II (0.8; 0.6; 0.7), Point III (0.8; -0.3; 0.7) and Point IV (0.8; -2.2; 0) (noted in Figure 3 with the unit was m).

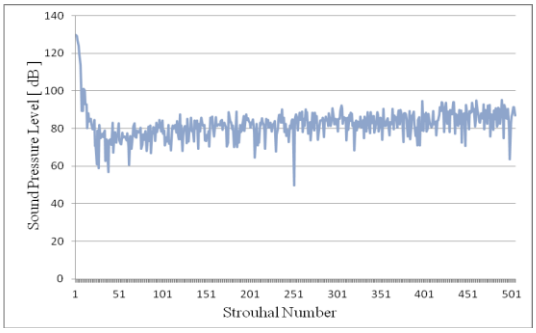
The fluctuation pressures at these points were transformed as sound pressure level (SPL) by using Fast Fourier Transform (FFT) in post-ANSYS FLUENT software. The SPL of these points were found in the range of 49 – 95dB. These values were in the threshold of human's ear. This was the main reason why the aerodynamic noise needed to be reduced as noted at the aim of this paper.



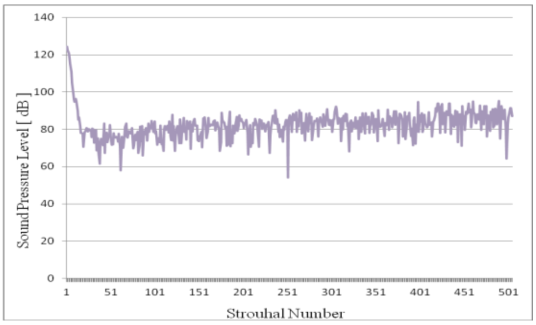
a. Acoustic survey points



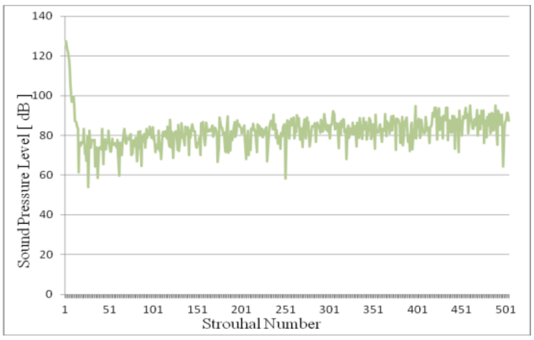
b. Point I (0.8; 2.2; 0)



c. Point II (0.8; 0.6; 0.7)



d. Point III (0.8; -0.3; 0.7)



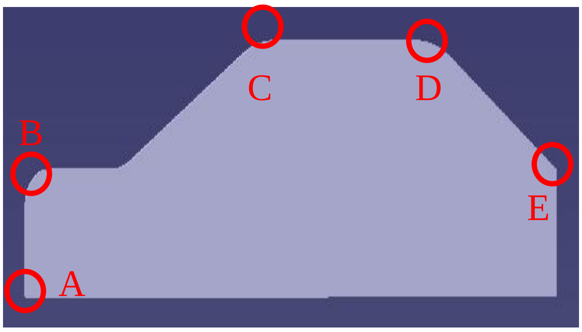
e. Point IV (0.8; -2.2; 0)

Figure 3. Acoustic analysis

Improvement of the Aerodynamic Noise

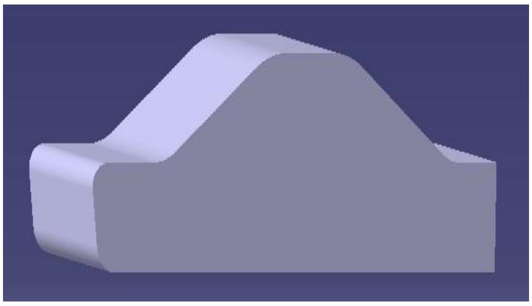
Figure 4a showed four positions that could optimize the shape of the car as remarked by Hucho and Sovran [5]. Four different car models, including basic model, were chosen and presented in Figure 4b-e. Model 1 was basic model (Figure 4b). Model 2 was the first change of only the rear car (E, Figure 4c). Model 3 included the change of tip (A, B) and rear (E) car (Figure 4d). Finally, model 4 included all the four positions: tip (A, B), top (C, D) and rear (E) car (Figure 4e).

With the changes of car model from model 1 to model 4, the distribution of pressure seemed to be more and more continuous (Figure 5). The streamlines were also monotone except the sides and trailing points (Figure 6). The aerodynamic characteristics (drag and lift force coefficients) of car were decreased when the body shape was improved (Table 1).



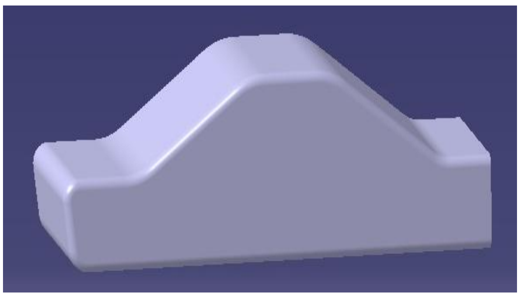
a. Improved points

b. Model 1 – Basic model



d. Model 3 – A+B+E

c. Model 2 - E



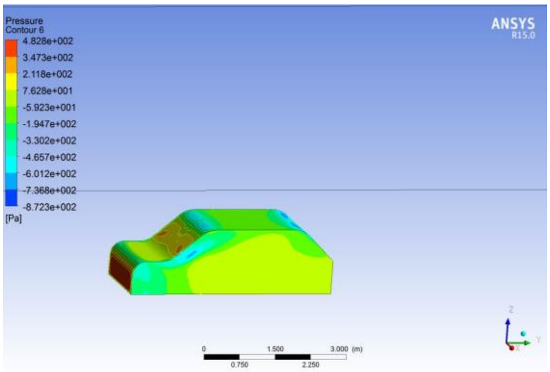
e. Model 4 – A+B+C+D+E

Figure 4. Improved the car model

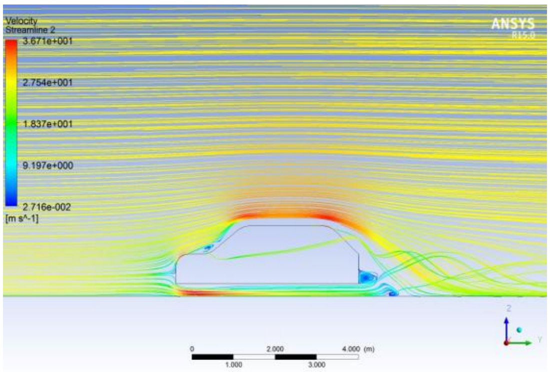
Table 1. Aero-Acoustic Properties

Model	C_D	C_L	SPL (dB)
Model 1	0,71	0,18	49 - 95
Model 2	0,49	0,04	42 - 87
Model 3	0,40	0,05	38 - 87
Model 4	0,30	0,036	37 - 81

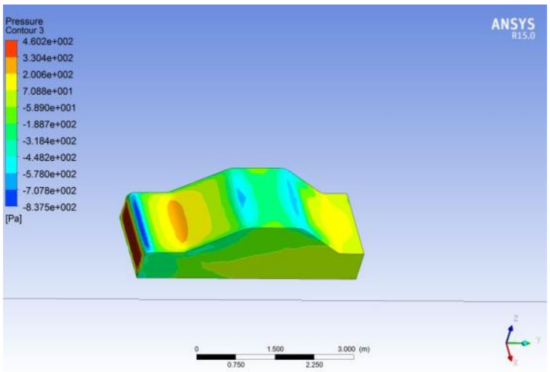
The SPL, at these four positions, was estimated by using FFT tool in ANSYS FLUENT software and presented in Figure 7. The range of SPL for model 2, 3 and 4 became lower than that of the basic model (Table 1). The model 4 was remarked as the best shape, which had the lowest drag, lift coefficient and SPL, same remark with Hucho and Sorvan [5].



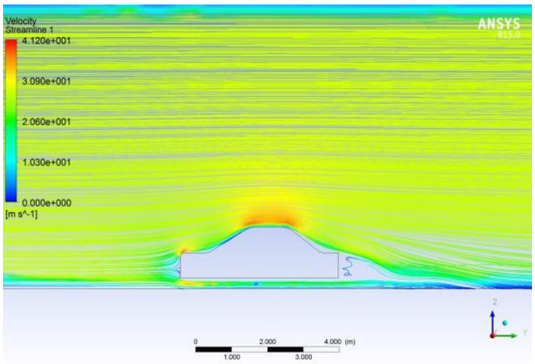
a. Model 1



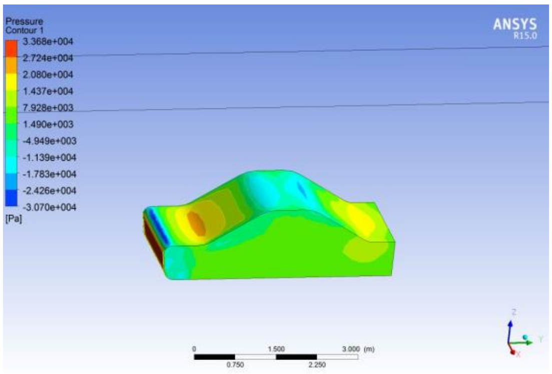
a. Model 1



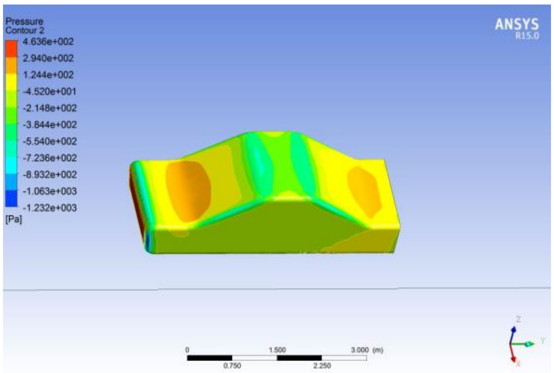
b. Model 2



b. Model 2

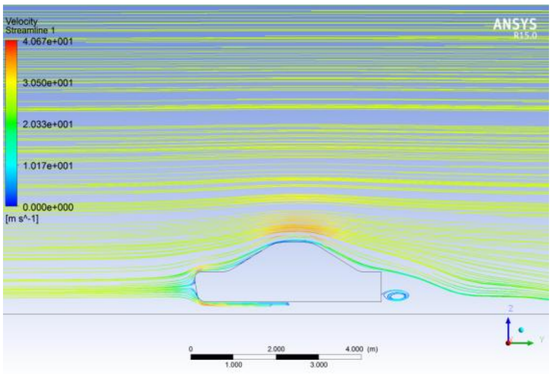


c. Model 3

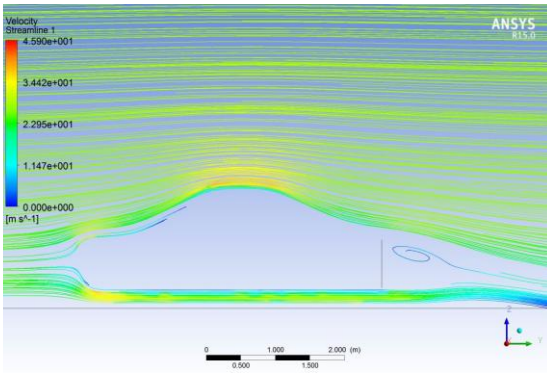


d. Model 4

Figure 5. Pressure distribution of improved model

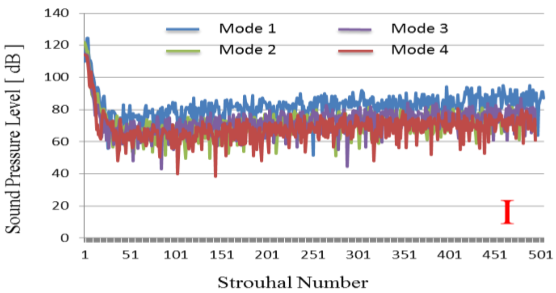


c. Model 3

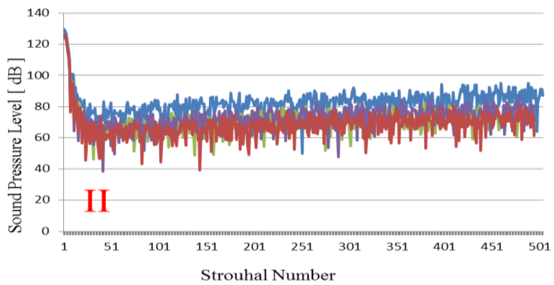


d. Model 4

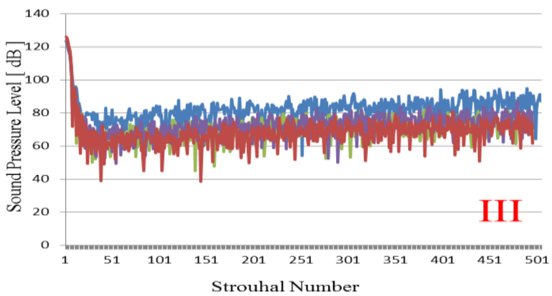
Figure 6. Streamline flow of improved model



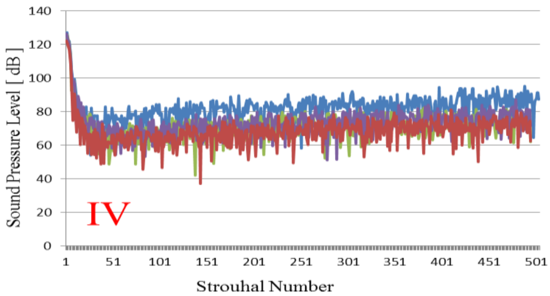
a. Point I (0.8; 2.2; 0)



b. Point II (0.8; 0.6; 0.7)



c. Point III (0.8; -0.3; 0.7)



d. Point IV (0.8; -2.2; 0)

Figure 7. Acoustic flow of improved model

Conclusions

The aerodynamic noise problem was successfully solved by using ANSYS FLUENT software. The major results were:

- Aerodynamic noise linked with the aerodynamic characteristics of car,
- More improvement for the aerodynamic characteristics of car (reduced both lift coefficient and force coefficient), more reduction for the aerodynamic noise properties of car.

It is necessary to compare these numerical results with other corresponded results such as: experimental results or other numerical results in the future.

Acknowledgements

This work was supported by bilateral Project No. 10/2014/HĐ-NĐT of Ministry of Science and Technology in Vietnam and CRI Project 06/CRI 2014-2016 of AUNSEED/Net JICA - Japan.

References

[1] J. Liu, *Simulation of Whistle Noise Using Computational Fluid Dynamics and Acoustic Finite Element Simulation*, Thesis(Master's), University of Kentucky, Lexington, United States, 2012.

[2] C. Domenico, *Industrial CFD Simulation of Aerodynamic Noise*, Thesis(PhD), Napoli Federico II University, Italy, 2008.

[3] S. Uosukainen, "Foundations of acoustic analogies," *VTT Publication* 757, pp. 1-34, 2011.

[4] S. Itagaki, *Estimation of Aircraft Noise*, Report of Mitsubishi Heavy Industries, 2013.

[5] W.H. Hucho, and G. Sovran, "Aerodynamics of road vehicles," *Annual Review of Fluid Mechanics*, Vol. 25, pp. 485-537, 1993.