A Qualitative Study of the Wind Noise Cavity Inside of a Two Way Radio

W M Hairudin1, N M Kamarudin1, L E Ooi2, N A Ismail1*

1School of Aerospace Engineering, Engineering Campus, Universiti Sains Malaysia, 14300 Nibong Tebal, Pulau Pinang, Malaysia.
2School of Mechanical Engineering, Engineering Campus, Universiti Sains Malaysia, 14300 Nibong Tebal, Pulau Pinang, Malaysia.

*Corresponding author: aenorilmi@usm.my

Abstract. The cavity inside microphone of two-way radio can induced noise when involved in outdoor and windy condition. An investigation was performed in Viwika wind tunnel involving a flow visualizations techniques to clarify the flow noise behavior generated by the wind, particularly within the cavity. A smoke wire test was conducted using a cavity model at varying aspect ratio (L/D) between 0.7 and 3 in angle of attack of 0º with Reynolds number of 7831. A two-dimensional Computational Fluid Dynamic (CFD) models based on k-Epsilon turbulence model were conducted to validate the flow visualization result. The results revealed that the deep cavity with an aspect ratio (L/D) of 0.7 produced a single clockwise vortical flow structure and the aspect ratio (L/D) of 3 show the complex flow structure which two recirculation flow vortices were produced inside the cavity. The flow behavior in the CFD simulation was similar to the smoke test. It is shown the flow behaviour inside cavities was control by the aspect ratio (L/D).

1. Introduction

Wind noise is known as one of the issues in mobile telecommunication such as two-way radio. In this application, communications involve the use of a microphone inside of two-way radio. When two-way radio are used in outdoor activities and windy condition, wind noise interferes with the signals, particularly in the low-frequency range. The wind noise existed due to the wind that passes the microphone’s cavity structure and vibrating the diaphragm, creating a signal that equal to or greater than the voice receiver. High wind flow affects the quality of the signal receiver due to the presence of turbulent flow over the microphone’s cavity and contaminates the desired output signal. Thus, it severely degrades the quality and intelligibility of the speech signal [1].

The phenomenon of wind noise has been studied by numerous applications in the past such as in landing gear [2], car sunroof [3], vehicles [4-5], high speed train (pantograph) [6 -7], building [8-9], closed side branches of pipelines [10], wind turbines [11], submarine hulls [12], musical instruments [13] and microphones [14-15]. The microphone’s cavity is a typical kind of structure that can induced noise. The flow cavity noise rises from the feedback mechanism between the shear layer instabilities and acoustic waves that cause an aeroacoustic resonance. The aeroacoustic resonance also leads to fatigue and vibration in structures. The aeroacoustic cavity resonant can be divided into three groups; (1) Rossiter resonance which includes the high-speed Mach flow over the shallow cavities (L/D >1) [2], (2) the cavity resonance by Helmholtz resonator or deep cavity or side branch [16-17] and (3) the acoustic resonance inside the corrugated or coaxial pipe [18-20]. The unsteady oscillations of the air often caused
due to disturbance, such as sharp edges of a rigid structure causing turbulence or vortices. The turbulence and vortices produced when the large pressure fluctuates by the impinging a shear layer on the leading edge (A) of the cavity. The shear layer flow is separated from the leading edge of the cavity and breaks down into downstream, and the emerging vortex shedding impinging on the trailing edge (B) results in pressure waves. The pressure waves lead to a self-sustained oscillation, inducing acoustic noise as shown in Figure 1. The self-sustained oscillations flow over a cavity can radiate tonal noise as a cavity tone.

One of the parameters that influence the flow behaviour and oscillation modes within the cavity is cavity geometry which includes an aspect ratio (L/D). Yilmaz et al. [21] conducted a 2D simulation in various L/D in ANSYS Fluent. The result shows that by increasing the L/D, the flow field within cavities become complex and have multiple modes. In L/D of 1, there is no pressure oscillation due to the less energy in shear layer. Yoshida et al. [22] investigated the flow field and oscillation mode of varied cavity length to depth ratio (L/D). The result revealed that flow field is influenced by the relationship of cavity shear layer oscillating mode and recirculating vortices within the cavity. The oscillating mode can be switch into three groups which includes non-oscillating mode, shear layer and wake mode.

![Figure 1: Mechanism of cavity flow](image)

The wind induced noise inside two-way radio has been studied by several researchers. Fisol et al. [23] has investigated the effect of the flow speed and the angle of attack with noise level in open wind tunnel. They concluded that the noise level is increased when flow speed and angle of attack increases. Another work by [24] performed a sound measurement in term of threshold hearing level intelligibility, angle of orientation and wind speed of the signal receiver. The results have shown that the angle leads to worst wind-induced noise effect is at 90° to the leading edge, which is showing the highest sound pressure level. Yow [ ] performed CFD simulation and validate the result with actual prototype model. They result shown that. Based on this review, not much attention into flow behaviour and acoustic analysis were taken into the cavity of two-way radio. Thus, it is important to determine the flow behaviour that causes the noise inside the cavity and clarify the method to suppress the cavity resonant.

In this paper, the flow behavior is focused on the open cavity inside the two-way radio with a different aspect ratio of (L/D) were qualitatively studied by smoke visualization technique and compared with CFD simulation. The research results provides the reference for reducing the noisy speech signal for the two-way radio during windy environments.
2. Flow visualisation technique

2.1 Experimental set-up
Experiments of flow visualization were performed in small scale closed loop wind tunnel. The wind tunnel has a dimensions of 0.2m (width) x 0.4m (height) and 0.8m (length). The maximum achievable speed is 11 m/s.

2.2 Cavity model
Two cavity models with different L/D were designed in Solid work and and printed using in house 3D Expresso printer at the Space System Laboratory at Universiti Sains Malaysia. The cavities are open cavity with different aspect ratio of L/D = 0.7 and 3 as shown in Figure 2. The depth was kept fixed. All cavities model were printed using black PLA material to reduce the light reflection during experiment.

![Figure 2: 3D printed test model with two cavities at length to depth ratio (a) L/D = 0.7 (b) L/D = 3](image)

2.3 Experimental procedure
A picture of wind tunnel set up is shown in Figure 3(a). The cavity model was mounted on the axis in the centre of the measurement section. The axis has an outer diameter of 6.2mm and the alignment bolt is 2mm in diameter. The different model cavity of length to depth ratio (L/D) of 0.7 and 3 were tested with Reynolds number of 7831 corresponding to the speed of 0.8m/s at angle of 0°. A video camera with 30 frames per second (fps) was located outside the test section door to capture the snapshots of the flow. All the images were recorded and analyzed. However, in this paper only clear image of vortex flow is illustrated. The door test section was covered with black cloth to avoid any reflection. The velocities at the inlet and inside the test section was measured by a portable digital hot wire anemometer.

In this experiment, water was used as a working fluid to produce the smoke line. The water was installed in the tank up to a level within the Min - Max indicators as shown in figure 3(b). The waterproof ultrasonic vaporizer was used to generate the fog in the wind tunnel. An ultrasonic vaporizer uses a ceramic diaphragm vibrating at an ultrasonic frequency to create water droplets that exit the fog inlet in the form of cool fog. The fog then gets forced out by mini fan blower that placed upper the working fluid tank. The fog flows to the tunnel and started to flow along smoke inlet by using fan blower. The fan blower was used to reduce the turbulence and straighten the flow only in axial direction. A heat grid (hot wire) generates the streamlines in the test section.
3. Computational Fluid Dynamic (CFD)

The commercial CFD software Ansys – FLUENT 16.0 [23] was used to simulate the flow field inside the small cavity. The simulation was based on the 2-dimensional (2D) using Reynolds Averaged Navier-Stokes (RANS) equation approach. A k-epsilon turbulence model was chosen to simulate the flow characteristics due to the shear layer, recirculating flow and small pressure gradients as reported by [25]. A semi-implicit method for pressure-linked equations (SIMPLE) algorithm of the pressure-velocity coupling was selected to solve the governing equations for low speed incompressible flow. The flow conditions were specified as \( \text{Re} = 7831 \) corresponding to \( U = 0.8 \text{ m/s} \). Table 1 shows the condition for the simulation parameter of the cavity model. The CFD structured meshes of each cavity shown in figure 4 meanwhile the nodes and element were summarized in Table 2. The computational time step of the transient flow field was 0.01s at number of step size of 350. Simulation parameters were tabulated in Table 1. Figure 5 shows the rake location point for velocity profile in ANSYS Fluent.
Table 1. Simulation parameters of the cavity model.

| Simulation parameter        | Value   |
|-----------------------------|---------|
| Air density, $p$ (kg/m$^3$) | 1.225   |
| Kinematic viscosity, $\nu$  | $1.7 \times 10^{-5}$ |
| Pressure, $P$ (Pa)          | 101325  |
| Time step size (s)          | 0.01    |

Table 2. Nodes and elements of each CFD meshes.

| Cavity model      | Nodes | Elements |
|-------------------|-------|----------|
| Cavity model (L/D= 0.7) | 41860 | 41371    |
| Cavity model (L/D = 3)   | 57885 | 57324    |

Figure 5: The rake point for velocity value in CFD simulation

4. Mesh Independence Study
The effect of the mesh size can be seen in Figure 4. In this figure, the blab la show close agreement with each other. The coarse mesh influence the ability to resolve the

5. Result and Discussion
In this section the flow behaviour inside cavity were presented. The effect of different of aspect ratio (L/D) were discussed details in term of the velocity contour.

5.1 Effect of aspect ratio (L/D)
The analysis on the experiment and computational results indicate that some vortex generates from flow separation in the shear layer inside the cavity as shown in Figure 6. The flow structures of the effect of aspect ratio (L/D) are presented in Figure 6(a) and (b). From the figures, it can be seen as the aspect ratio (L/D) increase, the flow structure inside cavity also affected. In aspect ratio L/D of 0.7, the flow separates at the leading edge of the cavity and impinges at the trailing edge. A single vertical flow structure was present within the cavity. For L/D of 3, the flow seems reattached at the floor of the cavity downstream before the flow separating again at upstream leading edge. It can be observed that for L/D =3, two counter rotating vortices present within the cavity due to more energy to the forming shear layer, compared with L/D=0.7 that has less interaction with forming shear layer near the leading edge [26]. It
also can be seen the rake point (R2) in Figure 7 was the highest velocity compared the others rakes in both aspect ratio (L/D). This result revealed that R2 is the area of oscillating vortex flow mode.

**Figure 6.** Comparison result of the flow structure at Reynolds number of 7831 between smoke test experiment and CFD simulation for different aspect ratio (L/D): (a) L/D = 0.7 and (b) L/D = 3

**Figure 7:** The velocity profile inside cavity of two-way radio at varied length to depth ratio (L/D): (a) L/D: 0.7 (b) L/D: 3
6. Conclusion

The flow structure inside different aspect ratio (L/D) of the cavity were investigated by experimentally and numerically. The present study concludes that, as the cavity length changes from 0.7 to 3, the flow structure become complex and the secondary vortex flow inside the cavity becomes larger. However, the snapshot of secondary clockwise vortex inside a larger aspect ratio was not clearly captured due to the poor resolution by the camera. In length to depth ratio (L/D) of 0.7, only one continuous recirculation vortex flow occurred within the cavity. It was also found that the velocity of the vortex shed in the middle of the cavity (R2) at length to depth ratio (L/D=0.7) was the highest velocity compared with the other point rake location and length to depth of 3. The comparison of flow structure in CFD and smoke test were in good agreement. The smoke wire flow visualization performed inside the Viwika wind tunnel was able to visualise the vortex flow inside cavity. However, the smoke lines were big due to the heat grid size inside wind tunnel. The effect of aspect ratio in flow structure was important parameter that causes an oscillating flow mode. Further work is needed to measure the oscillation flow mode inside the cavity using microphones.

7. References

[1] Xu Y, Zheng ZC, Wilson DK 2011 A computational study of the effect of windscreen shape and flow resistivity on turbulent wind noise reduction J Acoust Soc 129(4) 1740-7.
[2] Neittaanmäki P, Rossi T, Korotov S, Oñate E, Périaux J, Knörzer D 2004 Landing gears aerodynamic interaction noise,” Eur. Congr. Comput. Methods Appl. Sci. Eng 115–35.
[3] Wang Y, Deng Y, Yang Z, Jiang T, Su C, Yang X. 2013 Numerical study of the flow-induced sunroof buffeting noise of a simplified cavity model based on the slightly compressible model. J Auto Eng. 8 1187-99.
[4] Chen KH, Johnson J, Dietschi U, Khalighi B 2010 Wind noise measurements for automotive mirrors. SAE Int. J. Passeng. Cars - Mech. Syst 2, 419–33.
[5] Alam F, 2000 The Effects of Car A-pillar and Windshield Geometry on Local Flow and Noise
[6] Thompson DJ, Latorre Iglesias E, Liu X, Zhu J, Hu Z. 2015 Recent developments in the prediction and control of aerodynamic noise from high-speed trains. Int. J of Rail Trans. 3 119-50.
[7] Sun, X. and Xiao, H 2018. Numerical modeling and investigation on aerodynamic noise characteristics of pantographs in high-speed trains. Complexity.
[8] Čeheľová D, Urbán D, Bielek B, Rychtáriková M, Roozen NB 2017 Literature review on wind induced sound on buildings. Akustika. 1 42-7.
[9] Wei Xu and Feng Xu, “Numerical Study on Wind-Induced Noise of High-Rise Building Curtain Wall with Outside Shading Devices,” Shock and Vibration. 12
[10] Mohamed, S Graf, H.R. and Ziada, S 2012 Aeroacoustic Source of a Shallow Cavity in a Pipeline. In ASME 2011 Pressure Vessels and Piping Conference, Am Soc of Mech Eng Dig Col 269-276
[11] Gallo P, Fredianelli L, Palazzuoli D, Licitra G, Fidecaro F. 2016 A procedure for the assessment of wind turbine noise. Applied Acoustics Dec 15; 114 213-17.
[12] Zeng, W.D., Wang, Y.S. and Yang, Q.F., 2010. Numerical calculation of flow noise of submarine with full appendages. Acta Armamentarii, 9.
[13] Fabre B, Gilbert J, Hirschberg A, Pelorson X. 2012 Aeroacoustics of musical instruments. Annual review of fluid mechanics 44:1-25
[14] Wang, L Zander, A.C. and Lenchine, V.V 2012 Measurement of the self-noise of microphone wind shields. Measurement, 37.
[15] Staple, B.W., Schladt, G.A. and Holmes, M.E., Motorola Solutions Inc, 1994. Wind noise and vibration noise reducing microphone. U.S. Patent 5, 288,955.
[16] Amandolese X, Regardin C 2004 An experimental study of the acoustic oscillations by flows over cavities. J. of Vib and Acou. 126(2) 190-5.
[17] Ghanadi F, Arjomandi M, Cazzolato B, Zander A. 2014 Interaction of a flow-excited Helmholtz resonator with a grazing turbulent boundary layer. Exp. Therm. and Flui Sci. 58:80-92.
[18] Meissner M 2002 Acoustic modes induced by flow in a pipe with two closed side-branches. *App. Acous.* Oct 1; 63(10):1071-83.

[19] Tonon D, Hirschberg A, Golliard J, Ziada 2011. Aeroacoustics of pipe systems with closed branches. *Int. J. of Aero* 10(2-3) 201-75.

[20] Slaton WV, Nishikawa A 2015 Aeroacoustic response of coaxial wall-mounted Helmholtz resonators in a low-speed wind tunnel. *The J. of the Acous. Soc* 37(1) 253-60.

[21] Yilmaz I, Ayli E, Aradag S 2013 Investigation of the effects of length to depth ratio on open supersonic cavities using CFD and proper orthogonal decomposition. *The Scie World J*

[22] Yoshida T, Watanabe T. 2016 Numerical simulation of flow over an open cavity with self-sustained oscillation mode switching. *Open J of Fluid Dynamics* 6(04) 361.

[23] Fisol UM, Ripin ZM, Ismail NA, Saad AA, Hashim MK, Chan PY, Yow HT, Tan CH, Hii UH, Saw TS 2013 Wind noise analysis of a two-way radio. *IEEE International Conference on Smart Instrumentation, Measurement and Applications (ICSIMA) I-5*

[24] Hashim MK, Ismail NA, Saad AA, Chan PY, Fisol UM, Ripin ZM, Yow HT, Tan CH, Hii UH, Saw TS 2013 Sound quality analysis for two-way radio under wind noise. *IEEE International Conference on Smart Instrumentation, Measurement and Applications (ICSIMA) I-5.*

[25] ANSYS Fluent version 16 Documentation

[26] Fisol UM, 2016 Numerical and Experimental Analysis of Two – Way radio, Master Thesis.

**Acknowledgement**

The authors acknowledge this research work was fully funded by the Research Universiti Sains Malaysia under RUI Grant (1001/PAERO/8014005). Thanks to technical supports, Mr Mahmud Isa from School of Aerospace Engineering for the help.