

Experimental and Numerical Investigations on Control Methods of Cavity Tone by Blowing Jet in an Upstream Boundary Layer

Hiroshi Yokoyama, Ryo Adachi, Taiki Minato, and Akiyoshi Iida
Toyohashi University of Technology

ABSTRACT

The objective of this paper is to clarify the mechanism for the reduction of cavity tone with blowing jets aligned in the spanwise direction in the upstream boundary layer. Also, the effects of spacing of the jets on the reduction are focused. To achieve these objectives, direct aeroacoustic simulations were conducted along with wind tunnel experiments. The depth-to-length ratio of cavity was $D/L = 0.5$. The incoming boundary layer was laminar, where the boundary layer thickness was $\delta/L = 0.055$. The predicted flow fields without control show that two-dimensional large-scale vortices are shed and become acoustic sources in the cavity. The effects of spanwise spacing of spanwise-aligned jets on the cavity flow and tone were clarified with computations and experiments with the different pitches of $s/L = 0.1 - 1.0$ ($s/\delta = 1.8-18.2$). As a result, the largest reduction level was obtained for $s/L = 0.5$. For this control condition, the predicted results show that longitudinal vortices are introduced by the jets in the incoming boundary layer of cavity and suppress (break down) the two-dimensional large-scale vortices which are related with cavity tone. The measured effects of the control on the distributions of Coherent Output Power (COP) with reference to the sound support it. Meanwhile, for $s/L = 0.1$, the distance of induced longitudinal vortices become so narrow that the neighboring vortices interact with each other and intense longitudinal vortices are not developed. For $s/L = 1.0$, the large-scale vortices were clarified to sustain two-dimensionality between the induced longitudinal vortices due to the wide spacing of the introduced longitudinal vortices. As a result, the reduction of cavity tone was smaller for $s/L = 0.1, 1.0$. In summary, $s/L = 0.5$ is an adequate spacing for intense longitudinal vortices to be introduced into the cavity flow and to reduce cavity tone effectively.

CITATION: Yokoyama, H., Adachi, R., Minato, T., and Iida, A., "Experimental and Numerical Investigations on Control Methods of Cavity Tone by Blowing Jet in an Upstream Boundary Layer," *SAE Int. J. Passeng. Cars - Mech. Syst.* 10(3):2017, doi:10.4271/2017-01-1786.

INTRODUCTION

Self-sustained oscillations in a flow over a cavity shown in Fig.1 such as a sunroof of an automobile and various gaps of parts of a transportation vehicle often radiate intense tonal noise. In the development of various transport vehicles, it is important to predict these oscillations during the design stage and invent ways to suppress them.

Many researchers over the past 50 years have investigated the mechanism of acoustic radiation from the self-sustained oscillations in the cavity flows. Rossiter [1] described an oscillation mechanism similar to that presented for edge tones by Powell [2]. In this mechanism, the interactions of vortices with the downstream edge of the cavity radiate acoustic waves, which lead to the formation of new vortices at the upstream edge.

Zhuang *et al.* [3] investigated the control of cavity tone by blowing jets at the upstream edge of cavity. Huang *et al.* [4] succeeded in the reduction of cavity tone with the jets induced by spanwise aligned plasma actuators, which introduced spanwise variation of velocity in the cavity flow. Yokoyama *et al.* showed that the cavity tone was reduced by blowing jets aligned across the span in the upstream boundary layer, where the reduction level was affected by the spacing

of the jets. However, the reduction mechanism and effects of the spanwise pitch of the jets on the controlled cavity flow and tone have not been sufficiently clarified.

The objective of this paper is to clarify the mechanism of the reduction of cavity tone by blowing jets aligned across the span in the incoming boundary layer, where the jet velocity is vertical to the wall, and the effects of the spacing of the jets on the controlled cavity flow and cavity tone. To do this, wind tunnel experiments and direct aeroacoustic simulations were performed.

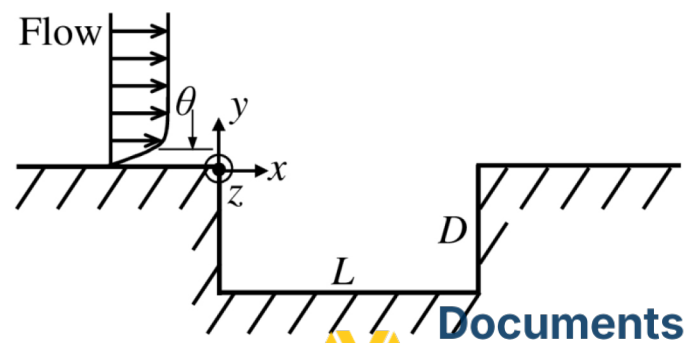


Figure 1. Configurations for a flow over a two-dimensional cavity

METHODOLOGIES

Flow Conditions and Control by Blowing Jets

Experiments and computations were performed regarding the control of the flow and tone around a two-dimensional cavity shown in Fig.1. The parameters of the cavity flows are shown in Table 1. The cavity length was $L = 20$ mm, and the depth was $D/L = 0.5$, where the acoustic resonance such as a depth mode does not occur.

The freestream velocity was $U_0 = 30$ m/s in both the experiments and simulations, while the flow visualization of the jets was performed at a slower freestream velocity of $U_0 = 4$ m/s. At the freestream velocity of $U_0 = 30$ m/s, the Reynolds number based on the cavity length is $Re \equiv U_0 L / \nu = 3.9 \times 10^4$ and the Mach number is $M \equiv U_0 / a = 0.09$, where ν and a are the dynamic viscosity and the speed of sound in air. The incoming boundary layer was laminar, and the 99% boundary layer thickness and momentum thickness were $\delta/L = 0.055$ and $\theta/L = 0.0071$, respectively, at the streamwise position corresponding to the upstream edge of the cavity, where these values were measured and predicted for the flow over the flat plate without a cavity.

Figure 2 shows the variation of the fundamental frequency and sound pressure levels with the freestream velocity measured by preliminary experiments along with the Rossiter mode modified by the actual convective velocity of the vortices [5]. As shown in this figure, it is clarified that the tonal sound was gradually increased with a higher velocity. Also, the Rossiter mode [1] was constant ($m = 3$) around the velocity of 30 m/s.

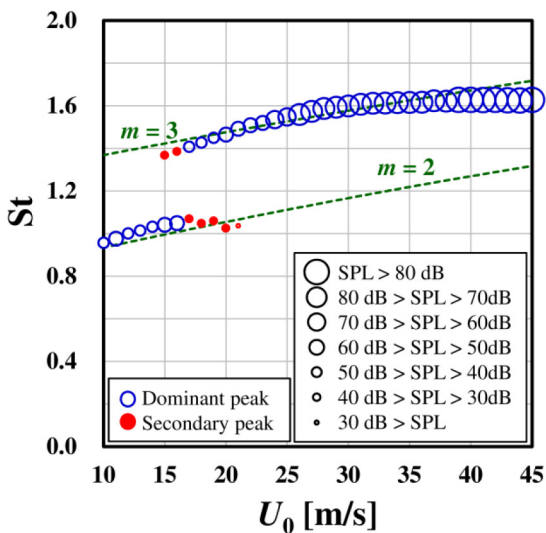


Figure 2. Variations of measured fundamental frequency and sound pressure level with the velocity.

Table 1. Parameters of Cavity flow.

Cavity length L	20 mm
Depth-to-length ratio D/L	0.5
Freestream velocity U_0	30 m/s
Reynolds number $Re \equiv U_0 L / \nu$	4.0×10^4
Mach number $M \equiv U_0 / a$	0.09
Boundary layer thickness δ/L and Momentum thickness θ/L	0.055, 0.0071

Table 2 shows the parameters of blowing jets for the control of the cavity tone. The jets were blown from circular holes aligned across the span at $x_j/L = -1.0$, where the diameter of the hole was set to be $d_j/L = 0.05$ ($d_j/\delta = 0.9$ at $U_0 = 30$ m/s). In order to clarify the effects of the jet velocity on the reduction in the sound pressure level, the cross-sectional average jet velocity was changed in the range of $V_j/U_0 = 0.0 - 0.3$ in the wind tunnel experiments. To clarify the effects of the spacing of the jets on the control effects, the spacing was changed in the range of $s/L = 0.1, 0.25, 0.5, 1.0$ ($s/\delta = 1.8, 4.5, 9.2, 18.2$).

Table 2. Parameters of blowing jets.

Cross-sectional average jet velocity V_j/U_0	0.0-0.3
Diameter of ejection hole d_j/L (d_j/δ at $U_0 = 30$ m/s)	0.05 (0.9)
Spacing s/L (p/δ at $U_0 = 30$ m/s)	0.1, 0.25, 0.5, 1.0 (1.8, 4.5, 9.2, 18.2)

Experimental Methodology

Wind Tunnel and Test Section

Figure 3 shows the experimental setup. The experiments were carried out using a suction-type, low-noise wind tunnel with a rectangular test section with a cross section of 150 mm $\times$ 75 mm. The intensity of freestream turbulence and non-uniformity of the freestream were less than 1% and 0.5%, respectively. The background noise level was 58 dB (A) at the freestream velocity of 30 m/s.

The test section composed of the cavity was terminated in the spanwise direction by the end walls composed of porous plates to suppress acoustic resonance in the spanwise direction. The width of the test section was $W/L = 7.5$.

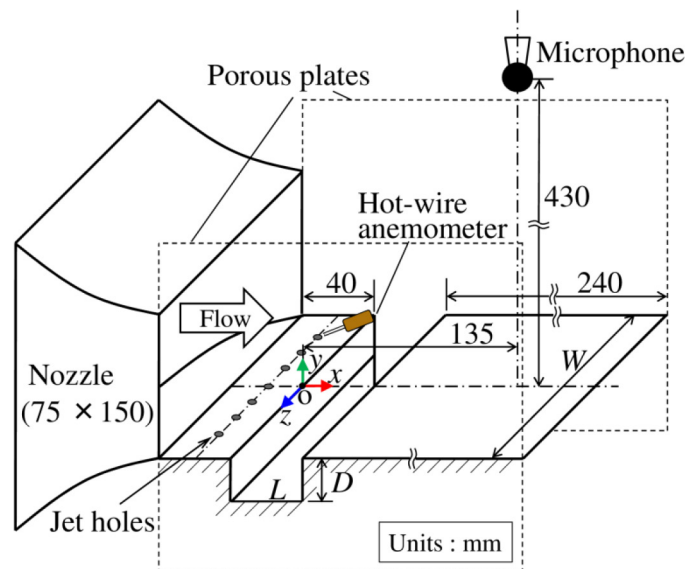


Figure 3. Experimental setup.

Measurement of Sound and Flow Fields

The magnitude of the acoustic pressure in the far field ($x/L = 6.75$, $y/L = 21.5$) was measured with a non-directional 1/2 inch microphone (NL-52, Rion Co., Ltd). The velocity profile of the blowing jets were measured by a hot-wire anemometer. Also, in order to clarify the

effects of the control on the vortices related with the cavity tone, the Coherent Output Power (COP) was computed by the simultaneous measurement of the velocity fluctuations in the cavity flow and sound pressure. The definition of the COP is shown below:

$$\text{COP} = \gamma^2(f) \cdot E_u(f) / U_0^2, \quad (1)$$

where $\gamma^2(f)$ and $E_u(f)$ are the coherence and power at the fundamental frequency.

Blowing Jets

The flow rate of the blowing jets was controlled by the compressor (ACP-50, Ryobi Limited) with the flow meter. The orifice coefficient of the meter was corrected by that estimated by the profiles of the jets measured by the hot-wire anemometer.

Figure 4 shows the visualized flow on the y - z plane in the downstream of the jets ($x/L = -0.25$) at the freestream velocity of $U_0 = 4.0$ m/s and the jet velocity of $V_j/U_0 = 0.25$. The figure shows that approximate symmetric pairs of streamwise vortices appear due to the blowing jets.

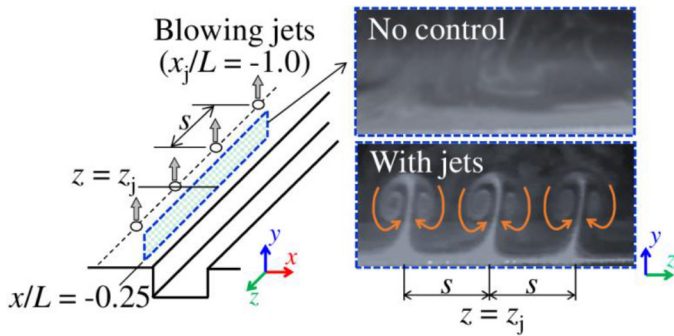


Figure 4. Visualized flow on y - z plane in the downstream of the jets ($x/L = -0.25$) at the freestream velocity of $U_0 = 4.0$ m/s and the jet velocity of $V_j = 1.0$ m/s with the spacing of the jets of $s/L = 0.25$, where z_j is the spanwise position of the jet.

Computational Methodology

Governing Equations and Finite Difference Schemes

To clarify the fluid-acoustic interactions in the cavity flows, flow and acoustic fields were simulated simultaneously by directly solving the three-dimensional compressible Navier-Stokes equations in the conserved form.

$$\mathbf{Q}_t + (\mathbf{E} - \mathbf{E}_v)_x + (\mathbf{F} - \mathbf{F}_v)_y + (\mathbf{G} - \mathbf{G}_v)_z = 0, \quad (2)$$

where $\mathbf{Q}$ is the vector of the conserved variables, $\mathbf{E}$, $\mathbf{F}$, $\mathbf{G}$ are the inviscid flux vectors, and $\mathbf{E}_v$, $\mathbf{F}_v$, $\mathbf{G}_v$ are the viscous flux vectors.

The spatial derivatives were evaluated by the sixth-order-accurate compact finite difference scheme (fourth-order accurate on the boundaries) [6]. Time integration was performed by the third-order accurate Runge-Kutta method.

The flow in the cavity became turbulent at the present conditions, although the incoming boundary layer was laminar. To reduce the computational cost, large-eddy simulations (LES) were performed in the present study. No explicit Subgrid-Scale (SGS) model was used. The turbulent energy in the Grid-Scale (GS) that should be transferred to SGS eddies was dissipated by a 10th-order spatial filter, as described below. A number of studies [7, 8, 9] have shown that the above-mentioned method, which combines low-dissipation discretization schemes and filtering, correctly reproduces turbulent flows. This filter, which is given below, also removed numerical instabilities [10]:

$$\alpha_f \hat{\psi}_{i-1} + \hat{\psi}_i + \alpha_f \hat{\psi}_{i+1} = \sum_{n=0}^5 \frac{a_n}{2} (\psi_{i+n} + \psi_{i-n}), \quad (3)$$

where ψ is a conserved quantity, and $\hat{\psi}$ is the filtered quantity. The coefficients a_n had the same values as those used by Gaitonde and Visbal [11], and the value of parameter α_f was 0.45. It has been confirmed that the cavity flows with incoming laminar and turbulent boundary layers could be predicted with high accuracy using the above-mentioned methods [12,13].

Computational Grid

Figure 5 shows the computational grid for the flow over the cavity. The spanwise length of the computational domain of $W/L = 2.0$ was sufficiently wide to reproduce the effects of the control with the widest spacing of $s/L = 1.0$.

The spanwise grid resolution was equally $\Delta z/L = 1/320$ and the streamwise resolution around the ejection holes of the jets was also $\Delta x/L = 1/320$. These resolutions were sufficiently fine to reproduce the profiles of the jets with the hole diameter of $d_j/L = 0.05$ (1/20). The streamwise resolution in the cavity was equally $\Delta x/L = 1/100$, and the normal resolution near the wall in the incoming boundary layer and in the shear layer in the cavity was $\Delta y_{\min}/L = 1/400$. The grid was stretched in the normal and streamwise direction into the far field. The grid resolution in the x - y cross-section in the cavity was the same as that in the past research [13], where the predicted flow field and sound were clarified to be in good agreement with those measured. Moreover, the quadrupling grid points (641 points) were used in the spanwise direction as mentioned above. The total grid points are 39 million.