A Study on the Process of Low-Frequency Noise Generation in a Multi-Blade Centrifugal Fan

Masatoshi Kawasaki¹, Hiroyuki Hirahara², and Donghyuk Kang²

¹Sanden Automotive Climate Systems Corporation
20, Kotobuki, Isesaki-shi, Gunma, 372-8502 Japan
masatoshi.kawasaki.sw@g-sanden.com
²Department of Mechanical Engineering, Saitama University
255, Shimo-Okubo, Sakura-ku, Saitama-shi, Saitama 338-8570, Japan,
hhira@mail.saitama-u.ac.jp, dhkang@mail.saitama-u.ac.jp

Abstract

This study numerically investigates the flow characteristics of a multi-blade centrifugal fan in order to identify the process of low-frequency aerodynamic noise generation. Computational analysis has been carried out using the large eddy simulation (LES) model. In the preliminary study, it was confirmed that the present computational fluid dynamics simulation using LES shows good agreement with the experimental results of the performance curve, sound pressure spectra, and the flow field obtained by using a time-resolved particle image velocimetry (PIV) system. Consequently, it was found that the appearance of an unexpected significant flow, which recirculated from the scroll casing, passed through the impeller, and returned to the scroll casing again to exit the scroll casing in a specific region, was similar to the flow suggested in the experimental results, which were obtained utilizing PIV measurement. The flow field related to the fluctuation, which was remarkably not included in the impeller main flow, existed beside the outer main flow. A certain amount of fluctuated flow bifurcated near the scroll end, one regurgitating into the impeller, and another colliding with the tongue. The former became a driving source of noise and the latter a direct noise-generating factor. Consequently, it was demonstrated that a significant fluctuation owing to flow collision with the tongue propagated through the impeller and formed a loop.

Keywords: Multi-blade centrifugal fan, Noise, Low flow rate, Fluctuation, CFD

1. Introduction

Air conditioner noise has become harsher relative to other noise sources like engine, motor, exhaust, and wind in the current automotive trend. Hence, the reduction of air conditioner noise is perceived as a critical challenge to attaining an excellent comfortable ambiance in automobiles. Thus far, multi-blade centrifugal fans have been adopted in automotive air conditioning systems, given their ability to provide high pressure and flow rates within relatively compact spaces. However, they also generate unpleasant noise because of the fluctuating internal flow.

Various studies have focused on the noise problem associated with multi-blade centrifugal fans. Velarde-Suárez et al. [1] experimentally investigated the influence of various shapes of the tongue and their geometrical relation to the noise generated and noted design conditions that reduced noise without confining the operating range. Similarly, Velarde-Suárez et al. [2] investigated the pressure fluctuation on the volute wall surface using computational fluid dynamics (CFD) and clarified the influence of the flow rate and tongue shape on the blade passing frequency (BPF) noise. Sakai et al. [3] measured the pressure fluctuation between blades using a semiconductor pressure sensor via a pressure tube and quantified the Ribner sound source and showed that the intensity of the sound source increases as a fan blade approaches the tongue. Norisada et al. [4] developed a noise prediction method for a multi-blade centrifugal fan using the Lattice Boltzmann method. They also discussed the internal flow and noise level for several blowers with different tongue clearances. Their paper noted that the pressure fluctuation at the inter-blade near the inlet of the scroll casing was closely related to the generated noise. Sasaki et al. [5] proposed a broadband noise prediction model for the shedding of Kármán vortex streets from the trailing edge of the blade where noise prediction was applied to blowers with a different number of blades. The prediction model was validated by comparing the test results. Furthermore, Kim et al. [6] optimized the fan blade by applying the Krigeing metamodel and an evolutionary algorithm to minimize the BPF and turbulent noise and employed the large eddy simulation (LES) and Flowes Williams-Hawkings (FW-H) model for aerodynamic noise
prediction. The prototype test performed, based on the optimum values of radius, thickness, and outlet angle of the blade showed that both the BPF and Over-All (O.A.) noise decreased by about 4 dB when compared to the initial design.

As described above, several studies have focused on broadband and the BPF noise at the design point and high flow rates. By contrast, it should be noted that studies have scarcely been conducted with regards to low-frequency noise, which is frequently observed at a low flow rate and under particular conditions. Such low-frequency noise is also undesirable for a practical device, therefore, it is crucial to identify a causal relationship between the noise generated and flow perturbation.

Extensive experimental and numerical research has been performed for this paper to address this problem. The following conclusions were obtained and reported from the previous experiment wherein time-resolved particle image velocimetry (PIV) measurements were conducted around the impeller and the scroll casing [7]. The flow exiting the impeller at \( \theta = 120^\circ \) shown in Fig. 1 is found to fluctuate at a low frequency. The fluctuation is transported downstream along Flow 1 to Flow 2. A primary flow through the impeller, Flow 4, does not involve the velocity fluctuation of the low frequency. The flow involving the low-frequency fluctuation convects through the scroll casing and bifurcates into the impeller and the tongue in the vicinity of the scroll end, one of which flows back to the impeller. An adverse flow, Flow 1 through the impeller, was observed and flowed out again to the scroll casing at \( \theta = 120^\circ \). Flow 3 towards the tongue collides with the tongue wall surface and is presumed to be a cause of the low-frequency noise generation. Such a loop consisting of Flows 1, 2, and 3 described in the previous paper will be referred to as the ‘disturbance channel loop’ in the present study.

The present study aims to identify an internal flow structure at low flow rates in order to verify the PIV results and clarify the process of low flow frequency noise generation due to flow perturbations in multi-blade centrifugal fans using numerical analysis.

![Fig. 1 Estimation of the disturbance channel loop obtained by PIV measurements [7]](image)

2. Measurement and numerical procedure

2.1 Experimental measurement of the performance and noise generated by the fan

The impeller’s specifications used in the present study are shown in Table 1. The performance of the blower was measured in an anechoic room, following the regulations prescribed by the Japanese Industrial Standard (JIS B 8330). The measuring equipment is shown in Fig. 2. The flow rate was calculated using the pressure difference across an orifice. Here, the static pressure was measured using a digital manometer (DM-3501, Cosmo Instruments) through a minute hole in the duct wall surface. By adjusting the flow rate using a damper, which was located downstream of the equipment, the flow rate, static pressure, motor voltage, motor current, rotating speed, and noise level was acquired for several flow conditions. Through the test run, it was confirmed that the error in each measurement was less than 1%. For noise measurements, a 1/2 inch condenser microphone with a built-in pre-amplifier (MI-1234, Ono Sokki) was installed along the impeller’s rotational axis at 1 m from the upper end of the suction port (bell-mouth). The frequency was analyzed using the fast Fourier transform (FFT) (DS-2000, Ono Sokki) and the sampling frequency, sampling number, and measurement duration period were 20.48 kHz, 2048, and 10 s, respectively. The results of the performance measurement will be discussed in section 2.3.

<table>
<thead>
<tr>
<th>Table 1 Specifications of the impeller</th>
</tr>
</thead>
<tbody>
<tr>
<td>Outer diameter</td>
</tr>
<tr>
<td>Blade height</td>
</tr>
<tr>
<td>Number of blades</td>
</tr>
</tbody>
</table>

![Fig. 2 Experimental apparatus for performance measurements](image)
2.2 Numerical scheme

Assuming an incompressible flow, the following set of equations, mass conservation, and momentum equations are solved numerically using the LES model.

\[ \nabla \cdot v = 0 \quad (1) \]

\[ \frac{\partial v}{\partial t} + (v \cdot \nabla)v = -\frac{1}{\rho} \nabla p + \nu \nabla^2 v \quad (2) \]

Here, \( v, p, \) and \( \rho \) are the velocity, pressure, and density, respectively. In the present simulation, the WALE model was used for the sub-grid-scale, and STAR CCM + Ver. 9.06 was employed as the solver, for which the discretization was based on the finite volume method and the time integration was constructed using the SIMPLE algorithm. The present computational domain is illustrated in Fig. 3. The inlet and the outlet regions involve a sufficient volume to avoid an incidence of numerical disturbances at the boundaries. The upper surface of the cylindrical inlet region was assigned an inlet boundary with atmospheric pressure. In the preliminary study, we investigated the inlet region that was either cylindrical or hemispherical. However, no difference in performance was found for the two cases. The side of the inlet region had a slip wall boundary condition and the outlet boundary was assigned a mass flow condition. The blower section consisted of a stationary domain constructed using the scroll casing and a rotating domain, including the impeller. Here, a sliding mesh interface was applied between the impeller and the scroll casing. The cross-sectional view of the computational grids is shown in Fig. 4. The computational grids were constructed using a polyhedral cell and prism layer. The impeller was resolved in a fine mesh carefully, and its surface was wrapped with the prism layer so that the \( y^+ \) value of the wall surface was about 1. The total number of grids was approximately 28 million. The preliminary execution checked the quality of the mesh and its number. In order to obtain sufficient accuracy within a reasonable time frame, a high number of grids, 28 million, was required for sufficient resolution. The time step was 6.087E-05 s, and the total physical time set after sufficient time lapse from the initial transition was 0.5405 s, which corresponds to 30 rotations. During the preliminary evaluation, the sound pressure levels at 30 and 40 rotations were compared, but were found to be nearly the same. Hence, the former data was used in this study. The driving conditions of the blower are \( \phi = 0.13, 0.23, \) and \( 0.35 \) in the present study, whereas the present blower has maximum efficiency when \( \phi = 0.35 \), as shown in the next section. However, special attention was given to the low flow operating condition \( \phi = 0.23 \) in particular.

![Fig. 3 Computational domain](image)

![Fig. 4 Computational grids](image)

2.3 Comparison of blower performance between experiment and CFD

Flow and pressure coefficients, \( \phi \) and \( \psi \) are defined as eqs. (3) and (4) in this paper.

\[ \phi = \frac{Q}{\frac{1}{2} \pi D^2 u} \quad (3) \]

\[ \psi = \frac{\Delta P_t}{\frac{1}{2} \rho u^2} \quad (4) \]

Where \( Q \) and \( D \) are the volumetric flow rate and the outer diameter of the impeller, respectively. \( \Delta P_t \) is the total pressure gain of the blower. Figure 5 shows the comparison of \( \psi\phi \) between the experimental curve and numerical results. \( \phi = 0.35 \) is the rated flow condition and has maximum efficiency. The CFD simulation shows a 5% incremental difference when compared to the experimental results under three conditions. However, the results qualitatively agree with the experiment.

Figure 6 shows a comparison of the measurement and the CFD simulation concerning the sound pressure spectrum for three driving conditions. The numerical prediction of the sound pressure level at a far-field, which is the same as the experimental sampling position, was estimated using the FW-H equation, which has the advantage of estimating the sound pressure level emitted from the impeller. The FW-H equation takes into account the effect of the wall motion and calculates sound pressure using pressure fluctuations on a solid wall such as the impeller. Utilizing this technique, the sound pressure level in the far-field
can be obtained by employing an unsteady incompressible flow calculation.

In this figure, two discrete peaks at about 100 and 150 Hz are detected in the low-frequency region at $\phi = 0.23$ in both the experiment and CFD simulation which are henceforth referred to as the first and second frequencies, respectively. Although the sampling duration is different for the experiment and the CFD simulation, the experimental results were obtained for 10 s and 555 rotations, and the CFD simulation was only carried out for approximately 0.5405 s and 30 rotations, the spectra show a fair agreement. Therefore, we can discuss the noise problem given the present CFD simulation.

On the other hand, at $\phi = 0.13$, small peaks are observed, and at $\phi = 0.35$, a discrete noise peak cannot be detected in the low-frequency band. Therefore, it is concluded that the two discrete noises are prominent at $\phi = 0.23$. Here, it should be noted that the first and second frequencies do not coincide with the higher harmonics of rotational speeds of 55.5 Hz. Further, these peaks also occur in the CFD results, calculated as rigid bodies, and hence are caused by fluid flow rather than natural vibration of structures.

---

**Fig. 5** Fan characteristic

**Fig. 6** SPL spectra

### 3. Result and discussion

#### 3.1 Instantaneous flow profiles

The geometry and coordinates of the blower are illustrated in Fig. 7 and $\theta$ is measured from the tongue of the scroll casing. $z/b$ is the normalized span height and $b$ is the height of the blade. Figure 8 shows the instantaneous velocity distributions in the horizontal section at $z/b = 0.4$ at $\phi = 0.13$, 0.23, and 0.35. The enlarged flow view is depicted below. The optimal flow pattern is observed at $\phi = 0.35$, owing to the rated condition. Contrastingly, at the low flow rate conditions $\phi = 0.13$ and 0.23, the disturbed flow is observed in the velocity vector maps and given that these conditions are driven at low flow rates, backflow occurs inside the blower.

Investigations pertaining to the internal impeller flow at low flow rates, especially relating to the recirculation flow near the tongue, have been discussed in this sub-section. Between $\theta = 30^\circ$ and $330^\circ$, an adverse flow is observed from the trailing edge to the leading edge and the flow passing through the impeller between $\theta = 90^\circ$ to $120^\circ$, is indicated with a closed dashed line in Figs. 8 (a) and (b). This flow is referred to as ‘adverse impeller flow’, hereafter. At $\phi = 0.13$, the adverse impeller flow region increases further than $\phi = 0.23$. As well as at $z/b=0.4$, the adverse impeller flow was also observed at $z/b < 0.7$. On the other hand, for $\phi = 0.35$, the impeller outlet flow is almost uniform in the circumferential direction, and the disturbance such as impeller adverse flow is not observed inside the impeller. The presence of the adverse impeller flow does not necessarily generate the noise. That is, although the adverse impeller flow at $\phi = 0.13$ is more extensive than $\phi = 0.23$ the noise peaks at $\phi = 0.23$ are larger than at $\phi = 0.13$. Therefore, it is suggested that the condition of this flow plays an important role.
3.2 Adverse impeller flow

As described in the previous section, the adverse impeller flow appears with a decrease in flow coefficients. In this section, the influence of the flow rate on the low-frequency noise is considered. Figure 9 illustrates the time-averaged flow rate distribution along the scroll angle. The partial flow rate was calculated for 12 sections along the scroll casing with 30° increments, as shown in Fig. 9 (a). $Q_{f1}$ to $Q_{f12}$ is the flow rate fractions to the total flow rate at the impeller outlet of each cross-section, which are equivalent to the difference between the downstream and upstream cross-sections for each divided section in the scroll casing. The amount of $Q_f$ indicates the flow rate ratio of the impeller in terms of the scroll angle. As shown in these curves, $\phi = 0.35$ shows uniformity over the whole region except for the region near the tongue. When the flow rate is lower, a defect in the flow rate is observed between $\theta = 300^\circ$ to $60^\circ$ at $\phi = 0.23$, and $\theta = 240^\circ$ to $90^\circ$ at $\phi = 0.13$. In particular, the negative value of $Q_f$ implies an occurrence of adverse flow in the impeller. Such adverse flow is considered as a necessary condition to generate low-frequency noise.

![Fig. 8 Instantaneous velocity distributions at $z/b = 0.4$](image)

(Upper : overall view / Lower : partially enlarged view)

![Fig. 9 Flow rate in each section of the impeller](image)
Observation point is 7 mm inside from the leading edge at $\theta = 60^\circ$ and $\alpha/b = 0.4$.

Fig. 10 Fluctuation characteristics of velocity magnitude inside the impeller

To demonstrate the fluctuation of the adverse impeller flow, the time-sequential velocity and frequency spectrum are shown in Fig. 10. The point of interest is located 7 mm inside with respect to the leading edge, at $\theta = 60^\circ$ and $\alpha/b = 0.4$. Assuming regular inlet flow without prewhirl inside the impeller, the mean velocities at the blade inlet are estimated to be 2.5, 4.1, and 5.7 m/s at $\phi = 0.13, 0.23$, and 0.35, respectively. Let $v_{in}$ and $v_{ave}$ be the estimated velocities at the blade inlet and time-averaged velocities at the observation point, respectively. The velocity fluctuation at $\phi = 0.13$ shown in Fig. 10 (a) involves serious fluctuations. The large velocity magnitude implies the effect of the adverse impeller flow, rather than regular flow.

Similarly, Fig. 10 (b) shows a relatively large velocity magnitude and severe fluctuations. On the other hand, at $\phi = 0.35$, which is the optimal efficiency point shown in Fig. 10 (c), the velocity is almost constant and close to the estimated value. In Fig. 10 (d), the velocity fluctuation spectrum profiles are indicated for three conditions. It can be observed that the broadband fluctuation occurs at $\phi = 0.13$, whereas a few of discrete peaks are found at $\phi = 0.23$. The primary peak is about 100 Hz and is nearly the same as the first frequency shown in Fig. 6. However, no dominant fluctuations were not observed at $\phi = 0.13$. Thus, the adverse impeller flow at $\phi = 0.23$ is explicitly related to the low-frequency noise generation. For this fluctuation, the authors have confirmed that the dominant peak was detected inside the impeller as well as in the experimental investigation [7].

In this sub-section, the noise generated at $\phi = 0.13$ is discussed when compared to the noise generated at $\phi = 0.23$. As shown in Fig. 8, $\phi = 0.13$ is influenced by a recirculation flow with the adverse impeller flow because the flow towards the scroll exit is relatively weak, and an adverse flow in the impeller generates a significant recirculation in comparison to the condition at $\phi = 0.23$. Further, shown in Fig. 10, the fluctuation is not periodic but broadband. That is, since the flow is unstable at $\phi = 0.23$ as the adverse impeller flow coexists with the main flow, noise with a specific frequency was generated. On the other hand, when $\phi = 0.13$, the recirculation is relatively strong and stable, and thus the specific frequency noise, which was observed $\phi = 0.23$, is suppressed.

3.3 Fluctuation and mean flow profile

The first discrete frequency peak at $\phi = 0.23$ exists inside the impeller, as described in the previous section. In order to analyze the behavior of the low-frequency fluctuation inside the blower, let us consider the correlation between the low-frequency fluctuation and the time-average velocity along with scroll angles at $\phi = 0.23$. Figure 11 shows the distributions of the velocity fluctuation amplitude at the first frequency and the time-averaged velocity distributions of the vertical cross-sections. The cross points inside and outside the impeller in the fluctuating distributions will be described in section 3.5.

While examining a remarkable region of velocity fluctuations of the first frequency, we find two passages in terms of this frequency, which corresponds to Flows 1 and 2 in Fig. 1. The first flow is related to the adverse flow within the impeller, and the other exists inside the scroll casing. The former was represented in the figures at $\theta = 0^\circ$ to $90^\circ$ and $330^\circ$. Therefore, it can be estimated that these peaks are accompanied by the adverse impeller flow, which was explained in the previous section. The latter is observed in the scroll casing at $\theta = 120^\circ$ to $330^\circ$. As the adverse impeller flow discharges from the inside of the impeller through the blades, the fluctuation in this scroll casing side increases after $\theta = 120^\circ$. Apart from $\theta = 120^\circ$, no severe fluctuation close to the impeller outlet was observed. The fluctuation of the first frequency at the impeller outlet was not observed globally but only near $\theta = 120^\circ$. Therefore, this phenomenon is not a rotating stall. If a rotating stall causes this fluctuation, the fluctuation peak should appear close to the whole circumference of the impeller and stall cells were not observed in between the blades. Further, the fluctuation peak gradually shifts to the outer sidewall of the scroll casing with an increase in $\theta$.
Velocity fluctuation amplitude

Time-averaged velocity magnitude

Velocity fluctuation amplitude

Time-averaged velocity magnitude

**Fig. 11** Distribution of velocity fluctuation and magnitude in vertical cross-sections by CFD at $\phi = 0.23$

A: Emanating flow with disturbance from the impeller, B: Channel flow with disturbance existing just outside impeller main flow, C: Bifurcated flow from B, D: Main flow. +: Points for constructing channel of the first frequency disturbance

On summarizing the above observations, an interesting aerodynamic phenomenon can be observed when $\theta \geq 150^\circ$, i.e., a strong fluctuation of the first frequency, appears just outside the main flow. For example, when $\theta = 180^\circ$, one can also recognize that the main flow is located at the impeller outlet, as shown in the velocity magnitude distribution, whereas the strong fluctuation is observed just outside this flow. Here, the velocity magnitude of the main flow was calculated using the velocity triangle at the operating point and was found to be about 32 m/s. Therefore, the main flow was determined using the criteria of 32 m/s or more. Furthermore, this main flow is not accompanied by such strong fluctuation. The fluctuating flow is excluded by the radial velocity component in the main flow and gradually approaches the sidewall while passing through a particular region downstream. After $\theta = 270^\circ$, the fluctuation region is significantly deformed and divided owing to the secondary flow in the scroll casing, which is more prominent than the upstream region. Further, one of the divided fluctuating flow intrudes into the impeller from around $\theta = 300^\circ$ to $330^\circ$, that is, the origin of the adverse impeller flow, as mentioned in the previous section. Therefore, the fluctuations existing inside the impeller and the scroll casing are composed as one passage while traversing the impeller blades. This phenomenon has been confirmed by PIV measurement, as shown in Fig. 12 [7]. Although there are some differences in the value of the fluctuation amplitude and the shape of the distribution, the disturbance is composed of a closed loop through the impeller to the scroll casing.

**Fig. 12** Distribution of velocity fluctuation amplitude at first frequency in vertical cross-sections by PIV($\phi = 0.23$) [7]
3.4 Structure of fluctuation and the main flow

To exhibit a three-dimensional structure of the phenomena described in the previous section, the fluctuated region with the low frequency and the main flow were reconstructed using CFD data. Figure 13 shows the three-dimensional structure from three viewing points. By analyzing the CFD results, the fluctuated region with 100 Hz to 110 Hz frequency was extracted as the red iso-surface of 0.6 m/s or more. Also, a high-velocity region with a velocity of 32 m/s or more is represented by the blue iso-surface in this figure. This demonstration helps in understanding the relationship between the three-dimensionally complicated structure of the fluctuating region and the main flow. Furthermore, the fluctuated region consisting of Flow 1 and the subsequent Flow 2 makes a closed-loop outside the main flow and the main flow does not include the low-frequency disturbance. That is, the fluctuated region and the main flow are isolated from one another. Comparing Figs. 1 and 13, Flows 1, 2, and 3 are included in the red region, and Flow 4 corresponds to the blue region.

Red:
Fluctuated region ≥ 0.6 m/s

Blue:
Main flow region ≥ 32 m/s

Fig. 13 Flow structures of the fluctuated region with low frequency (red) and main flow (blue) at $\phi = 0.23$

3.5 Low-frequency noise source and determinant of disturbance frequency

To detect the position of noise sources, spectrum analysis by means of FFT was conducted for the static pressure on the wall surface of the impeller and the scroll casing. The root mean square of the pressure fluctuation for the band of interest, 95 to 115 Hz at $\phi = 0.23$, is shown in Fig. 14. In the contour of this band, a vigorous intensity was observed on the tongue (a) and the blade surfaces (b). The former is due to the collision of Flow 3 with the tongue surface, and the latter is accompanied by the adverse impeller flow (Flow 1). The pressure fluctuation caused by these collisions leads to considerable noise.

Fig. 14 RMS distributions of pressure fluctuation on the wall surface from 95 Hz to 115 Hz at $\phi = 0.23$

In order to determine the process of the flow frequency, the fluctuating flow passage was extracted precisely using CFD as two trajectories, Paths 1 and 2, respectively, as shown in Fig. 15. Here, these trajectories were reconstructed by connecting the symbols ‘+’ in Fig. 11, where these positions correspond to the maximum point in the relevant disturbance channel flow or based on the adverse flow position of the internal flow to the impeller. As discussed in section 3.4, the disturbance channel in the scroll casing is located outside the main flow. With this extraction, the path 2 disturbance channel loop has been demonstrated precisely. Lagrangian path lines were extracted to compare Path 1 with the actual flows. The yellow lines in Fig. 15 show the path lines during ten rotations of the impeller, in which tracer particles were ejected from the blade trailing edge at $\theta = 300^\circ$. It is found that Path 1 coincides with these path lines. From this analysis, traveling distances and net velocity, averaged along the paths, were calculated for Paths 1 and 2, respectively. The estimated values are presented in Table 2. As shown in this table, Paths 1 and 2 have inconsistent values, both in the distance and averaged velocity. By using typical values, the periods and cycle frequencies were estimated, as shown in this table. Eventually, the estimated cycles are relatively close, even though the estimation has been carried out using Eulerian sampling. The cycle estimated from these values corresponds to the fluctuating flow frequency. The cycles for Paths 1 and 2 may suggest the development of low frequency fluctuating flow oscillation and synchronization. That is, the low frequency fluctuating flow is continuously circulating on Path 1 and Path 2. The fluctuating frequency is decided by the trajectory distance and velocity on paths. The low-frequency noise is generated when Path 1 and Path 2 collide with the impeller blades, and the flow bifurcated from Path 2 near the scroll end collides with the tongue.
4. Conclusions

The low-frequency noise observed in a practical multi-blade centrifugal fan was investigated by means of numerical simulation and then compared with PIV results obtained from previous studies [7]. When the blower was driven under a low flow rate at $\phi = 0.23$, and since the low-frequency noise was observed experimentally, a numerical simulation was conducted to detect the noise generation mechanism. In the numerical simulation, the first frequency of about 100 Hz has been found in the velocity fluctuations as well as the experimental results. On the incidence of the low-frequency noise, the adverse flow was observed remarkably inside the impeller, which was different from the maximum efficiency condition at $\phi = 0.35$. Also, in the scroll casing, the prominent fluctuated flow passage was found outside the impeller’s main flow. The adverse impeller flow and the fluctuated flow passage in the scroll casing form a loop for the low-frequency disturbance channel. The two trajectories connecting the maximum fluctuation points inside and outside the impeller were reconstructed as the disturbance channel loop, which is the representative passage of the fluctuating flow. Furthermore, when the traveling period of each trajectory was estimated, it was indicated that the frequency of both trajectories coincided at about 100 Hz, as well as the discrete frequency of the noise. Concludingly, when the fluctuating flow was connected with the adverse impeller flow near the tongue, the disturbance channel loop was established as a synchronized trajectory of the low frequency fluctuating flow. The analysis of low-frequency fluctuation flow obtained by the present CFD results verified the phenomenon observed in the experiment by means of the time-resolved PIV [7]. Finally, the oscillation and the synchronization process of low-frequency noise were determined in terms of the disturbance channel loop.

**Nomenclature**

- $b$: Blade outlet height [mm]
- $D$: Fan outlet diameter [mm]
- $N$: Rotating speed [rpm]
- $p$: Pressure [Pa]
- $\Delta P_t$: Total pressure [Pa]
- $Q$: Flow rate [m$^3$/h]
- $Q_f$: Flow ratio of each section of fan [m$^3$/h]
- $U$: Tip speed [m/s]
- $v$: Velocity [m/s]
- $z$: Axial directional height [mm]
- $\theta$: Scroll angle [°]
- $\nu$: Kinematic viscosity [m$^2$/s]
- $\rho$: Density [kg/m$^3$]
- $\phi$: Flow coefficient [-]
- $\psi$: Pressure coefficient [-]
- $\omega$: Angular velocity [rad/s]

**References**
