

# Mechanism and Control of Blade Passing Frequency Noise in Forward-Curved Centrifugal Fans

Biping Li<sup>1\*</sup>, Yankun Liu<sup>1</sup>, Zitao Wang<sup>1</sup>, Xu Gao<sup>1</sup>, Junwen She<sup>1</sup>

<sup>1</sup> TCL Industries Air Conditioning Business Unit, Zhongshan, China

\*Correspondence: Biping Li

Received: 22 April 2026; Accepted: 12 May 2026; Published: 24 June 2026

## Abstract:

To address the abnormal blade passing frequency (BPF) noise generated by a forward-curved multi-blade centrifugal fan in the fresh air module of a ducted air conditioner under internal circulation mode, aeroacoustic mechanism analysis and structural optimization were conducted. The study reveals that the lack of effective flow guidance in the intake path under the internal circulation mode causes the airflow to directly impinge on the impeller with a large angle of attack, resulting in severe inflow distortion. This distortion leads to intense periodic pressure pulsations at the top and leading edges of the impeller, which consequently induce high-amplitude discrete BPF noise. By combining experimental spectral analysis and CFD numerical simulations, this paper elucidates the physical mechanism of the BPF noise generation. Furthermore, noise reduction strategies based on inlet flow conditioning are proposed to mitigate the inflow distortion. Specifically, by introducing a V-shaped splitter structure, the flow field organization within the intake chamber is optimized. Meanwhile, a half-shielded bellmouth is employed to prevent the airflow from directly impacting the top region of the impeller, thereby avoiding local pressure pulsations. Experimental results demonstrate that these combined strategies effectively ameliorate the inflow distortion at the suction side and attenuate the pressure pulsation intensity on the impeller, significantly reducing the BPF noise of the centrifugal fan. After the optimization, the peak value of the BPF noise is reduced by 8.8 dB, while the aerodynamic performance (airflow rate) remains stable.

## Keywords:

Centrifugal Fan, Blade Passing Frequency Noise, Inflow Distortion, Spectrum Analysis, CFD

## 1. Introduction

Multi-blade centrifugal fans are widely used in heating, ventilation, and air conditioning (HVAC), automotive air-conditioning, and electronic equipment cooling due to their high flow coefficient, large pressure coefficient, and compact structure. However, under certain installation conditions, unrectified airflow may impinge directly on the impeller, easily generating a single-frequency rotational noise at the blade-passing frequency, which seriously affects sound quality and reduces user experience.

The noise of multi-blade centrifugal fans mainly consists of aerodynamic noise, mechanical noise, and motor noise, among which aerodynamic noise is the dominant component [1]. Huang Qibai et al. [2] pointed out that, in the composition of aerodynamic noise in centrifugal fans, the rotational noise generated by the interaction between the volute tongue and the rotating impeller accounts for a large proportion. Based on this, Mao Yijun et al. [3] found that there is a strong unsteady interference phenomenon between the wake flow at the impeller outlet and the volute tongue. To optimize the rotational noise at the volute tongue, Han Yanlong et al. [4] proposed an arc-shaped volute tongue structure, and Tai Guoliang et al. [5] used an inclined volute tongue to alter the phase difference of the unsteady interaction between the impeller and the volute tongue, thereby optimizing the rotor-stator interaction noise at the tongue. Liu Xiaomin et al. [6,7,8] proposed a bionic design of a volute tongue with an owl-wing leading edge to control the rotor-stator interaction noise between the blades and the volute tongue in multi-blade centrifugal fans. These methods are all devoted to adjusting the rotor-stator interference between the volute tongue and the blade-driven airflow so as to eliminate the source of rotational noise.

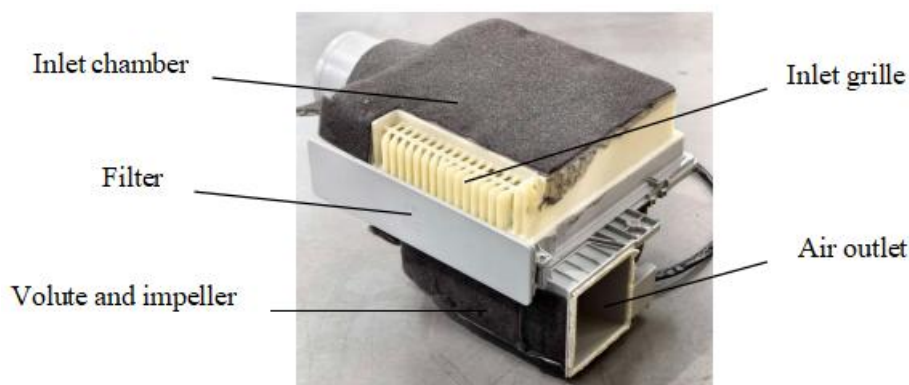
At present, research on rotational noise in centrifugal fans mainly focuses on impeller parameter optimization and volute structure improvement. However, these measures are often based on the assumption of a uniform inlet flow field and periodic pressure pulsations generated only at the impeller and volute tongue. For noise control induced by inlet distortion under confined-space conditions, existing studies mostly focus on flow rectification in external ducts, while relatively little research has been conducted on active flow-field control within the fan inlet chamber through fan structural design.

Against this background, this paper takes a forward-curved multi-blade centrifugal fan for fresh-air air-conditioning as the research object. In response to the abnormal rotational noise occurring under the internal circulation mode, experimental tests and CFD simulations were combined to investigate in depth the mechanism by which inlet distortion affects the internal flow field and pressure pulsations of the impeller. On this basis, several control strategies based on inlet flow rectification and modulation of rotor-stator interaction are proposed, aiming to provide theoretical support and engineering guidance for the design of high-performance, low-noise centrifugal fans operating under confined-space conditions.

## **2. Centrifugal Fan and Test Methods**

### ***2.1. Research Object and Physical Model***

The research object of this study is a centrifugal fan for air purification in an air conditioner. As shown in Figure 1, its main components include the inlet chamber, filter, volute, and impeller. Indoor air is drawn in through the inlet chamber, passes through the filter, and is then discharged out of the unit by the rotating impeller, thereby achieving air purification and circulation. A grille is installed at the inlet of the inlet chamber to provide insect protection, foreign-object prevention, and airflow rectification.



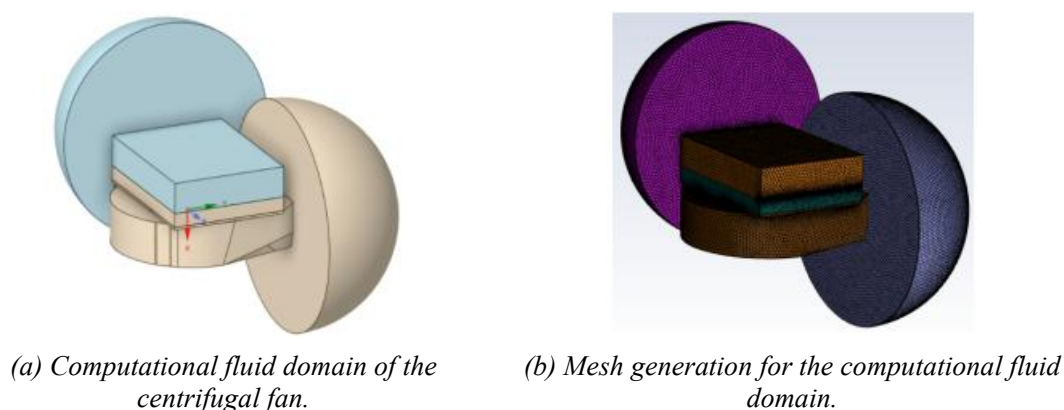
**Figure 1.** Physical diagram of the centrifugal fan.

## 2.2. Noise Spectrum Test Method for the Centrifugal Fan

The microphone array was positioned at the lower side of the centrifugal fan outlet with an offset. The acoustic signals were transmitted to a computer through the SCADAS data acquisition system, and frequency spectrum acquisition and analysis were carried out using Simcenter Testlab software.

## 2.3. Flow Field Simulation Method for the Centrifugal Fan

First, a three-dimensional modeling software was used to extract the fluid domain of the centrifugal fan model, as shown in Figure 2(a). Then, mesh generation was carried out using Ansys Fluent, as shown in Figure 2(b), followed by flow field simulation. The turbulence model adopted was the SST k- $\omega$  model, i.e., the Shear Stress Transport model. This model is well suited for handling near-wall flow behavior. By capturing shear stress characteristics, it can effectively predict flow separation and reattachment, ensuring computational accuracy in high-shear regions while also accurately capturing vortex generation within the flow field. At the same time, it offers relatively balanced computational efficiency.



**Figure 2.** Flow field simulation methodology for the centrifugal fan.

Based on the simulated air volume, a mesh independence study was conducted for the fan simulation model, and the final mesh scheme with 5.08 million cells was selected.

A steady-state flow field simulation was performed for the fluid domain mesh of the centrifugal fan. The turbulence model was set to the SST k- $\omega$  model, namely the Shear Stress Transport model. This model is well suited for handling near-wall flow behavior. By capturing shear stress characteristics, it can effectively predict flow separation and reattachment, ensuring computational accuracy in high-shear regions

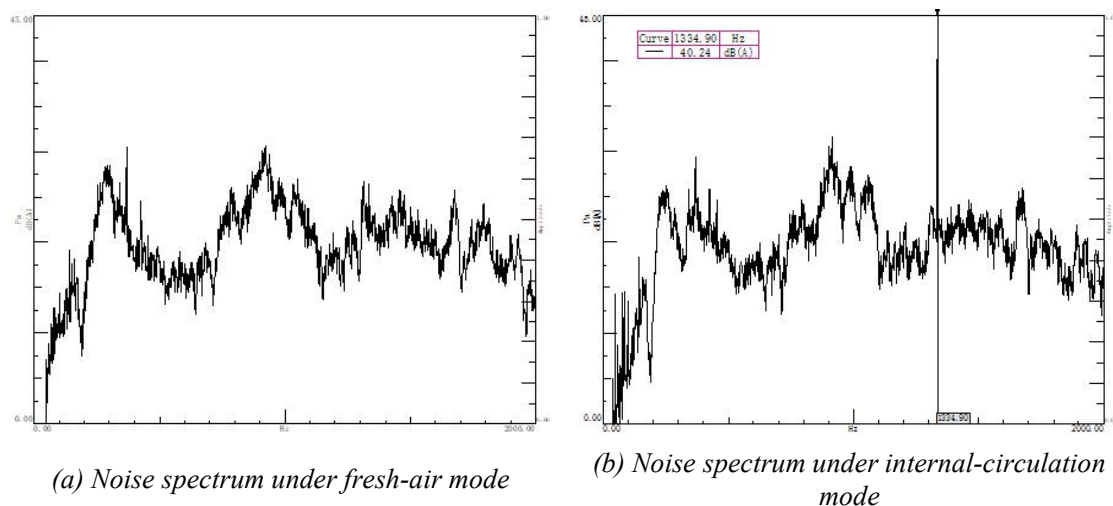
while also accurately capturing vortex generation within the fluid. At the same time, it provides a relatively balanced computational efficiency.

The reference frame was set as a multiple reference frame. The rotational domain was assigned an absolute rotational speed of 1950 rpm, the stationary domain was set as absolutely stationary, and the rotating wall was stationary relative to the rotating domain. The inlet total pressure was 0 Pa, and the outlet static pressure was 0 Pa.

### 3. Mechanism and Control Methods of Rotational Noise in Centrifugal Fans

#### 3.1. Noise Characteristics of the Centrifugal Fan

According to the actual operating scenarios of the centrifugal fan, frequency spectrum tests were carried out. Under the fresh-air mode, the inlet chamber valve was controlled to remain closed, and the impeller speed was set to 1950 rpm. Under the internal circulation mode, the inlet chamber valve was controlled to open, and the impeller speed was set to 1700 rpm. The noise spectra obtained under these two operating modes are shown in Figure 3.



**Figure 3.** Fan noise spectra under two intake modes.

As shown in the Figure 3, under the fresh-air mode, the fan noise spectrum is normal and is mainly composed of motor rotational harmonic noise (97.5 Hz and 162.5 Hz), broadband dipole aerodynamic noise (200–500 Hz), and high-frequency exhaust noise (above 500 Hz). However, under the internal circulation mode, a distinct single-frequency noise appears at 1334.9 Hz, which is highly perceptible in actual experience. As shown in Eq. (1), for the impeller with 47 blades, this frequency is equal to the blade-passing frequency and is therefore identified as the fan's rotational noise.

$$f = \frac{nN}{60} = \frac{1700 \times 47}{60} \approx 1334.9 \text{ Hz} \quad (1)$$

where (n) is the impeller rotational speed (rpm), and (N) is the number of impeller blades.

Rotational noise mainly originates from periodic unsteady pressure pulsations generated on the blade surfaces by the flow. Such pressure pulsations are generally caused by two factors. One is rotor-stator interference, in which the airflow carried by the blades periodically impinges on structural components at a fixed frequency (such

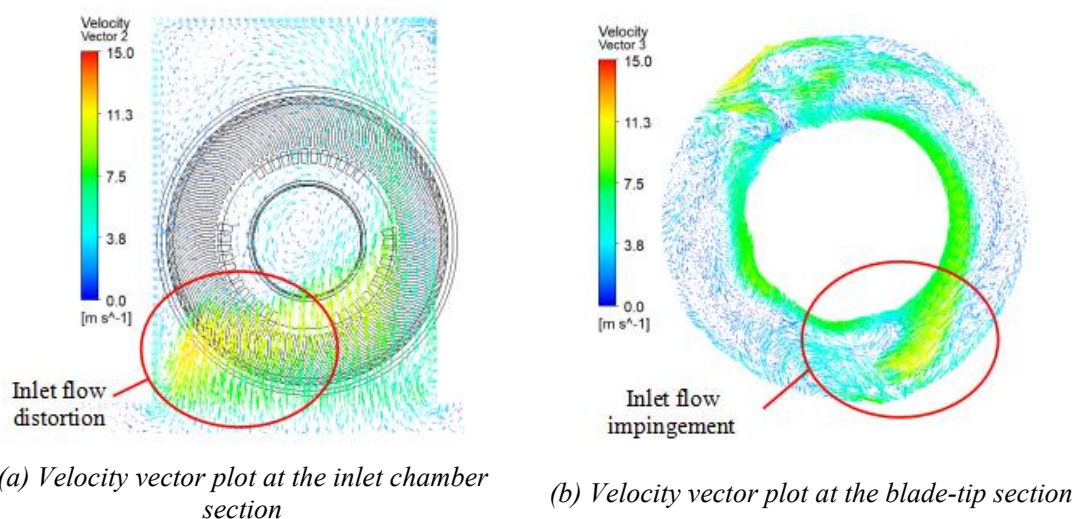
as the volute tongue). The other is flow distortion, under which, for a specific inlet condition, the incoming airflow undergoes periodic fluctuations in velocity direction due to the influence of the rotating impeller, thereby causing pressure pulsations at the blade-passing frequency. For the rotational noise phenomenon in the present fan, rectification measures will be carried out separately for the above two causes.

### 3.2. Generation Mechanism and Analysis of Rotational Noise

According to Lighthill's acoustic analogy theory, in centrifugal fans operating at low Mach numbers, aerodynamic noise is mainly dominated by dipole sound sources generated by the interaction between the fluid and solid boundaries. Among these, discrete tonal noise originates from high-intensity, fixed-frequency periodic pressure pulsations inside the fan. For the mitigation of abnormal rotational noise in the fan, the first task is to identify the characteristic regions in the flow field that induce pressure pulsations at this frequency.

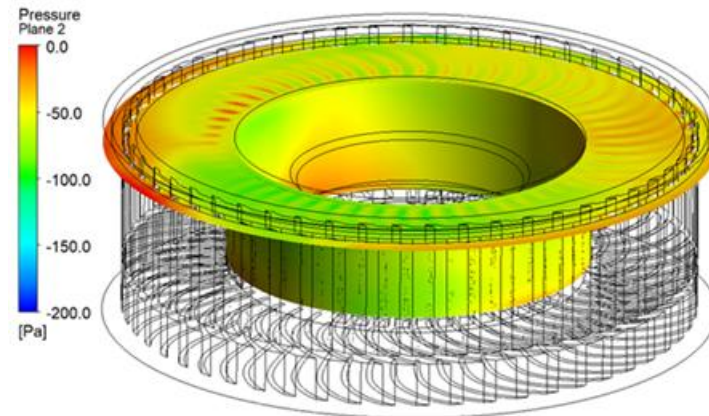
The internal flow field of the fan is complex and characterized by relatively high velocities. If pressure sensors were arranged within it for pressure measurements, it would be difficult to achieve the required sampling accuracy, and the sensors themselves would also significantly affect the flow distribution. Therefore, the computational fluid dynamics (CFD) numerical simulation method described above was adopted in this study for quantitative verification.

Figure 4 shows the velocity vector distributions on the inlet chamber and impeller top surfaces, from which the degree of inlet-flow distortion in the fan can be directly observed. In Figure 4(a), the projection of the impeller is plotted in the inlet chamber flow field for reference and analysis. The external airflow directly enters the inlet chamber from the lower right side, forming a highly non-uniform inlet distortion at the chamber entrance. Although the inlet direction is relatively consistent, the inlet velocity reaches a maximum of 12 m/s near the projected edge of the impeller. On this basis, as shown in Figure 4(b), this high-speed airflow forms an inlet impingement with a velocity of about 11 m/s at the top of the impeller. Because the impeller rotates clockwise and drives the flow in the circumferential direction, the corresponding impingement position at the impeller top is shifted slightly to the right relative to the inlet chamber.



**Figure 4.** Velocity vector fields from the fan simulation.

The pressure generated by the incoming airflow on the impeller can be directly observed in Figure 5. In terms of circumferential distribution, a high-pressure region is formed at the location where the airflow impinges on the top of the blades, while the remaining regions outside the high-speed airflow area exhibit a roughly uniform state.



**Figure 5.** Pressure distribution contour at the impeller top and leading edge.

A comprehensive analysis of the simulation results reveals the mechanism of rotational noise generation. In the stationary reference frame, the high-speed incoming airflow creates a fixed high-impingement region within a specific sector along the circumference of the impeller top. Each time a blade passes through this region, it interacts with the airflow and generates one pressure fluctuation. When all blades successively pass through this fixed region during one impeller revolution, pressure pulsations are produced at a frequency equal to the rotational speed multiplied by the number of blades, commonly referred to as the blade-passing frequency (BPF). According to the dipole sound source model in Lighthill's acoustic analogy theory, unsteady aerodynamic force fluctuations on a solid surface are the source of dipole noise; therefore, single-frequency rotational noise at the BPF is generated.

### **3.3. Control Methods for Rotational Noise**

In summary, this intense discrete noise originates from structural deficiencies in the inlet chamber itself. The distorted inlet flow interacts with the rotating impeller, thereby inducing this single-frequency dipole noise. Therefore, control measures can be implemented from two perspectives: the source of the inlet-flow distortion and its interaction with the impeller.

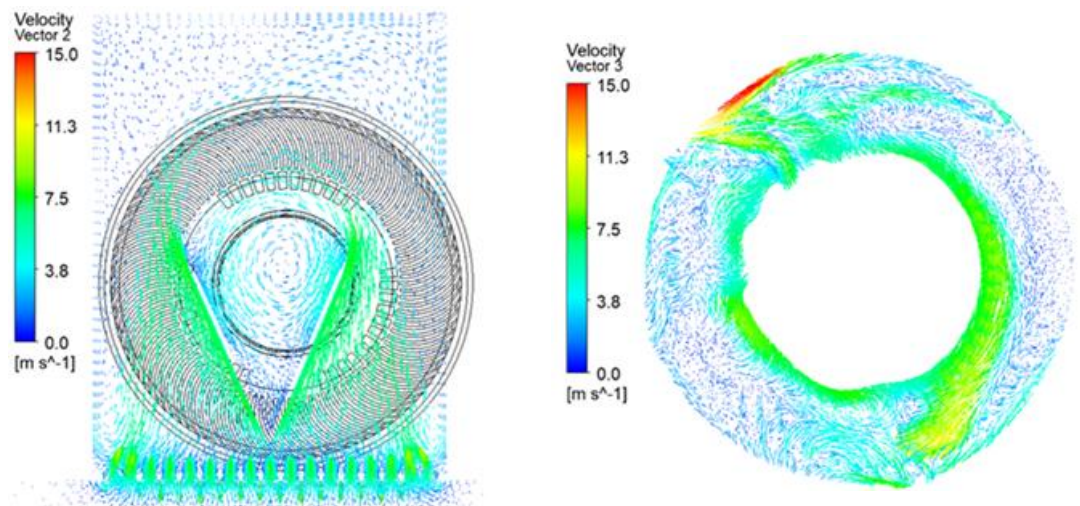
#### **3.3.1. Application of a V-shaped flow guide in the inlet chamber**

As shown in Figure 6, a V-shaped flow divider is installed inside the inlet chamber. First, this flow-straightening grille provides a guiding effect in the inlet-flow direction, enabling the airflow to obtain a stable inlet direction. Second, the V-shaped structure introduces a deflection angle in the horizontal direction, causing the high-speed airflow to enter from both sides of the inlet chamber and avoid the central region of the impeller, thereby reducing local blade pressure impingement



**Figure 6.** *V-shaped splitter cone in the intake chamber.*

A flow-field simulation was performed for this configuration, and the results shown in Figure 7 are generally similar to those of the V-shaped flow-guiding scheme. The high-speed airflow region is partially eliminated, and a certain low-speed vortex is formed. The high-speed incoming flow at the blade tip is also alleviated. In particular, the red high-velocity region in the upper-left corner of the blade-tip velocity vector plot is identified as a volute recirculation feature, which is concentrated outside the blade trailing edge and therefore does not interact with the blades to generate discrete tonal noise.



(a) *Velocity vector plot at the inlet chamber section*

(b) *Velocity vector plot at the blade-tip section*

**Figure 7.** *Velocity vector field for the V-shaped splitter cone strategy.*

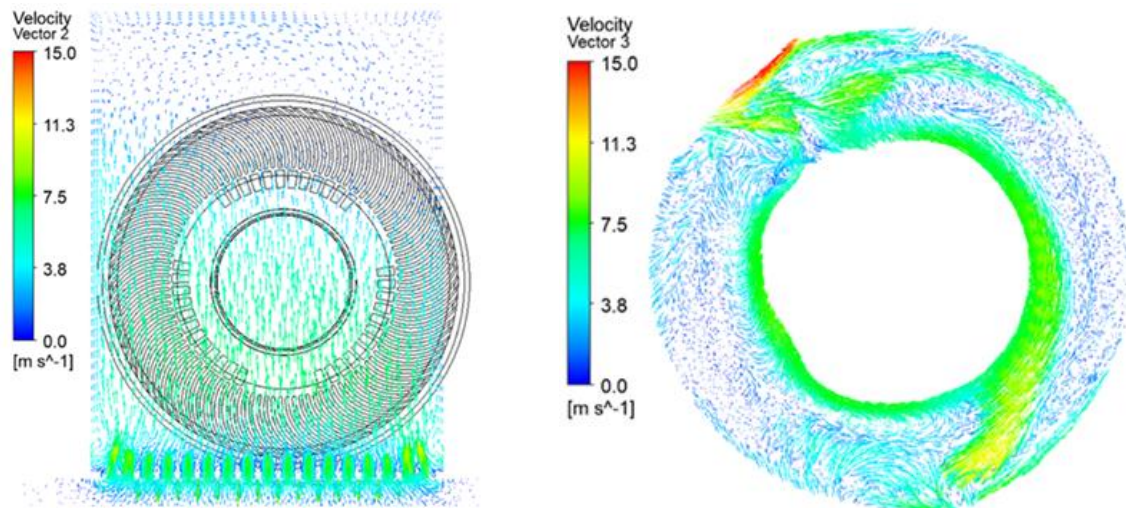
### 3.3.2. Application of a semi-shrouded flow collector

In addition to the pressure pulsations generated at the blade top by the airflow, the distorted inlet flow also causes pressure pulsations at the stationary volute tongue. As shown in Figure 8, an 8 mm shield was installed over an angular range of approximately  $30^\circ$  near the bellmouth adjacent to the collector, so that the wall-attached airflow impinges on the tongue.



**Figure 8.** Half-shielded bellmouth.

Flow-field simulation was carried out for this design. As shown in Figure 9, although the inlet chamber had been optimized, it still retained the characteristic that part of the inlet airflow impinges on the volute tongue. The flow field inside the inlet chamber still did not exhibit a fully uniform and stable pattern, and the maximum airflow velocity remained around 10 m/s. However, because a certain degree of shielding was added in the inlet impact zone, the high-speed airflow no longer affected the blade-top region, and the blade top still maintained a relatively uniform flow-field characteristic.



(a) Velocity vectors at the inlet chamber section

(b) Velocity vectors at the blade-tip section

**Figure 9.** Velocity vector field for the half-shielded bellmouth strategy.

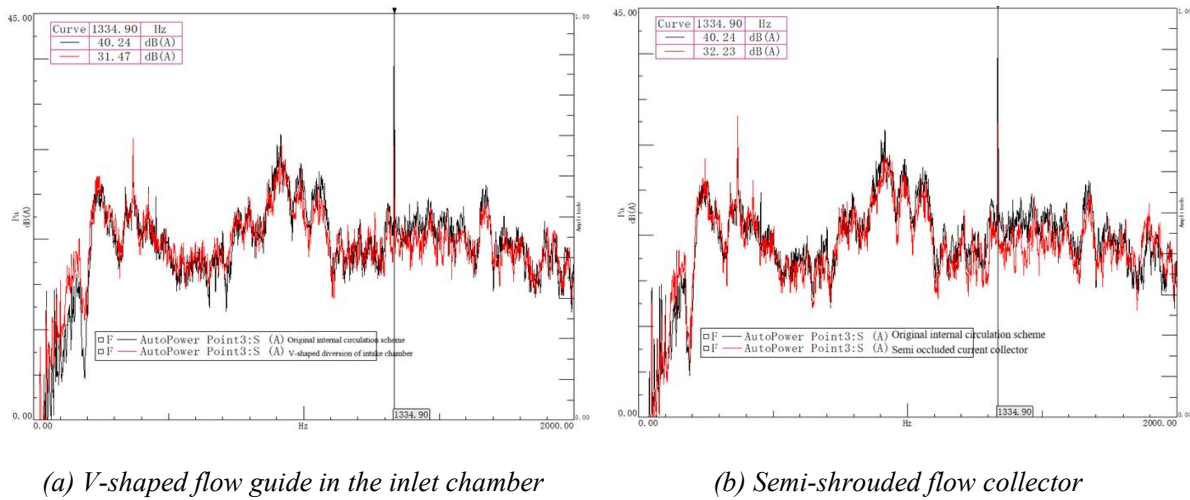
## 4. Results and Discussion

### 4.1. Comparative Verification of Rotational Noise Control Methods

To verify the effectiveness of the two noise-reduction strategies proposed above, prototype samples of each scheme were placed in a semi-anechoic chamber for noise spectrum testing under the internal circulation mode at a rotational speed of 1700 rpm. In combination with the CFD steady-flow simulation results, a comparative analysis of the flow-field characteristics was also conducted.

Figure 10 presents the measured noise spectra of the above schemes. Evidently, after optimization of the inlet-flow distortion, both schemes achieved significant suppression of the rotational noise and improved the acoustic quality of the fan. Among them, the V-shaped flow divider in the inlet chamber showed the best noise-

reduction performance, achieving a reduction of 8.8 dB at the discrete rotational-noise frequency.



**Figure 10.** Noise spectra of the optimized intake distortion strategies.

#### 4.2. Evaluation of Scheme Effectiveness and Engineering Application

By comprehensively considering both aerodynamic and acoustic performance, the engineering applicability of the proposed schemes can be evaluated.

The V-shaped flow divider is the optimal solution without changing the overall structural dimensions. This scheme exhibits superior performance: the discrete rotational-noise tone is reduced by 8.8 dB, while the airflow rate decreases by only 3 m<sup>3</sup>/h. No modification of the fan's external dimensions is required; only a V-shaped structure needs to be added inside the inlet chamber. In addition, the structure is simple, and the associated tooling, manufacturing, and modification costs are relatively low, making it suitable for low-cost retrofitting of existing fans. Furthermore, this structure occupies relatively little internal space in the inlet chamber and is therefore also suitable for fan designs with relatively large chamber volumes.

The semi-shielded collector is a low-cost modification scheme based on the existing collector. However, because the shield blocks part of the central inlet-flow region of the collector, the airflow loss is relatively large. At the same time, flow impingement in the shielded region may excite noise at other frequencies. Nevertheless, this scheme is highly cost-effective, and its noise-reduction performance is comparable to that of the flow-divider scheme. It is therefore suitable for engineering applications in which cost is a primary concern and airflow performance is not highly sensitive.

### 5. Conclusions

In this study, the abnormal discrete rotational noise of a forward-curved centrifugal fan used in the front-end fresh-air module under internal-circulation operation was investigated. By combining CFD numerical simulations with experimental validation, the underlying noise-generation mechanism and corresponding control strategies were systematically analyzed. The main conclusions are as follows:

The dominant mechanism of abnormal rotational noise under confined-space conditions was clarified. When the inlet chamber is narrow and no effective flow-guiding measures are provided, the sidewall-attached airflow in the inlet chamber

forms a high-speed impinging jet near the tongue side. This high-speed airflow periodically acts on the blade surfaces and induces strong pressure fluctuations, which is the key factor responsible for the abnormal discrete noise at 1334.9 Hz.

The limitations of conventional rotor–stator interference control methods were verified. Experimental results show that traditional measures for suppressing rotor–stator interaction, such as adjusting the circumferential tongue position or increasing the tongue clearance, are ineffective under severe inlet-flow distortion and cannot fundamentally eliminate this type of abnormal discrete noise.

An effective noise-control strategy based on inlet-flow rectification was proposed. By introducing a V-shaped flow divider and a semi-shielded flow-straightening structure at the air inlet and inside the inlet chamber, the inlet velocity field can be effectively homogenized, allowing the incoming flow to obtain a more concentrated velocity vector and thereby weakening the aerodynamic excitation acting on the blades.

Under the premise of maintaining stable system airflow performance, the peak rotational noise was significantly reduced by 8.8 dB, providing effective engineering guidance and theoretical support for the low-noise design of fans used in confined spaces in household appliances and HVAC applications.

## Conflicts of Interest

The authors declare that there is no conflict of interest regarding the publication of this article.

## Funding

This research received no specific grant from any funding agency in the public, commercial or not-for-profit sectors.

## References

- [1] Cheng, X.D. Centrifugal ventilator. Beijing: Chemical Industry Press, 2007; pp. 117-119.
- [2] Huang, Q.B.; Lu, W.Y.; Shi, H.M, et al. A study on the D-optimal acoustic model of worm tongue noise in centrifugal fan. *Journal of Vibration, Measurement&Diagnosis*, 1995, 15(4), 1-7.
- [3] Mao, Y.J.; Qi, D.T.; Xu, C.L, et al. Effect of interaction between centrifugal impeller and volute tongue on the performance and BPF noise. *Chinese Journal of Applied Mechanics*, 2006, 23(3), 368-372.
- [4] Han, Y.L.; Chen, E.Y.; Yang, A.L, et al. Influence of volute tongue structure on static interference noise of centrifugal fan. *Journal of Engineering for Thermal Energy and Power*, 2021, 36(6), 30-38.
- [5] Tai, G.L.; Wen, S.P.; Li, J.Y. Experiment and analysis on noise reduction of inclined volute tongue of multiwing centrifugal fan. *Fluid Machinery*, 1997, 25(6), 8-10.
- [6] Li, L.Y.; Han, J.A.; Zhang, W.Y, et al. Influence of groove on aerodynamic performances and noise of multi-blade centrifugal fan. *China Sciencepaper*, 2024, 19(07), 785-793.

- [7] Liu, X.M.; Li, S. Effects of bionic volute tongue bioinspired by leading edge of owl wing on aerodynamic performance and noise of multi-blade centrifugal fan. *Journal of Xi'an Jiaotong University*, 2015, 49(1), 14-20, 33.
- [8] Liu, X.M.; Liu, X, Li, D, et al. Numerical study of the effects of bionic blade on the aerodynamic noise of multi-blade centrifugal fan. *China Sciencepaper*, 2015, 10(11), 1309-1315.



© 2026 by the author(s); licensee International Technology and Science Publications (ITS), this work for open access publication is under the Creative Commons Attribution International License (CC BY 4.0). (<http://creativecommons.org/licenses/by/4.0/>)