

Article

Effects of Hub Geometry on the Aerodynamic and Acoustic Performance of Axial Flow Fans

Weihao Zhang, Renkui Tang, Yang Yu * and Yonghua Li 

Zhan Tianyou Honors College (College of China Railway Rolling Stock Corporation), Dalian Jiaotong University, Dalian 116028, China; zhangweihao_666@163.com (W.Z.); tangrenkui2025@163.com (R.T.); yonghuali@163.com (Y.L.)

* Correspondence: yuyang_1991131@163.com

Abstract

Axial flow fans are widely used in high-speed train cooling and ventilation systems, where both static efficiency and noise reduction are critical performance requirements. In this study, the effects of hub geometry variation on the aerodynamic and acoustic characteristics of an axial flow fan are numerically investigated through three-dimensional simulations. Five fan configurations with different hub angles are analyzed under identical operating conditions. Steady aerodynamic performance is first evaluated using the Reynolds-averaged Navier–Stokes (RANS) approach with the $k-\omega$ shear stress transport (SST) turbulence model. The unsteady flow field is then resolved using large eddy simulation (LES) to capture the vortex structures and blade surface pressure fluctuations responsible for noise generation. The far-field aerodynamic noise is predicted based on the Ffowcs Williams–Hawkings (FW–H) acoustic analogy, and both tonal and broadband noise characteristics are analyzed using multiple virtual microphones. The results show that reducing the hub angle leads to improved aerodynamic performance at lower volumetric flow rates. Meanwhile, a reduction in tonal noise at the blade-passing frequency (BPF) and broadband noise at higher frequencies is observed. The findings demonstrate that appropriate hub angle design provides an effective approach for the simultaneous improvement of static efficiency and the reduction of aerodynamic noise of axial-flow fans used in high-speed train applications.

Keywords: axial flow fan; $k-\omega$ shear stress transport (SST); large eddy simulation (LES); Ffowcs Williams–Hawkings (FW–H); aerodynamic performance; aerodynamic noise



Academic Editor: Alberto Benato

Received: 7 January 2026

Revised: 13 February 2026

Accepted: 20 February 2026

Published: 25 February 2026

Copyright: © 2026 by the authors.

Licensee MDPI, Basel, Switzerland.

This article is an open access article distributed under the terms and conditions of the [Creative Commons Attribution \(CC BY\)](https://creativecommons.org/licenses/by/4.0/) license.

1. Introduction

High-speed rail systems have become an important mode of modern transportation, for which passenger comfort and environmental sustainability are key competitive factors [1]. In this context, axial flow fans, as essential components of the environmental control systems of high-speed trains, play a crucial role in driving air circulation within passenger carriages and dissipating heat from onboard equipment. Consequently, improving the aerodynamic and acoustic performance of axial flow fans has attracted increasing research attention.

A substantial body of literature has focused on the influence of blade design on axial fan performance. Ding et al. [2] investigated the effects of radial twist distributions by constructing different blade installation angle profiles and demonstrated that twist regulation in distinct radial zones significantly alters the internal flow structure. To further

enhance aerodynamic performance, Ye et al. [3] modified blade tip structures and analyzed the influence of tip slot configurations on fan characteristics. Subsequently, the same research group proposed a novel serrated Gurney flap with a height approximately 0.8% of the chord length, achieving simultaneous improvements in efficiency and total pressure rise while reducing high-frequency noise emissions [4]. Park et al. [5] reported that forward-swept blades exhibit superior aerodynamic and acoustic performance compared with straight and backward-swept blades under most system resistance conditions. In addition, Chen et al. [6] employed a Taguchi loss function to optimize blade installation angles, leading to a bio-inspired fan design with enhanced flow rate and significantly reduced noise. Lee et al. [7] developed a response surface-based optimization framework using the RANS method with the SST turbulence model, resulting in the improved efficiency of a low-speed axial flow fan through blade geometry optimization. Meyer et al. [8] conducted RANS based analyses of subsonic axial flow fans and applied multi-objective numerical optimization to blade tip geometries, achieving improvements in total static pressure and static efficiency.

Beyond blade modifications, hub-related structures have also been recognized as important factors affecting fan performance. The hub cap, typically located upstream of the rotor and often integrated with supporting struts, guides the inflow toward the rotor and influences the near-hub flow field [9]. Shi et al. [10] generated hub caps with different length ratios and geometric profiles using polynomial functions and numerically investigated their effects on root flow characteristics and overall aerodynamic performance. An analytical model based on velocity non-uniformity was further proposed to predict the critical hub cap length ratio, enabling low-loss and compact designs. Similarly, Jung et al. [11,12] examined vortex structure variations under different inlet hub lengths and found that losses near the hub zone were substantially reduced at a specific hub length, leading to improved fan efficiency. Regarding noise characteristics, Wang [13] analyzed tandem small-scale axial cooling fans and demonstrated that installing a rectifier between the upstream and downstream fans weakened non-axial flow and reduced broadband noise by approximately 3.5 dB under partially blocked conditions.

The influence of downstream guide vanes has also been widely investigated. Located downstream of the rotor, guide vanes are primarily designed to straighten the flow and reduce pressure losses [9]. Chen et al. [14] performed RANS-based numerical simulations on guide vane geometries and showed that reducing chord length while increasing vane number could achieve higher airflow rates at the same rotational speed. Krasnyuk et al. [15] proposed an efficient reverse operation strategy for axial fans using rotatable guide vanes and designed a fan with similar performance characteristics in both forward and reverse flow directions. Jin et al. [16] investigated the multi-objective optimization of outlet guide vanes for nuclear-grade axial fans, showing that appropriate design optimization can simultaneously improve aerodynamic performance and reduce noise levels.

For axial flow fans with small hub-to-tip diameter ratios, downstream backflow and hub zone flow separation are commonly observed. Vortex distribution-based design methods have been proposed by Wang et al. [17] to optimize the hub-to-tip ratio and improve aerodynamic performance. In such low hub-to-tip configurations, classical guide vane designs often fail to increase static efficiency due to hub recirculation, and optimized guide vane geometries have been developed by Angelis et al. [18] to mitigate this effect and enhance performance. Computational fluid dynamics (CFD) based on RANS has been demonstrated by Wang et al. [19] to accurately analyze the aerodynamic performance of small hub-to-tip axial flow fans. Numerical methods, including LES and semi-empirical models, have been employed by Carolus et al. [20] to predict broadband noise, highlighting the role of ingested turbulence on unsteady blade forces. LES has further been shown by

Reese et al. [21] to capture the effects of inflow turbulence and blade-to-blade or blade-to-hub interactions on noise generation. Combined LES and RANS approaches have also enabled scale-model-based noise predictions for full-scale fans by Du Plessis et al. [22], capturing contributions from tip vortices, turbulence ingestion, and blade interactions.

Although extensive efforts have been devoted to blade modifications, hub caps, and guide vanes, the influence of hub geometry variations, particularly variations in hub-to-tip ratio on coupled aerodynamic and acoustic performance, remains insufficiently understood. Most existing studies focus on optimizing individual components or identifying a single optimal configuration, whereas the systematic impact of varying hub geometry on flow structures, aerodynamic performance, and noise radiation has received limited attention.

Therefore, the present study investigates an axial flow fan used in high-speed train applications and systematically examines the effects of a conical hub design, which introduces a spanwise-varying hub-to-tip ratio, through three-dimensional numerical simulations. Five hub configurations, including the baseline design, are analyzed under identical operating conditions. The influences of the conical-hub-induced hub-to-tip ratio variations on the internal flow field, aerodynamic performance, and noise radiation characteristics are quantitatively evaluated and compared. The objective of this work is to clarify the role of conical hub geometry and the associated hub-to-tip ratio variation in the integrated aerodynamic and acoustic performance of axial flow fans, thereby providing practical guidance for fan design and optimization. In this study, the conical hub geometry is parameterized by the hub angle, which directly controls the effective hub-to-tip ratio. Therefore, the hub angle is adopted as the governing geometric parameter in the following analysis.

2. Computational Strategy

2.1. Physical Model

As illustrated in Figure 1, the axial flow fan employed in the high-speed train is composed of an impeller, guide vanes, and a diffuser. The configuration includes seven rotating blades and eight guide vanes. The rotating blades are designed with an airfoil profile to enhance aerodynamic performance, whereas the guide vanes are configured as arc plates with uniform thickness to effectively direct the airflow. The flow rate of $8 \text{ m}^3/\text{s}$ corresponds to the design operating condition of the fan, which is used as the reference point for evaluating the aerodynamic and acoustic performance of different hub geometries. The principal geometric and operating parameters are summarized in Table 1.

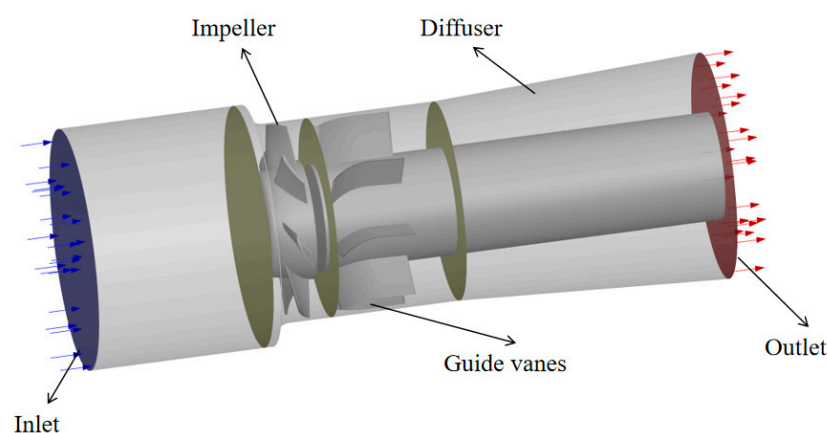


Figure 1. Schematic representation of the axial flow fan applied in high-speed railway systems.

As shown in Figure 2, the fan impeller consists of a hub cap, blades, and a cylindrical hub. In this study, the hub angle is defined as the angle, measured on an axial cross section, between the diameter of the lower base and the generatrix of the cylindrical hub,

as illustrated in Figure 2. To investigate the influence of hub geometry on aerodynamic performance, four additional optimized models, besides the original impeller, are examined, with hub angles of 87.5° , 85° , 82.5° , and 80° , respectively, as shown in Figure 3. Since the hub adopts a conical geometry, the hub diameter varies along the axial direction, resulting in a non-uniform hub-to-tip ratio within the rotor passage. Therefore, three representative axial cross-sections (Section 1, Section 2, and Section 3) are defined in Figure 2, and the corresponding hub-to-tip ratios for different hub angles are summarized in Table 2 to quantitatively describe this geometric variation.

Table 1. Main parameters of the fan.

Parameters	Values
Rotating speed	1270 rpm
Static pressure rise	1049.64 Pa
Designed volumetric flow rate	$8 \text{ m}^3/\text{s}$
Designed static efficiency	60.63%
Chord length	148.06 mm
Root pitch angle	36.51°
Middle pitch angle	30.44°
Tip pitch angle	22.04°
Tip clearance	5 mm
Impeller diameter	665 mm
Hub diameter	360 mm

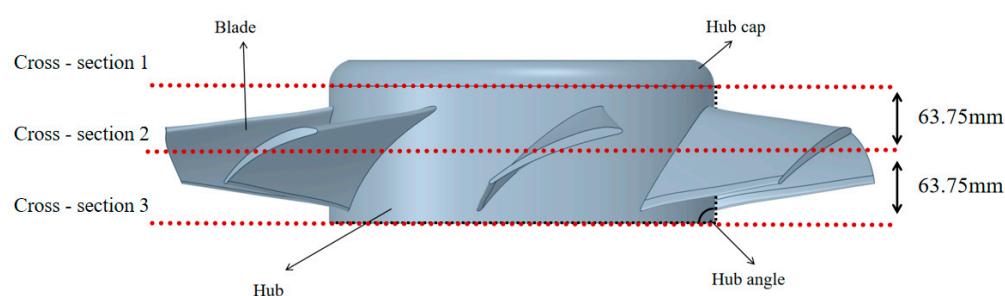


Figure 2. Schematic diagram of the fan impeller.

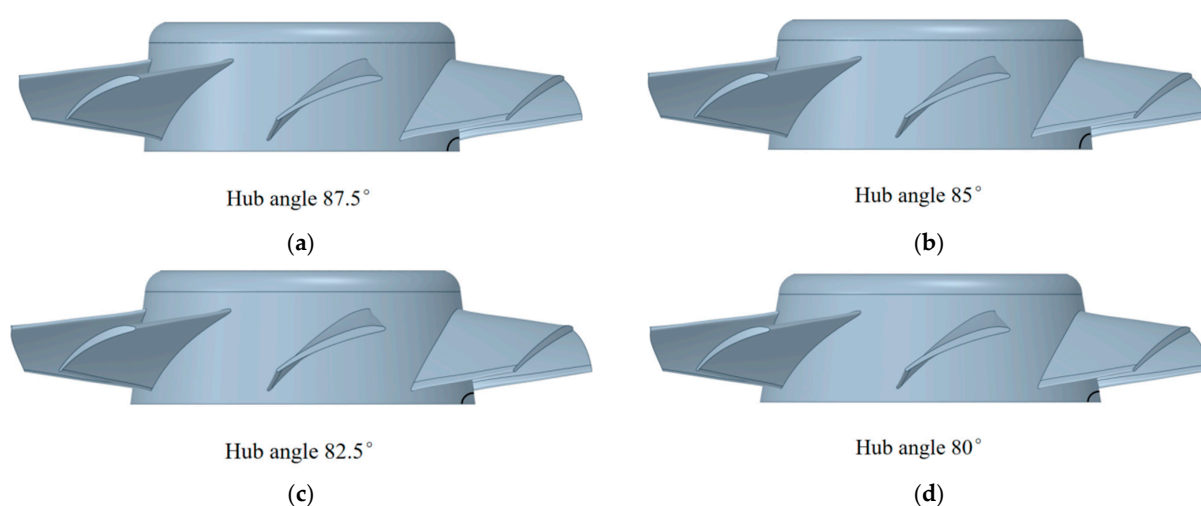


Figure 3. Schematic diagram of four different hub angles.

It can be observed that although the hub-to-tip ratio at Section 1 remains unchanged, reducing the hub angle significantly increases the hub-to-tip ratio at downstream sections. This demonstrates that the hub angle directly controls the axial distribution of the hub diameter within the rotor passage.

Table 2. Hub-to-tip ratios at different axial cross-sections for various hub angles.

Hub Angle (°)	Hub-to-Tip Ratio Cross-Section 1	Hub-to-Tip Ratio Cross-Section 2	Hub-to-Tip Ratio Cross-Section 3
90	0.54	0.54	0.54
87.5	0.54	0.55	0.56
85	0.54	0.56	0.58
82.5	0.54	0.57	0.59
80	0.54	0.59	0.61

2.2. Numerical Method and Boundary Conditions

In this study, the commercial computational fluid dynamics (CFD) solver ANSYS Fluent 24.1 is employed to simulate the three-dimensional flow field of the axial fan. Fluent, as a general CFD platform based on the finite volume method, is capable of generating unstructured meshes for complex geometries and achieving efficient flow-field solutions. To accurately capture the complex flow characteristics near the wall zones, the Reynolds-averaged Navier–Stokes (RANS) method is adopted. Based on the blade chord length and the characteristic inflow velocity, the preliminary Reynolds number of the present fan is estimated to be on the order of 1.2×10^6 , indicating a fully turbulent flow regime. For turbulence modeling, the shear stress transport (SST) k - ω model is selected to calculate turbulent viscosity. Compared with the conventional k - ϵ model, the k - ω model provides more accurate predictions of boundary layers under low-Reynolds-number conditions. The k - ω SST turbulence model employs the k - ω formulation in the near-wall zone and switches to the k - ϵ formulation in the far-field zone, effectively combining the advantages of both. This enables more reliable predictions of hub flow characteristics and the effects of shed vortices. It should be noted that this model imposes relatively strict requirements on near-wall mesh resolution, and the first layer of mesh is typically required to be located within the viscous sublayer to ensure accurate calculation of separated flows.

The governing equations of the k - ω SST turbulence model are formulated as follows [12]:

k transport equation:

$$\frac{\partial \rho k}{\partial t} + \frac{\partial u_i \rho k}{\partial x_i} = \frac{\partial}{\partial x_i} \left(\frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_i} \right) + \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} + g_i \beta \frac{\mu_t}{\sigma_t} \frac{\partial T}{\partial x_j} - \rho \epsilon \quad (1)$$

ω transport equation:

$$\frac{\partial}{\partial t} \rho \omega + \frac{\partial}{\partial x_j} u_j \rho \omega = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + \frac{\gamma \rho}{\mu_t} \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} - \beta \rho \omega^2 \quad (2)$$

The eddy viscosity is defined as:

$$\mu_t|_{k-\omega} = \rho \frac{k}{\omega} \quad (3)$$

In the above equations, ρ denotes the fluid density, u_i represents the mean velocity component in the Cartesian coordinate system, and μ and μ_t denote the molecular and eddy viscosities, respectively. The turbulent kinetic energy and the dissipation rate per unit of turbulent kinetic energy are denoted by k and ω , respectively. $\sigma_t, \beta, \sigma_k, \sigma_\omega, \gamma$ are empirical coefficients.

The overall computational domain is divided into four parts: the inlet domain, the rotating domain of the impeller, the stationary domain of the guide vanes, and the outlet domain. Interface surfaces are set between adjacent zones. For the steady-state simulations,

the coupled scheme is employed for the pressure–velocity coupling. To improve numerical accuracy, a second-order upwind discretization is applied for the convection terms. The effects of gravity and surface roughness are neglected in the flow-field calculations, and air is assumed to behave as an ideal gas. A pressure inlet (0 Pa) and a mass-flow outlet are applied to replicate the actual operating conditions of the axial fan as closely as possible. The simulations are considered converged when the residuals of all monitored parameters fall below 10^{-4} . The division of the computational domains is illustrated in Figure 4.

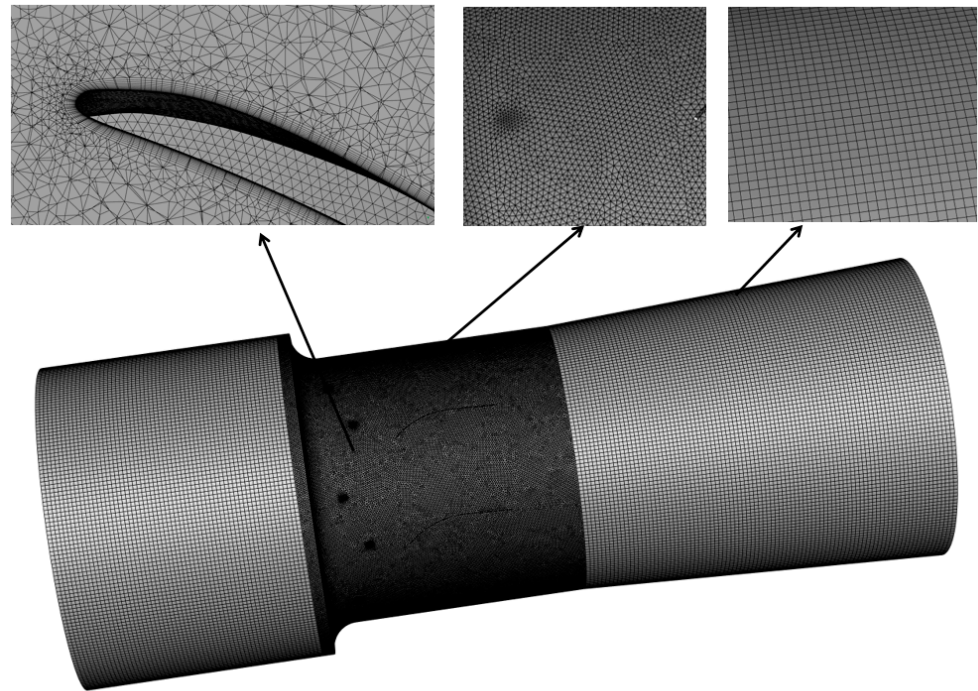


Figure 4. Overall computational domain and mesh division in different zones.

The LES approach is employed to resolve the large-scale turbulent structures while modeling the effects of unresolved subgrid-scale (SGS) motions; the fully developed LES computations are initialized from steady RANS results, with the wall-adapting local eddy-viscosity (WALE) model adopted to accurately capture pressure fluctuations.

The governing equations of the LES method are described as [23]:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \quad (4)$$

$$\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \nu \frac{\partial^2 \bar{u}_i}{\partial x_j \partial x_j} - \frac{\partial \tau_{ij}}{\partial x_j} \quad (5)$$

where $\bar{u}_i$ and $\bar{p}$ denote the filtered velocity and pressure, respectively, and τ_{ij} represents the SGS stress tensor, defined as:

$$\tau_{ij} = \bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j \quad (6)$$

The SGS stress is closed using the eddy-viscosity assumption:

$$\tau_{ij} = \frac{1}{3} \tau_{kk} \delta_{ij} - 2\nu_t \bar{S}_{ij} \quad (7)$$

where $\bar{S}_{ij}$ is the resolved strain-rate tensor. In the present study, the wall-adapting local WALE model proposed by Nicoud and Ducros [24] is adopted to evaluate the SGS eddy viscosity:

$$\nu_t = (C_w \Delta)^2 \frac{(S_{ij}^d S_{ij}^d)^{3/2}}{(\bar{S}_{ij} \bar{S}_{ij})^{5/2} + (S_{ij}^d S_{ij}^d)^{5/4}} \quad (8)$$

The traceless symmetric part of the square of the velocity gradient tensor, S_{ij}^d , is defined as:

$$S_{ij}^d = \frac{1}{2} (g_{ik} g_{kj} + g_{jk} g_{ki}) - \frac{1}{3} \delta_{ij} g_{kk}^2 \quad (9)$$

$$g_{ij} = \frac{\partial \bar{u}_i}{\partial x_j} \quad (10)$$

The model constant is set to $C_w = 0.325$, and the local filter width is defined as $\Delta = V^{1/3}$. The WALE model implementation provided in Fluent is employed in the present simulations.

In the present study, the unsteady flow field is resolved using LES in conjunction with the Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) algorithm to ensure robust coupling between the pressure and velocity fields. A second-order implicit temporal discretization scheme is adopted to improve the accuracy of the aerodynamic noise prediction. The transient computation employs a time step of 4.0×10^{-5} s, with a total of 5000-time steps performed to capture the unsteady flow evolution.

2.3. Meshing and Independence Verification

The three-dimensional models with variable hub geometries are constructed using SpaceClaim, and the computational mesh is generated in ANSYS Meshing. To reduce the total cell count while maintaining sufficient computational accuracy, structured hexahedral elements are applied in the inlet and outlet domains. In contrast, the rotating domain and the stationary impeller domain, where the flow is more complex, are discretized with unstructured tetrahedral elements to improve mesh quality. Furthermore, since the rotating domain represents the primary zone of interest in this study and exhibits the most intricate flow behavior, local mesh refinement is employed to enhance the resolution of critical flow features.

For turbulence modeling, the near-wall mesh resolution is commonly evaluated using the non-dimensional wall distance, y^+ . The required y^+ value depends strongly on the turbulence model employed. For the k- ω model, particularly the SST variant, accurate integration of the boundary layer necessitates that the first grid point be located within the viscous sublayer, corresponding to $y^+ < 1$. In contrast, LES imposes even stricter requirements on near-wall resolution, as a broad range of vortical structures must be directly resolved, and the accuracy of near-wall flow prediction therefore relies heavily on the computational grid. Accordingly, for wall-resolved LES, the first cell center should also satisfy $y^+ \approx 1$ [25]. To ensure the reliability of the present simulations and to simultaneously meet the near-wall resolution requirements of both the k- ω SST model and LES, the mesh density is designed to maintain $y^+ < 1$ throughout the wall-adjacent zone. Specifically, the first-layer grid height is set to 2.0×10^{-5} m, with a growth rate of 1.2 and a total of 18 inflation layers. The detailed mesh schematic is shown in Figure 4.

Considering the wide range of models and operating conditions analyzed in this study, the mesh size is optimized to achieve a balance between computational accuracy and efficiency, minimizing simulation time without compromising precision. Accordingly, five different meshes are generated for the original model, containing 1.24 million, 4.81 million, 14.31 million, 16.53 million, and 19.15 million elements, respectively. The static pressure difference between the inlet and outlet is calculated for each mesh under the design condition. As shown in Figure 5, a mesh independence analysis based on the static pressure

difference indicates that the simulation results exhibit negligible variations when the total number of elements exceeds 16.53 million. Using meshes larger than 16.53 million would significantly increase the computational cost; therefore, the 16.53-million-element mesh is selected for all subsequent studies.

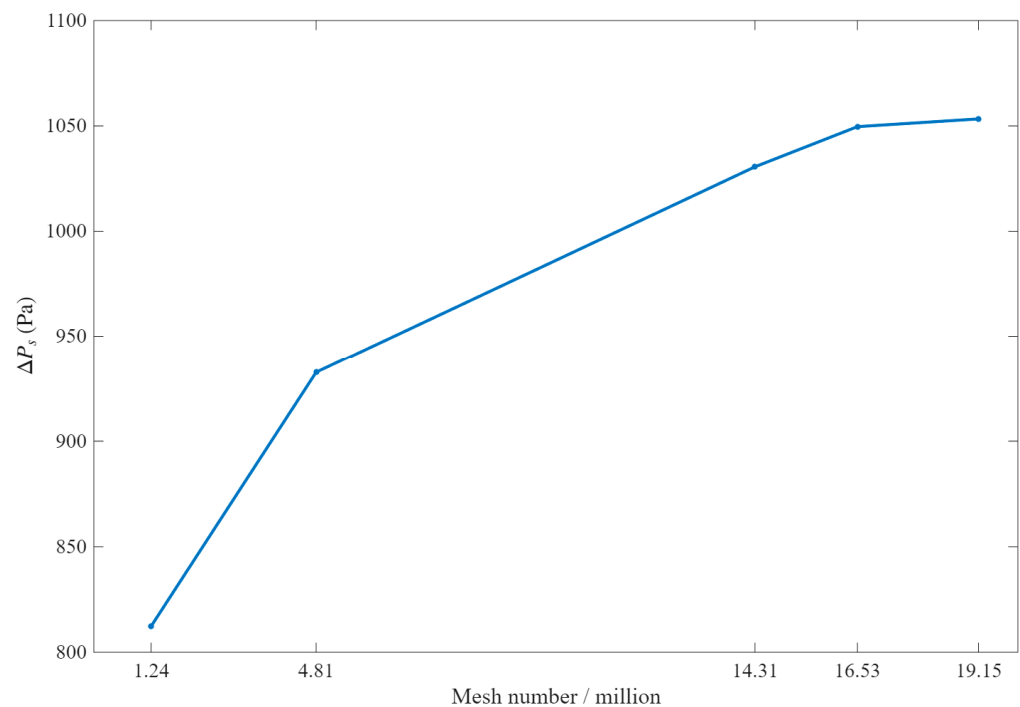


Figure 5. Mesh-independent verification.

3. Results and Discussion

3.1. Flow Characterization

3.1.1. Aerodynamic Performance

The static pressure rise is an important indicator for evaluating the aerodynamic performance of a fan, as it reflects the ability of the fan to increase the static pressure of the airflow through mechanical energy input during operation. A higher static pressure rise enables the fan to more effectively overcome system resistance and ensure stable airflow transport. However, the static pressure rise alone cannot fully represent the overall performance of the fan; therefore, it is usually analyzed in conjunction with static efficiency. All aerodynamic performance data presented in this section are obtained from steady-state simulations using the $k-\omega$ SST turbulence model under the rotating reference frame (MRF) approach. The expression for static efficiency is given as follows:

$$\eta_s = \frac{\Delta P_s Q}{P_{sh}} \quad (11)$$

Shaft power P_{sh} is given by:

$$P_{sh} = \frac{n}{9550} M \quad (12)$$

where ΔP_s denotes the static pressure difference between the fan outlet and inlet, Q represents the volumetric flow rate, and P_{sh} is the shaft power. In addition, n denotes the rotational speed of the impeller, and M refers to the torque acting on the impeller.

Since reducing the hub angle modifies the flow passage width and thereby reduces the maximum volumetric flow rate, this parameter is evaluated first for the five models prior to the analysis of the aerodynamic performance of the optimized configurations. During

the calculation of the maximum volumetric flow rate, both the inlet and outlet boundaries are set to zero pressure. As shown in Figure 6, the maximum volumetric flow rates of the five models with hub angles of 90° , 87.5° , 85° , 82.5° , and 80° are 11.06, 10.96, 10.85, 10.65, and $10.49 \text{ m}^3/\text{s}$, respectively. To ensure that the optimized models maintain a sufficiently broad operating range, this study limits the analysis to configurations with a hub angle no smaller than 80° .

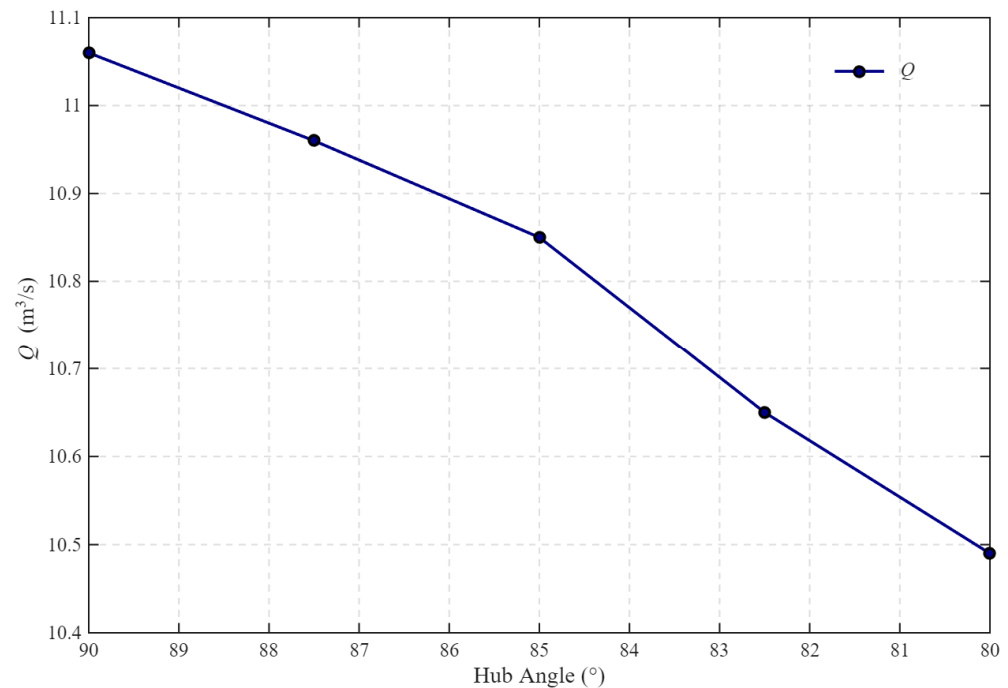


Figure 6. Maximum volumetric flow rates of the five models.

As shown in Figure 7, the static pressure–volumetric flow rate curves of the five fan models, are presented. The original impeller with a hub angle of 90° exhibits relatively poor aerodynamic performance at the required operating condition of $8 \text{ m}^3/\text{s}$, performing significantly worse compared with its behavior at higher volumetric flow rates. In contrast, the impeller with a hub angle of 80° shows a 22.06% improvement in aerodynamic performance at the same design condition relative to the baseline model. Moreover, the peak static pressure point of the 80° impeller is closer to the required design operating condition. Interestingly, as illustrated in Figure 7, a clear trend can be observed among the five models: as the hub angle decreases, the peak static pressure point gradually shifts toward lower volumetric flow rates. This observation indicates that, as the hub angle decreases, the fan static pressure rise is enhanced under a low volumetric flow rate regime.

However, this trend is accompanied by a gradual reduction in static pressure rise in the high-flow-rate zone. As the hub angle becomes smaller, the increased flow resistance within the rotor passage weakens the blade’s ability to effectively perform work on the airflow at larger volumetric flow rates, leading to a deterioration in aerodynamic performance under high-flow operating conditions.

Figure 8 shows that when the volumetric flow rate Q exceeds $8.6 \text{ m}^3/\text{s}$, the static efficiency of the original impeller is higher than that of the other four models. However, as the volumetric flow rate decreases, the efficiencies of the modified impellers gradually surpass that of the original model. When the airflow falls below $8.4 \text{ m}^3/\text{s}$, all four impellers with adjusted hub angles exhibit higher efficiencies than the original impeller. This indicates that an appropriate reduction in the hub angle can effectively improve fan efficiency under low volumetric flow rate operating conditions.

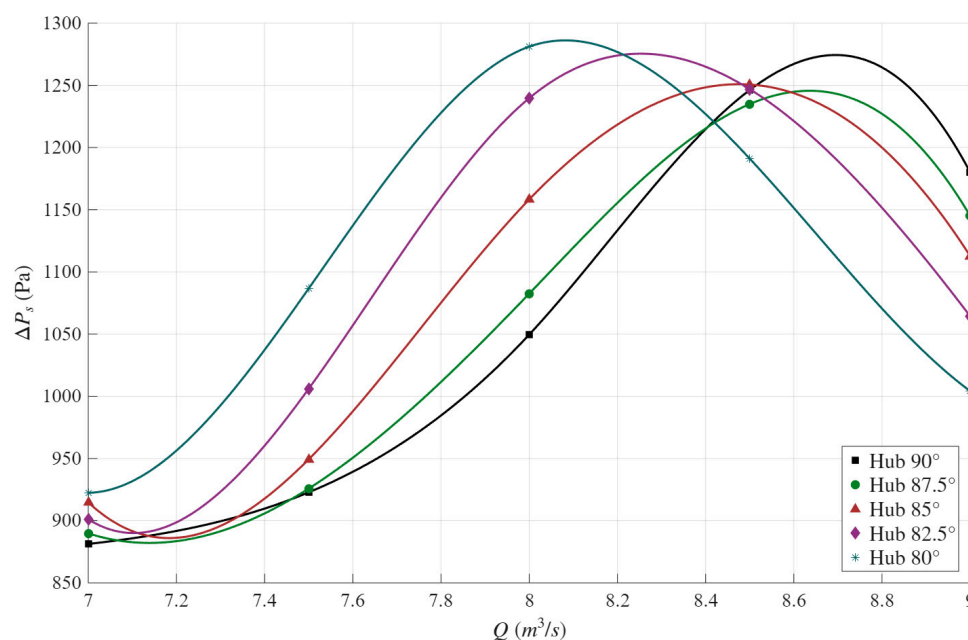


Figure 7. Static pressure rise and volumetric flow rate characteristic curves of the five models.

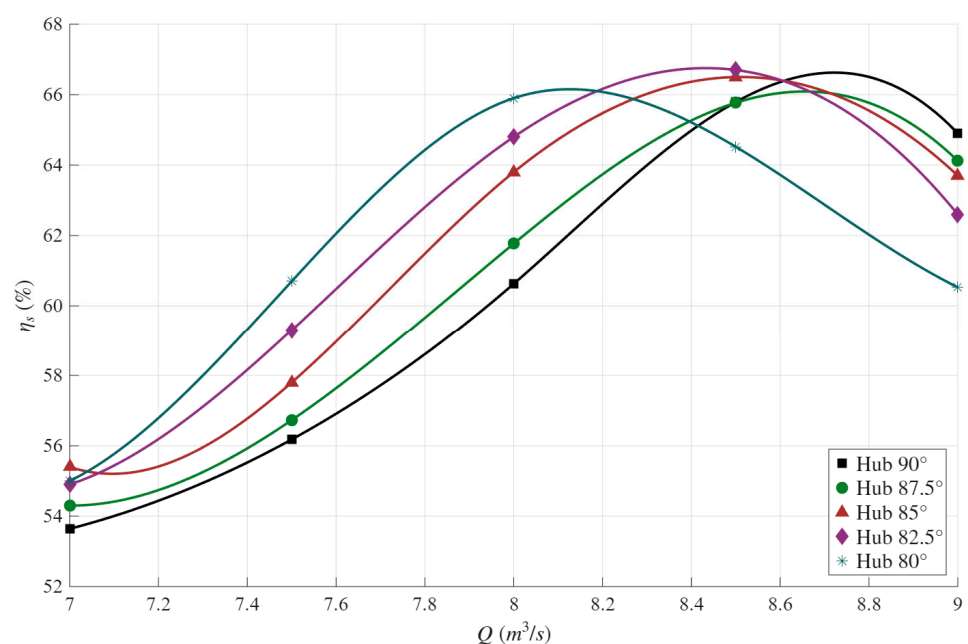


Figure 8. Static efficiency curves of the five models.

Notably, at the design operating point, the impeller with an 80° hub angle achieves peak efficiency, showing a 5.27% improvement compared with the original model at the same condition. Although the 80° model exhibits lower efficiency than the other four models, including the original, when the volumetric flow rate exceeds 8.2 m³/s, it demonstrates superior performance near the design point. This phenomenon further confirms the existence of a design trade-off associated with hub angle reduction. While smaller hub angles significantly enhance efficiency in the low-flow regime and near the design operating condition, they simultaneously lead to a gradual efficiency deterioration in the high-flow zone due to increased passage resistance and weakened blade work capability. Therefore, for the purposes of this study, the 80° hub angle impeller is considered the preferred configuration.

3.1.2. Static Pressure and Velocity Distributions

To provide a clearer visualization of the aerodynamic variations among different hub angle configurations under the design operating condition, cylindrical surface-based unwrapped contour maps are generated at several blade height locations. Specifically, three cylindrical surface positions, corresponding to 46%, 78%, and 92% of the original blade height, are selected for analysis. The resulting unwrapped distributions are shown in Figure 9.

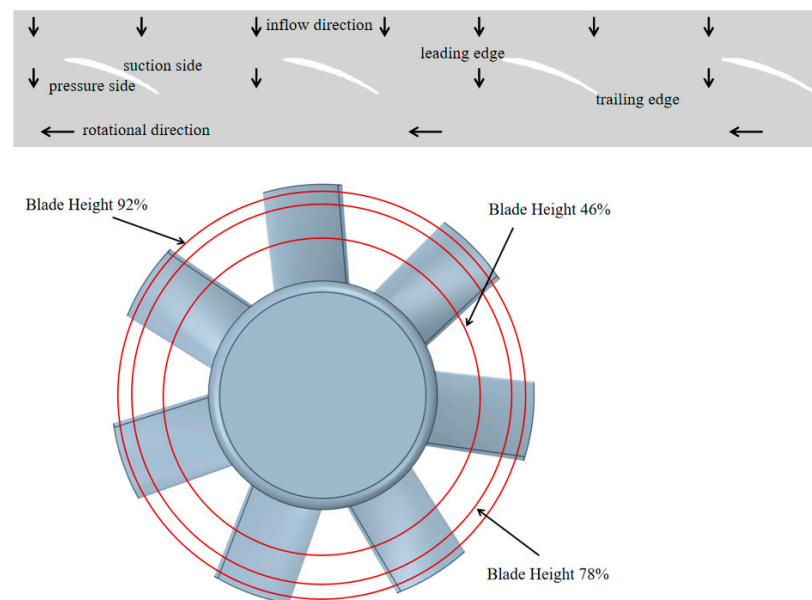


Figure 9. Schematic diagram of unwrapped contours at different blade heights.

Figure 10 displays the static pressure contours at blade heights of 46%, 78%, and 92% for various hub angles. At the 46% blade height, the overall static pressure field exhibits minimal variation as the hub angle decreases. The static pressure distribution remains relatively uniform across all configurations, except for a distinct pressure separation line that forms near the trailing edge outlet when the hub angle is reduced to 85° . This observation suggests that, for hub angles smaller than 85° , sufficient kinetic energy remains near the pressure-side trailing edge to maintain a high static pressure, whereas for hub angles larger than 85° , the near-wall flow at the trailing edge loses kinetic energy earlier and exhibits a more diffuse pressure field.

At the 78% blade height, when the hub angle decreases to 87.5° , the pressure rise within the blade passage becomes more clearly defined, and the high-pressure zone on the pressure side exhibits a more distinct contrast with the neighboring low-pressure zone on the suction side. This indicates that a stronger pressure gradient is formed at this height as the hub angle decreases, which may enhance the formation of vortices in this zone.

The most pronounced static pressure variations appear at the 92% blade height. As the hub angle decreases, the originally broad and diffused low-pressure zone located on the suction-side trailing edge contracts into a narrower and more elongated low-pressure band. This band also shifts slightly toward the axial direction, reflecting changes in the local flow turning near the blade tip. In contrast, the high-pressure zone on the pressure side expands progressively toward the trailing edge. The high-pressure zone near the pressure-side leading edge also grows substantially in both area and intensity, indicating a stronger pressure build-up as the hub angle is reduced.

Another observable phenomenon is the appearance of several localized high-pressure zones near the lower portion of the contour at small hub angles. These concentrated zones arise because a reduced hub angle causes the flow leaving the blade to impinge on the hub

wall, leading to localized high-pressure accumulation on the hub wall downstream of the blade passage.

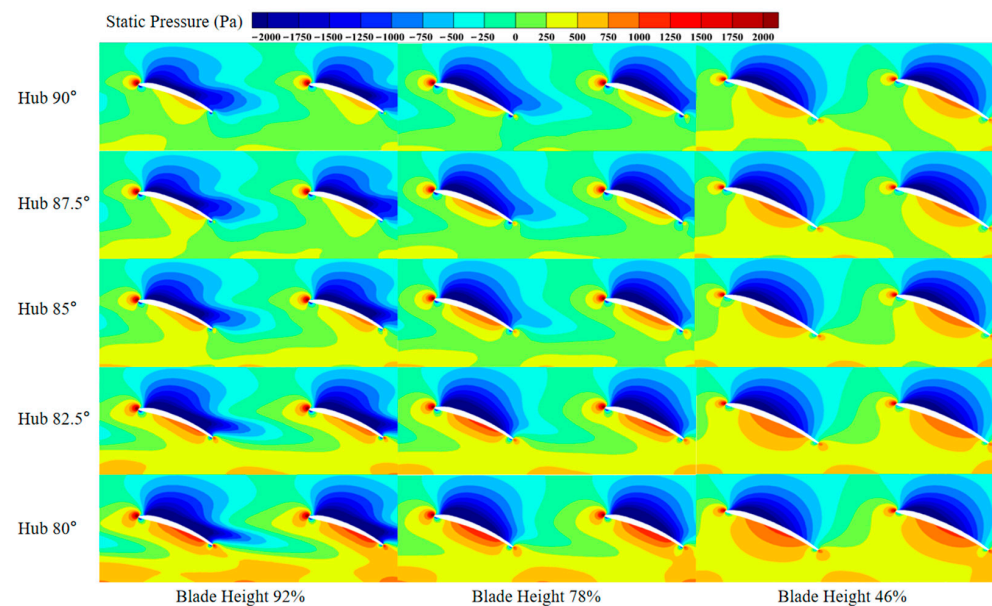


Figure 10. Static pressure contours at different blade spans for varying hub angles.

As shown in Figure 11, the unwrapped contours illustrate the axial velocity distributions of the 90° impeller under different volume flow rates. From the low flow rate of 7 m³/s to the high flow rate of 9 m³/s, the axial velocity distribution in the 90° impeller passage exhibits a clear evolution along the blade height at 46%, 78%, and 92%.

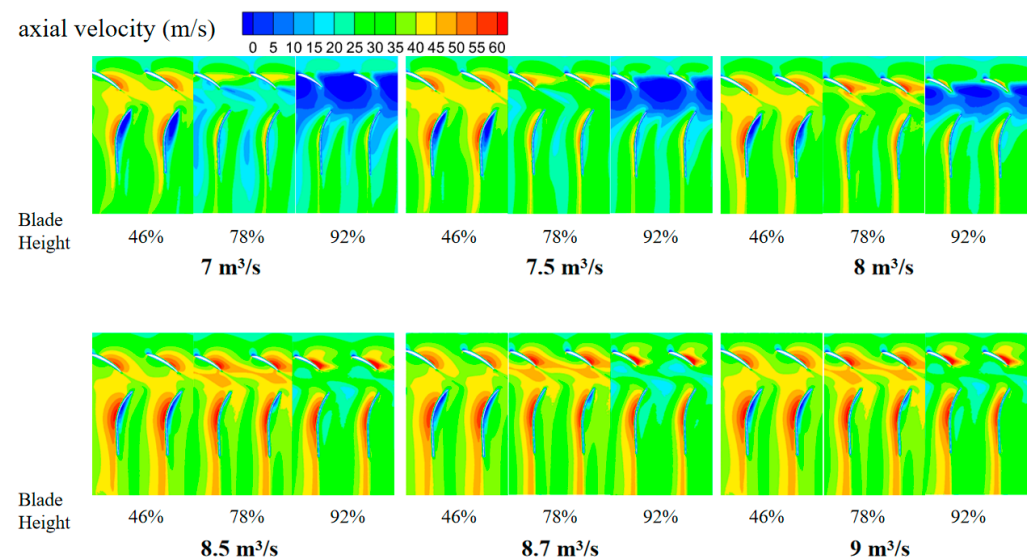


Figure 11. Axial velocity distributions at different volume flow rates for the 90° impeller.

At 7 m³/s, a large zone of low axial velocity occupies the suction side and near-hub zone, indicating insufficient momentum exchange and the presence of weak flow zones. As the flow rate increases to 7.5 m³/s and 8 m³/s, the low-velocity zone gradually shrinks and the velocity distribution becomes more uniform, suggesting improved flow attachment and energy transfer.

At 8.5 m³/s, the low-velocity zones are further suppressed, and the mainstream flow occupies most of the passage. At 8.7 m³/s, corresponding to the best efficiency point,

the low-kinetic-energy air is minimized across all blade heights, and the axial velocity field becomes the most uniform and continuous. This observation is consistent with the performance curve results, where the fan reaches maximum efficiency due to the reduced presence of low-momentum air.

When the flow rate increases to $9 \text{ m}^3/\text{s}$, localized non-uniformity reappears near the blade-tip zone, indicating the onset of flow deterioration under high-flow conditions.

As shown in Figure 12, the unwrapped contours present the axial velocity distributions of different hub-angle impellers at the design flow rate of $8 \text{ m}^3/\text{s}$ along blade heights of 46%, 78%, and 92%. For the 90° hub angle, a pronounced low-velocity zone appears near the suction side and hub zone, indicating the accumulation of low-momentum air and weak flow development inside the passage. When the hub angle decreases to 87.5° and 85° , the low-velocity area is gradually reduced, and the mainstream flow becomes more continuous along the blade passage.

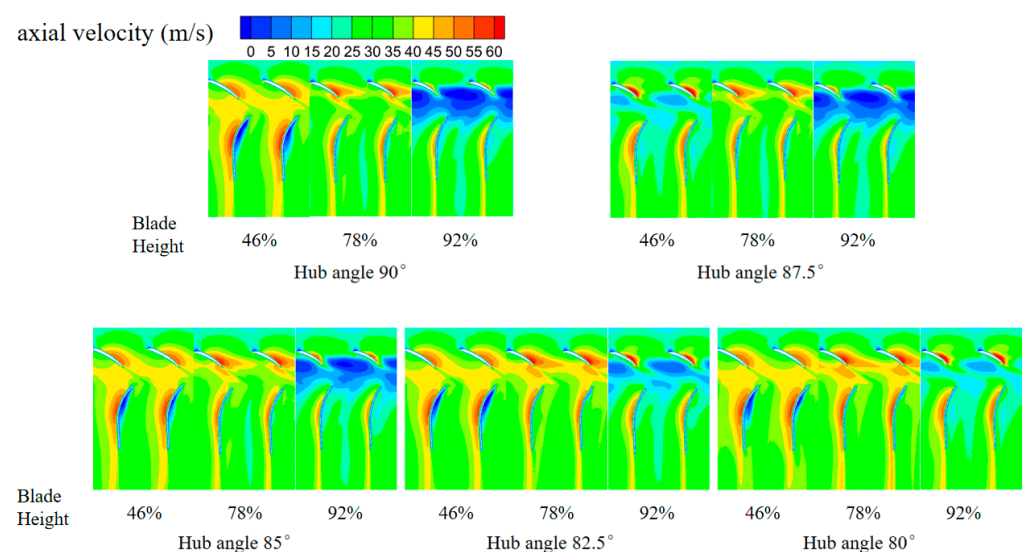


Figure 12. Axial velocity distributions of five impellers at the volume flow rate of $8 \text{ m}^3/\text{s}$.

At 82.5° , the axial velocity field becomes noticeably more uniform across all blade heights, and the low-kinetic-energy zones are further suppressed. When the hub angle is reduced to 80° , the axial velocity distribution becomes the most uniform, with the smallest low-velocity zone and the strongest continuity of the mainstream flow throughout the passage. This progressive improvement in axial velocity uniformity as the hub angle decreases is consistent with the efficiency trend.

3.1.3. Flow Characteristics

The internal flow field of an axial fan is highly complex, and variations in hub angle can significantly alter the generation and spatial distribution of a vortex, particularly the tip-leakage vortex. To examine how the tip-leakage vortex responds to changes in hub angle, the Q -criterion (Q_c) is employed to visualize the vortex structures within the impeller zone. The analysis focuses on the 92% blade span, where tip-leakage effects are the most pronounced.

The Q -criterion (Q_c) is a vortex-identification approach constructed from the velocity gradient tensor, enabling the description of a vortex's magnitude, spatial position, and evolution within a flow field. Its mathematical definition is given as follows [26,27]:

$$Q_c = \frac{1}{2} \left(\|\boldsymbol{\Omega}\|^2 - \|\mathbf{S}\|^2 \right) \quad (13)$$

with

$$\mathbf{\Omega} = \frac{1}{2} \left[\nabla \vec{v} - (\nabla \vec{v})^T \right] \quad (14)$$

with

$$\mathbf{S} = \frac{1}{2} \left[\nabla \vec{v} + (\nabla \vec{v})^T \right] \quad (15)$$

where $\mathbf{\Omega}$ is vorticity tensor and $\mathbf{S}$ is rate-of-strain tensor. Zones in which the second invariant of the velocity gradient tensor takes positive values (that is, $Q_c > 0$) are generally regarded as vortices. In these zones, fluid rotation dominates over the local strain rate. Varying thresholds of Q_c can therefore be used to distinguish vortical structures of different strengths within the flow field.

As shown in Figure 13, the Q -criterion (Q_c) contour ($Q_c = 4 \times 10^5 \text{ s}^{-2}$) at 92% blade height clearly illustrates the distribution and evolution of the tip-leakage vortex. The red zones represent the visualized coherent vortex structures. For hub angles greater than 82.5° , the tip-leakage vortex shows a relatively scattered and irregular spatial distribution. After the leakage flow escapes through the tip clearance of the upstream blade, the resulting vortex convects in both axial and circumferential directions and subsequently splits into multiple vortex branches. Among them, the strongest and longest branch develops near the suction surface. Upon interacting with the downstream blade, this branch impinges on the leading-edge zone, is cut off by the blade, and then convects toward the pressure surface and further downstream. This strong vortex blade interaction intensifies local pressure fluctuations and reduces the blade's ability to perform work on the airflow, thereby lowering the local working efficiency of the blade section, which is consistent with the previously observed performance trends.

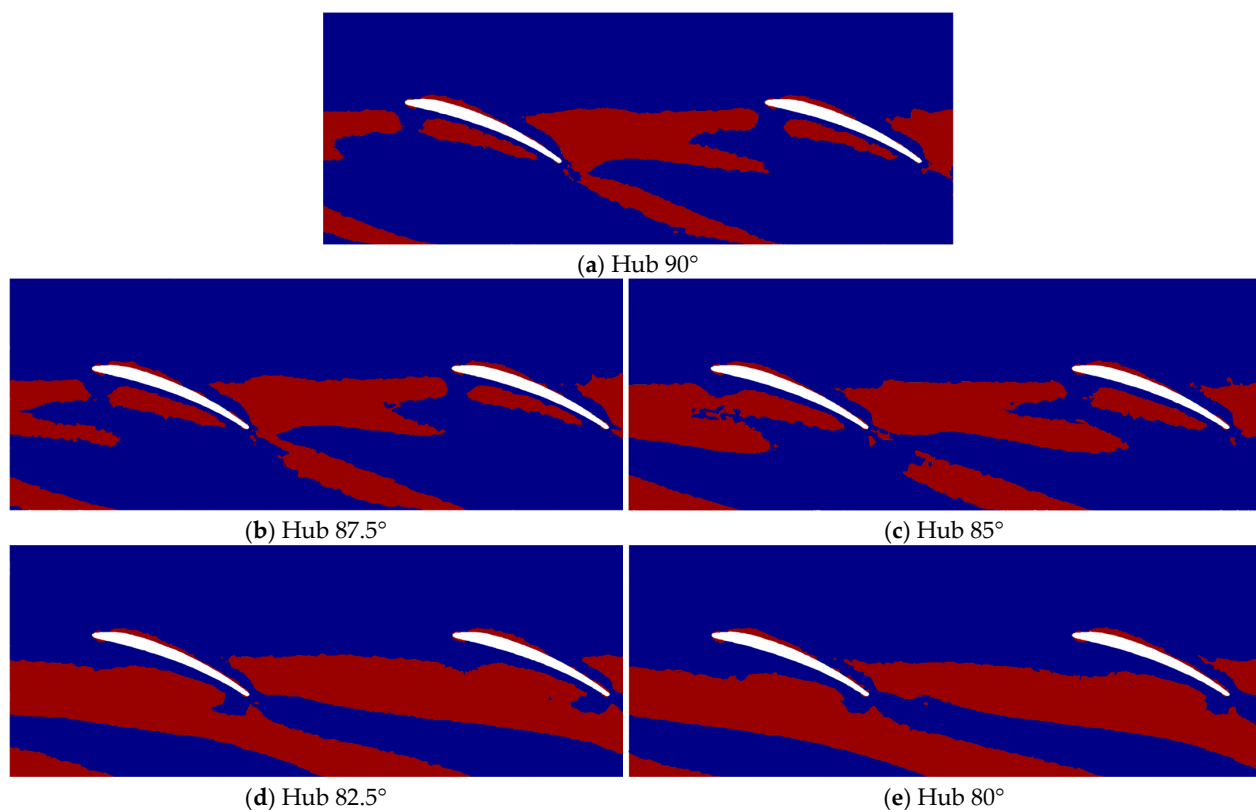


Figure 13. Q -criterion (Q_c) vortex structures at 92% blade height ($Q_c = 4 \times 10^5 \text{ s}^{-2}$).

More importantly, as the hub angle decreases, the axial velocity within the flow passage increases, causing the spatial location of the tip-leakage vortex to shift progressively downstream. As shown in Figure 13, at a hub angle of 82.5° , the formerly complex,

multi-branch leakage vortex structure collapses into a relatively continuous vortex, whose impingement position on the downstream blade is displaced toward the trailing edge zone. With a further reduction in the hub angle to 80° , this downstream migration becomes more pronounced, and the leakage vortex is observed to convect closer to the blade trailing edge zone and the fan outlet.

A similar dependence of the chordwise position of the tip-leakage vortex on flow conditions was reported by Adrián et al. [28], who investigated the rotor–stator interaction in a low-speed axial fan and observed that the tip vortex rolled up at different chordwise locations within the flow passage. They found that, as the volumetric flow rate decreased, the leakage vortex exhibited a clear tendency to migrate toward the blade’s leading edge. Although the present study varies the hub angle rather than the volumetric flow rate, both results consistently indicate that changes in the local flow momentum strongly govern the chordwise development and impingement location of the tip-leakage vortex.

To better investigate the internal flow characteristics of the axial flow fan, a turbulent kinetic energy analysis is conducted. The analysis sections start from the horizontal plane passing through the impeller center, and additional sections are defined at intervals of 70 mm, resulting in a total of six sections, as shown in Figure 14.

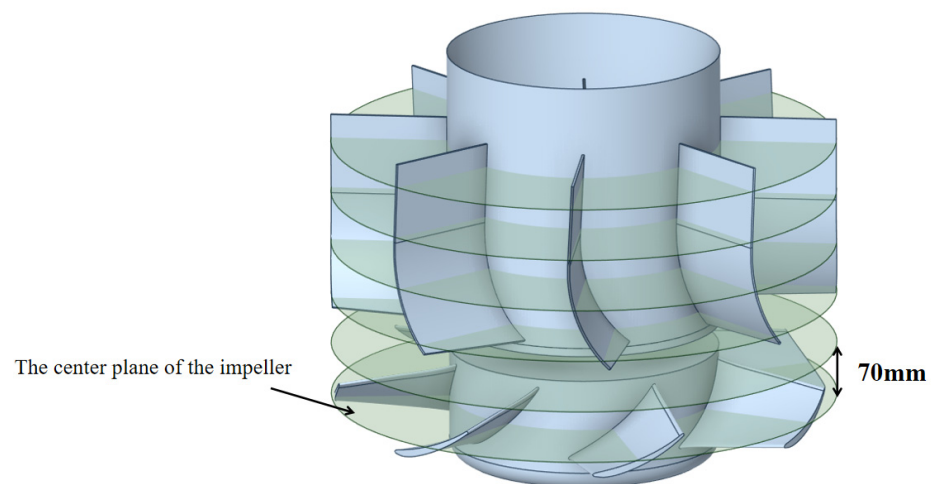


Figure 14. Schematic of the cross-section for turbulent kinetic energy analysis.

As shown in Figure 15, the turbulent kinetic energy in the 90° impeller passage is relatively high at $7 \text{ m}^3/\text{s}$ and $7.5 \text{ m}^3/\text{s}$, with large high- k zones distributed near the blade surfaces and hub area, indicating strong vortex activity under low-flow conditions.

When the flow rate increases to $8 \text{ m}^3/\text{s}$, the turbulent kinetic energy in the vane zone decreases noticeably, and the high- k zones begin to shrink. As the flow rate further rises to $8.5 \text{ m}^3/\text{s}$ and $8.7 \text{ m}^3/\text{s}$, the turbulent kinetic energy within the impeller passage is further reduced, and the high- k zones are largely suppressed. This trend is consistent with the previous analysis results, showing that the internal flow becomes more stable as the fan approaches the optimal operating condition. At $9 \text{ m}^3/\text{s}$, the turbulent kinetic energy in the impeller zone increases slightly again, indicating a mild deterioration of flow stability under higher flow rate conditions.

As shown in Figure 16, the turbulent kinetic energy distributions of the five impellers at the flow rate of $8 \text{ m}^3/\text{s}$ exhibit a clear variation with hub angle. For the 90° , 87.5° , and 85° impellers, relatively strong turbulent kinetic energy is observed within the impeller passage, especially near the blade surfaces and hub zone, indicating intense vortex activity and stronger flow disturbances.

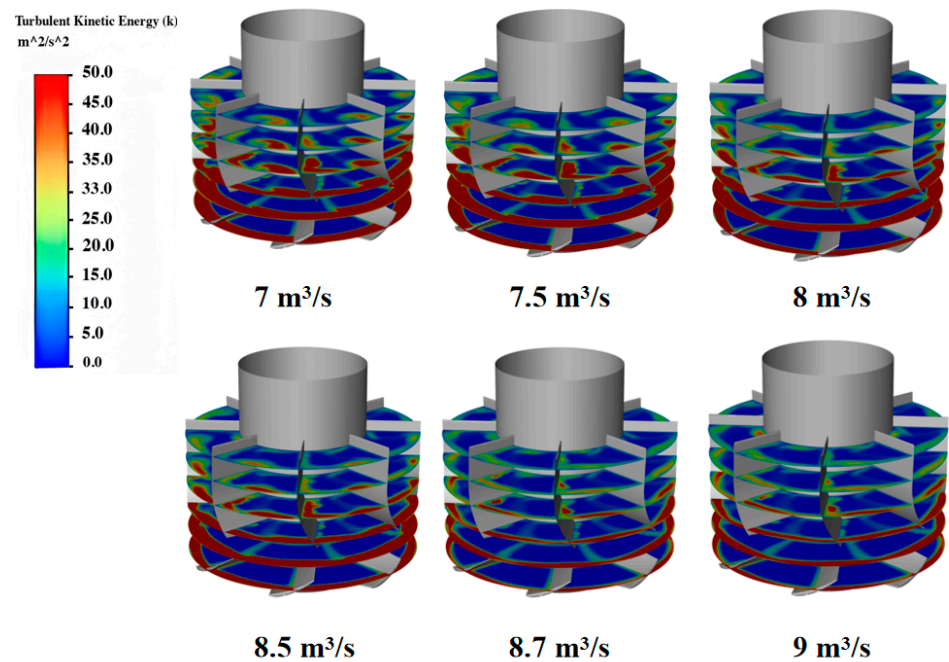


Figure 15. Turbulent kinetic energy contours at different volume flow rates for the 90° impeller.

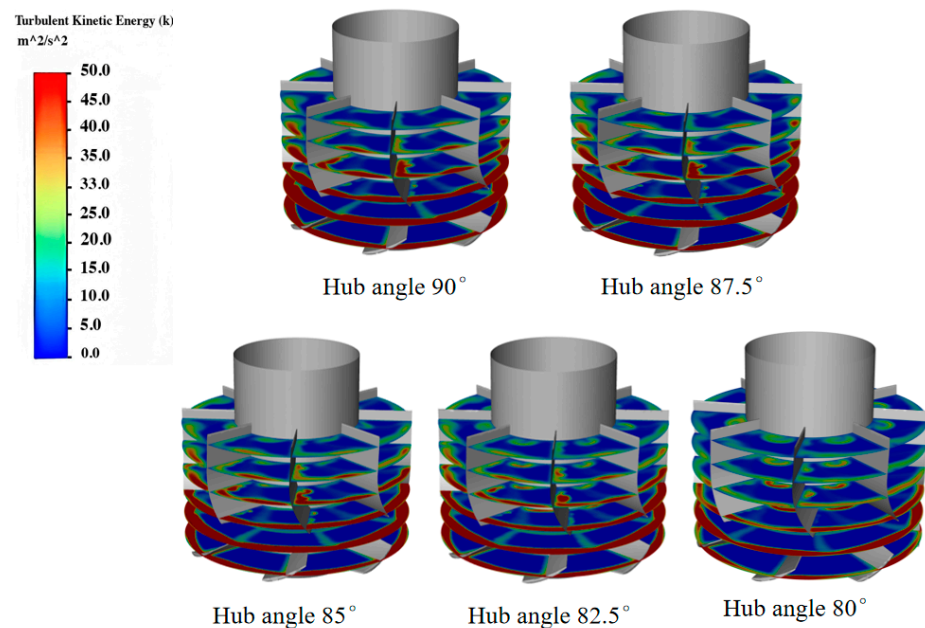


Figure 16. Turbulent kinetic energy distributions of five impellers at the volume flow rate of 8 m³/s.

When the hub angle decreases to 82.5°, the turbulent kinetic energy in the impeller zone begins to weaken noticeably. This reduction becomes most evident at 80°, where the overall k level inside the impeller passage is the lowest among all cases, suggesting a more stable internal flow structure. This observation is consistent with the previous Q -criterion vortex analysis results.

An interesting phenomenon is observed for the 82.5° and 80° impellers: more regular vortex structures appear in the guide vane zone. These organized vortical patterns indicate that the flow leaving the impeller becomes more ordered as the hub angle decreases, which is beneficial for downstream flow stability.

3.2. Sound Characteristics

3.2.1. Acoustic Modeling

To evaluate the aerodynamic noise characteristics of the axial fan under the design operating condition, an acoustic model is established based on the unsteady flow field solution. The aerodynamic noise prediction follows the hybrid computational aeroacoustics (CAA) framework, in which the flow field is first resolved using an unsteady CFD solver, and the resulting pressure and velocity fluctuations are subsequently employed as acoustic source terms. This approach effectively decouples the aerodynamic and acoustic scales, ensuring accurate capture of both broadband turbulence-induced noise and rotational tonal components associated with blade passing events.

The reliability of the hybrid LES + FW–H strategy for axial fan aeroacoustics prediction has been demonstrated in previous studies where numerical results were systematically compared with experimental measurements. For instance, Zhao et al. [29] applied LES combined with the FW–H acoustic analogy to an axial fan and reported good agreement between predicted and measured sound pressure spectra, with discrepancies generally within 2–4 dB. Such prior validations indicate that the LES + FW–H framework is capable of providing quantitatively reliable aeroacoustics predictions for axial fan configurations.

In the present study, the unsteady flow field is resolved using the LES turbulence model, and the impeller rotation is realized through the moving mesh (sliding mesh) approach to accurately reproduce the transient rotor motion. The time step is set to 0.00004 s, and a total of 5000-time steps are computed, corresponding to approximately 10 impeller revolutions at a rotational speed of 2970 rpm. During the transient calculation, the acoustic signal is recorded every 50-time steps for subsequent analysis. Accordingly, the time step determines the acoustic sampling frequency (25,000 Hz), yielding a maximum resolvable frequency of approximately 12,500 Hz according to the Nyquist criteria.

In this study, the acoustic sources are extracted from the computed unsteady flow field and projected into the acoustic domain using the Ffowcs Williams–Hawkings (FW–H) analogy. The FW–H control surface is explicitly defined on the impeller wall surfaces, corresponding to the impermeable (solid surface) FW–H formulation implemented in Fluent. These wall surfaces are treated as acoustic source surfaces to account for both thickness noise and loading noise generated by the rotating blades.

The far-field acoustic pressure is obtained by integrating the FW–H formulation in the time domain at the predefined virtual microphone positions. The recorded acoustic signals are finally processed using Fast Fourier Transform (FFT) to obtain the noise frequency spectra. The theoretical background and numerical implementation of this impermeable FW–H approach are consistent with the methodology reported by Mo and Choi [30]. This modeling strategy allows reliable prediction of the spectral characteristics of the radiated noise and provides a basis for analyzing how variations in hub angle influence the tonal and broadband noise components of the axial fan.

In addition, a total of 16 virtual microphones is arranged to capture the radiated acoustic field, as illustrated in Figure 17. The microphones are uniformly distributed on both the horizontal plane and the meridional plane. On the horizontal plane, each microphone is positioned 1 m away from the impeller center, while on the meridional plane, each is located 1 m away from the fan center. The sound pressure level (SPL) L_p is defined as follows:

$$L_p = 20 \times \lg \frac{p}{p_0} \quad (16)$$

where p is the local instantaneous sound pressure, and p_0 is the reference sound pressure of 2×10^{-5} Pa.

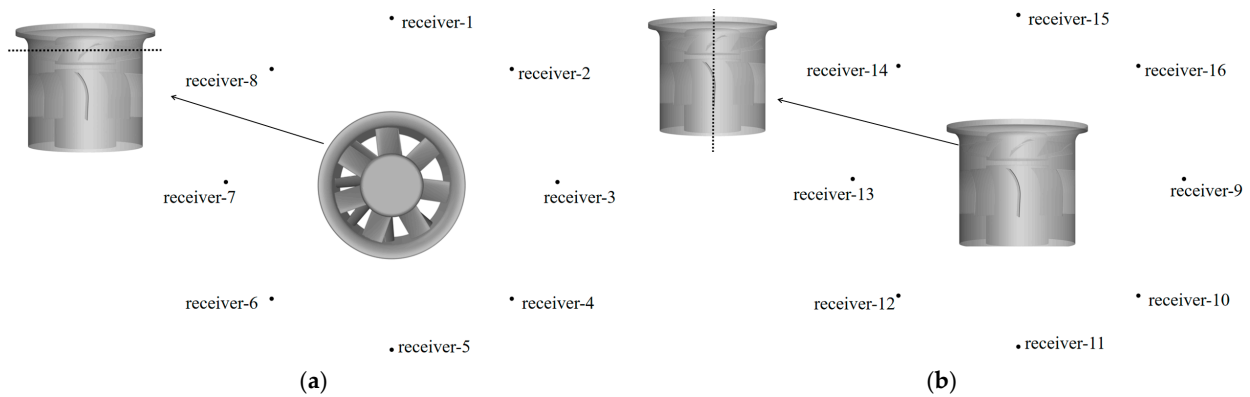


Figure 17. Schematic of the monitoring points. (a) Spatial distribution of virtual microphones on the horizontal plane for far-field noise evaluation; (b) spatial distribution of virtual microphones on the meridional plane for far-field noise evaluation.

3.2.2. Noise Directivity Analysis

To distinguish the noise received by each microphone more clearly, a noise directivity analysis is performed. Furthermore, the blade-passing frequency (BPF) noise level collected by each microphone, that is, the noise level at the BPF, is shown in Figure 18.

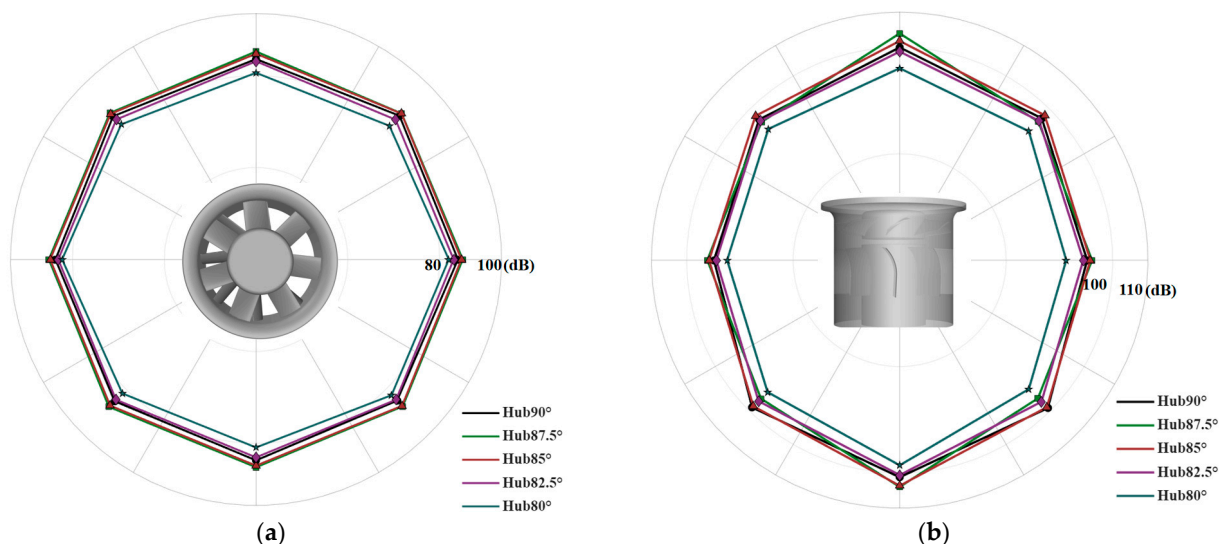


Figure 18. Acoustic directivity comparison at various hub angles. (a) Description of what is contained in the first panel; (b) description of what is contained in the second panel.

The BPF can be expressed as:

$$f = \frac{Zn}{60} \quad (17)$$

where Z is the number of rotating blades, and n is the rotating speed of the impeller.

Figure 18 illustrates the distribution of the fundamental noise level at different monitoring locations. On the horizontal plane of the fan, for all hub angle configurations, the variation in noise level among different azimuthal positions is minimal. Specifically, the differences between the maximum and minimum detected noise levels are only 1.5 dB for the 90° configuration, 0.9 dB for 87.5°, 0.25 dB for 85°, 0.3 dB for 82.5°, and 3.3 dB for 80°. These small variations indicate that noise propagation on the horizontal plane is relatively uniform.

However, for the 80° and 82.5° configurations, the fundamental noise levels measured at all azimuthal positions are noticeably reduced, with the reduction being more

pronounced for the 80° case. Compared with the baseline 90° impeller, the noise levels at monitoring points 1–8 for the 80° configuration decreased by 5.0 dB, 6.2 dB, 4.0 dB, 3.4 dB, 5.2 dB, 3.8 dB, and 5.0 dB, respectively. This reduction can be attributed to the decrease in hub angle, which leads to an increase in axial flow velocity that consequently alters the position and structure of the tip-leakage vortex. Under these hub angle conditions, the leakage vortex no longer impinges on the adjacent blade, thereby weakening blade–vortex interaction and reducing interference noise. This interpretation is consistent with the previously observed modifications in the structure and trajectory of the tip-leakage vortex.

In contrast, the noise distribution in the meridional plane exhibits a distinctly different pattern. At a radial distance of 1 m from the fan center, the noise levels measured at both the inlet and outlet are significantly higher than those recorded at other monitoring points. The noise decreases progressively as the observation point moves away from the inlet and outlet zone, reaching its minimum on the horizontal plane. The highest noise level is observed at the outlet for the 87.5° model, where it reaches 107.7 dB. An interesting observation is that, for the 87.5° configuration, the noise levels at monitoring positions 10 and 12 are lower than those of the other models, yet still higher than the corresponding values for the 80° configuration. This feature suggests that the 87.5° configuration produces the strongest directional noise projection in the outlet zone, whereas the 80° configuration benefits from reduced vortex blade interference.

3.2.3. The Noise Spectrum

The radiated noise characteristics of the axial fan are quantified by analyzing the sound pressure spectra obtained from the virtual microphone array. Based on the time-domain acoustic pressure signals predicted using the FW–H acoustic analogy, the frequency spectra are computed through Fast Fourier Transform (FFT). This procedure enables simultaneous identification of both tonal and broadband components, which are essential for evaluating the acoustic performance of the fan.

The dominant tonal peaks correspond to the blade-passing frequency (BPF) and its higher-order harmonics, reflecting the periodic aerodynamic loading induced by rotor–stator interactions. Meanwhile, the broadband components primarily originate from turbulent eddies in the wake zone and flow separation near the blade tip. The spectral evolution at different monitoring positions reveals the influence of hub angle variations on both tonal and broadband noise components.

Since a large number of monitoring positions are deployed, and the previous section has shown that the noise differences on the horizontal plane are relatively small, only representative spectra are selected for comparison. Specifically, spectra from Receiver-2, 4, 6, and 8 located on the horizontal plane, as well as Receiver-11 and 15 located on the meridional plane, are chosen to characterize the spectral variations among the five hub angle configurations.

Figure 19 presents the full-band noise spectra measured at all monitoring points under different hub angles, illustrating the overall spectral characteristics for each configuration. Across all configurations, the blade-passing fundamental frequency remains constant at every monitoring point, indicating that variations in the hub angle do not modify the fundamental tonal characteristic of the axial fan. Using the baseline configuration in Figure 19a as an example, the spectral evolution with increasing frequency shows a clear trend: the sound pressure level first rises and reaches a maximum of 88.1 dB at the fundamental frequency, after which it gradually decreases as the frequency continues to increase. When the frequency exceeds approximately 4000 Hz, the noise level reduces to around 20 dB and eventually stabilizes.

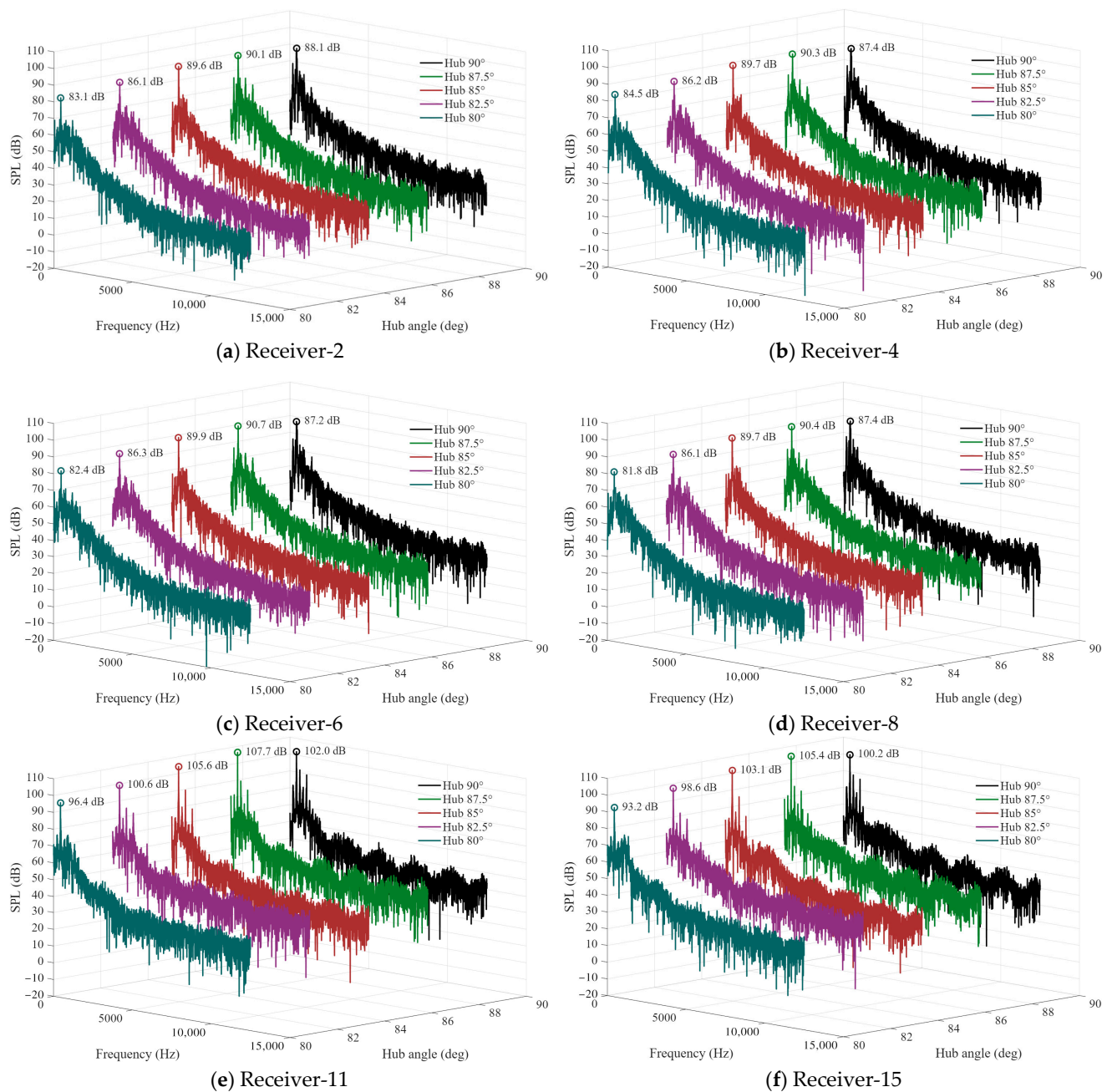


Figure 19. Distribution of SPL at each probe under different hub angles.

In the low-frequency range, no consistent monotonic trend is observed as the hub angle decreases. However, an important observation is that the sound pressure levels (SPL) at the fundamental frequency remain relatively close when the hub angle is 90°, 87.5°, and 85°. In Figure 19a, the maximum deviation among these configurations is only 2 dB, while the deviations in Figure 19b–d are 2.9 dB, 3.5 dB, and 3 dB, respectively. Notably, when the hub angle is reduced to 82.5° and 80°, the fundamental tone noise decreases significantly, with average reductions of approximately 2 dB and 4 dB compared with the 90° configuration. This reduction is likely related to changes in the shape and strength of the tip-leakage vortex and its interactions with adjacent blades, which is consistent with the previously observed aerodynamic behavior. In contrast, the variations in SPL at the fundamental frequency are substantially larger in Figure 19e. The noise level increases from 102 dB at 90° to a peak of 107.7 dB at 87.5°, after which it gradually declines as the hub angle decreases further. A pronounced drop occurs again at 82.5°, and by the time the hub

angle reaches 80° , the SPL at the fundamental tone has decreased by 5.6 dB compared with the baseline configuration. A similar trend is observed in Figure 19f, although the reduction at 80° is even stronger, reaching 6.8 dB. These results indicate that the inlet and outlet zones of the fan exhibit greater sensitivity to changes in the hub angle. In other words, variations in the hub angle induce more substantial modifications to the fundamental-frequency noise at the fan inlet and outlet than on the horizontal plane. Another interesting observation in Figure 19e,f is the behavior of the blade-passing frequency (BPF) harmonics. For the first four hub angle configurations, the first three BPF harmonics are clearly distinguishable. However, when the hub angle reaches 80° , only the fundamental tone remains dominant, and the second and third BPF harmonics become indistinct. This phenomenon suggests that the modified tip-leakage vortex dynamics at small hub angles weaken the periodic pressure fluctuations that typically generate higher-order tonal components.

Figure 20 presents the noise spectra for frequencies above 4000 Hz at all monitoring points under different hub angles. For the high-frequency noise components above 4000 Hz, the spectral evolution at the monitoring points on the fan's horizontal plane, shown in Figure 20a–d, demonstrates a generally decreasing trend as the hub angle decreases. From 90° to 87.5° , the reduction is modest, with an average drop of only approximately 2 dB. A more pronounced decline begins at 85° . Taking Receiver-2 in Figure 20a as an illustrative example, the average high-frequency SPL decreases by 3.2 dB when the hub angle changes from 87.5° to 85° , followed by additional reductions of 2.5 dB and 2.2 dB when the hub angle is further decreased to 82.5° and 80° , respectively. These results indicate that the high-frequency noise is increasingly suppressed once the hub angle falls below 87.5° , likely due to changes in tip-leakage vortex dissipation and reduced broadband turbulence generation.

A similar overall decreasing pattern is observed at the inlet and outlet monitoring points on the fan's meridional plane. However, an important distinction emerges: the SPL reduction between 87.5° and 85° is substantially larger than that on the horizontal plane. Specifically, the average high-frequency noise drops by 4.2 dB in Figure 20e and by 3.8 dB in Figure 20f. Interestingly, at a hub angle of 85° , the high-frequency noise level becomes even lower than that at 87.5° . The average reduction is 0.8 dB for the inlet location in Figure 20e and 1.3 dB for the outlet location in Figure 20f. This behavior suggests that modifications in the flow structures induced by hub angle variation, particularly the alteration of vortex stretching and turbulence intensity near the blade tip and casing, play a stronger role in shaping the high-frequency acoustic response on the meridional plane than on the horizontal plane.

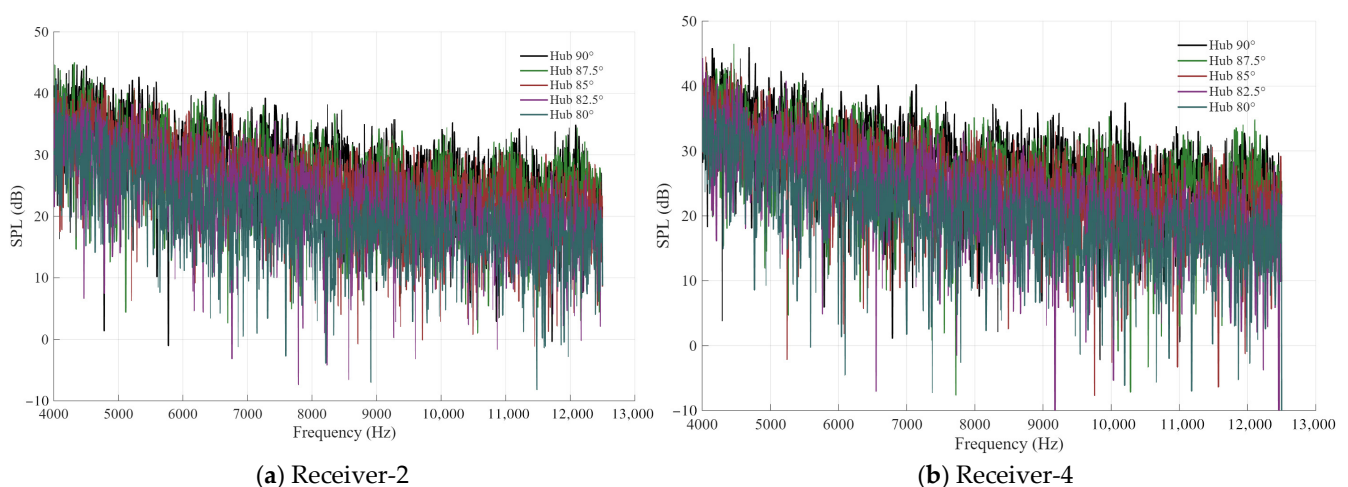


Figure 20. Cont.

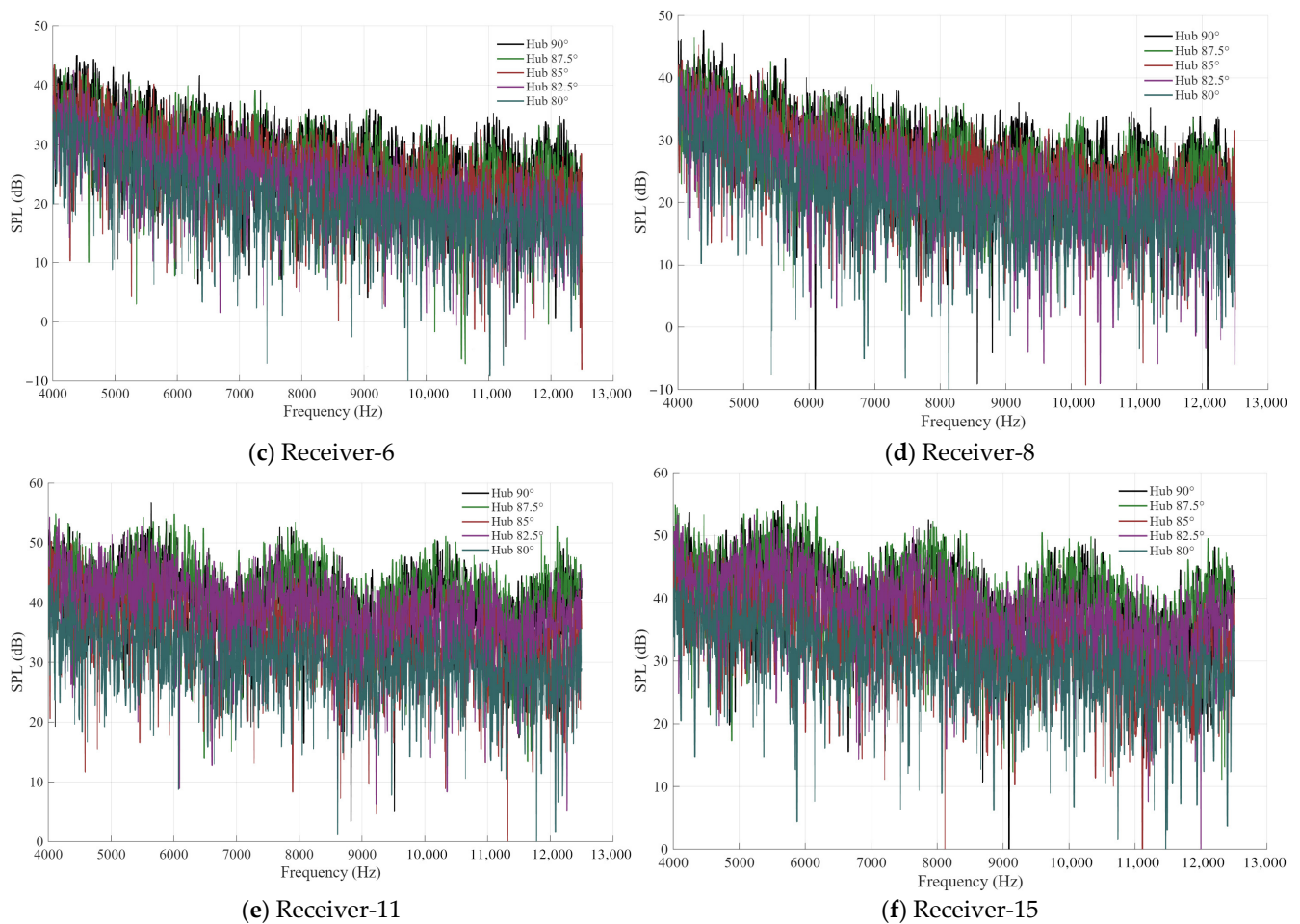


Figure 20. Distribution of SPL at each probe under different hub angles in the high-frequency range (Frequency > 4000 Hz).

4. Conclusions

- (1) The hub angle plays a key role in governing the internal flow development of the axial flow fan. A reduced hub angle modifies the flow passage geometry, leading to noticeable changes in pressure distribution and blade loading at different blade heights, with enhanced blade influence on the flow, particularly near the blade tip.
- (2) Hub angle variation has a significant influence on the aerodynamic performance characteristics of the fan. As the hub angle decreases, the static pressure and efficiency curves tend to shift toward lower volumetric flow rates, including the points of maximum static pressure and peak efficiency, indicating improved performance under low-flow conditions. An appropriately reduced hub angle enhances the matching between the impeller geometry and the design volumetric flow rate point, whereas excessive reduction may limit performance at higher volumetric flow rates.
- (3) The tip-leakage vortices are strongly affected by the hub angle. With decreasing hub angle, the leakage vortices migrate downstream and their spatial extent is reduced. The originally complex and multi-branch vortex structures evolve into more coherent and weakened vortices, which alleviate their interactions with downstream blades and enhance local flow stability near the blade tip.
- (4) The acoustic characteristics of the axial flow fan are closely linked to hub-angle-induced flow modifications. Smaller hub angles result in a clear reduction in tonal noise at the blade-passing frequency and suppression of higher-order harmonics, while high-frequency broadband noise is also reduced. These noise reductions are

primarily attributed to weakened blade–vortex interaction and reduced turbulence intensity near the blade tip. Overall, hub angle optimization offers a practical design strategy for achieving a balanced improvement in aerodynamic efficiency and noise performance of axial flow fans.

The present findings suggest that modifying the hub geometry represents a relatively simple and feasible approach for regulating axial fan performance compared with more complex structural redesigns such as blade reshaping. As an easily adjustable geometric parameter, hub angle variation offers a practical means of tuning aerodynamic and acoustic characteristics. These results may serve as a useful reference for performance optimization, fan selection, and ventilation system matching in engineering practices. Future experimental validation will be conducted to further confirm the applicability of this strategy.

Author Contributions: Conceptualization, W.Z. and Y.Y.; methodology, W.Z. and Y.Y.; software, W.Z. and R.T.; validation, W.Z. and Y.Y.; formal analysis, W.Z.; investigation, W.Z., R.T. and Y.Y.; resources, Y.Y. and Y.L.; data curation, W.Z. and Y.Y.; writing—original draft preparation, W.Z.; writing—review and editing, Y.Y.; visualization, W.Z.; supervision, Y.Y. and Y.L.; project administration, Y.Y. and Y.L.; funding acquisition, Y.Y. All authors have read and agreed to the published version of the manuscript.

Funding: This research project is supported by the National Natural Science Foundation of China (Grant Nos. 12102076, 51875073) and the Liaoning province Department of Education fund (Grant Nos. LJKQZ20222469).

Institutional Review Board Statement: Not applicable.

Informed Consent Statement: Not applicable.

Data Availability Statement: The datasets generated and analyzed during the current study are available from the corresponding author on reasonable request.

Acknowledgments: The authors would like to express their sincere gratitude to Xianzhen Su (Dalian Dingli Rail Transit Equipment Company Limited, Dalian 116033, China) for his valuable technical suggestions regarding the preprocessing of the original model and the mesh generation strategy during the early stage of this study.

Conflicts of Interest: The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

References

1. Lu, C.; Zhang, H. Analysis of Factors Influencing Riding Comfort of High-Speed Railway. *Chain* **2024**, *1*, 99–111. [\[CrossRef\]](#)
2. Ding, Y.; Wang, J.; Jiang, B.; Xiao, Q.; Yang, X.; Wu, L.; Xie, B. Numerical investigation of the effect of blade distortion laws on the corner flow separation of the axial-flow fan. *Aerosp. Sci. Technol.* **2023**, *138*, 108296. [\[CrossRef\]](#)
3. Ye, X.; Li, P.; Li, C.; Ding, X. Numerical investigation of blade tip grooving effect on performance and dynamics of an axial flow fan. *Energy* **2015**, *82*, 556–569. [\[CrossRef\]](#)
4. Ye, X.; Zheng, N.; Hu, J.; Li, C.; Xue, Z. Numerical investigation of the benefits of serrated Gurney flaps on an axial flow fan. *Energy* **2022**, *252*, 124072. [\[CrossRef\]](#)
5. Park, M.; Lee, D.J.; Lee, H. Tonal and broadband noise attenuation of an axial flow fan with forward and backward swept blades. *Appl. Acoust.* **2022**, *195*, 108837. [\[CrossRef\]](#)
6. Chen, S.; Wang, D.; Sun, S. Bionic fan optimization based on Taguchi method. *Eng. Appl. Comput. Fluid Mech.* **2011**, *5*, 302–314. [\[CrossRef\]](#)
7. Lee, K.S.; Kim, K.Y.; Samad, A. Design optimization of low-speed axial flow fan blade with three-dimensional RANS analysis. *J. Mech. Sci. Technol.* **2008**, *22*, 1864–1869. [\[CrossRef\]](#)
8. Meyer, T.O.; van der Spuy, S.J.; Meyer, C.J.; Corsini, A. Optimization of a tip appendage for the control of tip leakage vortices in axial flow fans. *J. Turbomach.* **2022**, *144*, 101006. [\[CrossRef\]](#)
9. Chunxi, L.I.; Xinying, L.I.; Xuemin, Y.E. Numerical Investigation of Blade Trimming Effect on Performance of Axial Flow Fan. *J. Mech. Eng.* **2014**, *50*, 183–190.

10. Shi, H.; Zhu, R.; Yu, Z.; Wang, J. Influence of hub cap shape on axial fan rotor root flow separation: An analytical model for quick design. *Phys. Fluids* **2025**, *37*, 064111. [[CrossRef](#)]
11. Jung, J.H.; Joo, W.G. The effect of the entrance hub geometry on the efficiency in an axial flow fan. *Int. J. Refrig.* **2019**, *101*, 90–97. [[CrossRef](#)]
12. Jung, J.H.; Joo, W.G. Effect of tip clearance, winglets, and shroud height on the tip leakage in axial flow fans. *Int. J. Refrig.* **2018**, *93*, 195–204. [[CrossRef](#)]
13. Wang, C. Noise source analysis for two identical small axial-flow fans in series under operating condition. *Appl. Acoust.* **2018**, *129*, 13–26. [[CrossRef](#)]
14. Chen, F.; Zhu, G.; Xi, D.; Miao, B. Air volume flow rate optimization of the guide vanes in an axial flow fan based on DOE and CFD. *Sci. Rep.* **2023**, *13*, 4439. [[CrossRef](#)] [[PubMed](#)]
15. Krasnyuk, A.M.; Kosykh, P.V. Aerodynamic Design of Axial Fan with Guide Vane for Reverse Air Flow. *J. Min. Sci.* **2023**, *59*, 424–432. [[CrossRef](#)]
16. Jin, W.; Mao, Z.; Zhou, S.; Zhang, T.; Hu, Y.; Wu, Z. Research on multi-optimal project of outlet guide vanes of nuclear grade axial flow fan based on sensitivity analysis. *Appl. Sci.* **2022**, *12*, 3029. [[CrossRef](#)]
17. Wang, J.; Krut, N.P. Design for high efficiency of low-pressure axial fans with small hub-to-tip diameter ratio by the vortex distribution method. *J. Fluids Eng.* **2022**, *144*, 081201. [[CrossRef](#)]
18. Angelis, W.; Lörcher, F. Fan Unit with Compact Guide Vane Designed for Low Hub Ratio Axial Fans. In Proceedings of the Fan 2018 Conference, Darmstadt, Germany, 18–20 April 2018; pp. 18–20.
19. Wang, J.; Krut, N.P. Computational fluid dynamics simulations of aerodynamic performance of low-pressure axial fans with small hub-to-tip diameter ratio. *J. Fluids Eng.* **2020**, *142*, 091202. [[CrossRef](#)]
20. Carolus, T.; Schneider, M.; Reese, H. Axial flow fan broad-band noise and prediction. *J. Sound Vib.* **2007**, *300*, 50–70. [[CrossRef](#)]
21. Reese, H.; Kato, C.; Carolus, T.H. Large eddy simulation of acoustical sources in a low pressure axial-flow fan encountering highly turbulent inflow. *J. Fluids Eng.* **2007**, *129*, 263–272. [[CrossRef](#)]
22. Du Plessis, C. Scaling of Axial Fan Noise. Doctoral Dissertation, Stellenbosch University, Stellenbosch, South Africa, 2022.
23. Ji, B.; Luo, X.; Peng, X.; Wu, Y. Three-dimensional large eddy simulation and vorticity analysis of unsteady cavitating flow around a twisted hydrofoil. *J. Hydrodyn.* **2013**, *25*, 510–519. [[CrossRef](#)]
24. Nicoud, F.; Ducros, F. Subgrid-scale stress modelling based on the square of the velocity gradient tensor. *Flow Turbul. Combust.* **1999**, *62*, 183–200. [[CrossRef](#)]
25. Li, C.X.; Zhang, C.; Zhang, R.X.; Ye, X.M. Effect of Gurney flap on performance and aeroacoustics of variable-pitch axial fans. *AIAA J.* **2020**, *58*, 2546–2559. [[CrossRef](#)]
26. Binjuan, Z.; Yuntong, X.I.E.; Wenyan, L.; Luyao, H.; Yanxia, F.; Zhongfu, H. Adaptability analysis of second generation vortex recognition method in internal flow field of mixed-flow pumps. *J. Mech. Eng.* **2020**, *56*, 216–223. [[CrossRef](#)]
27. Ghasemian, M.; Nejat, A. Aerodynamic noise prediction of a horizontal axis wind turbine using improved delayed detached eddy simulation and acoustic analogy. *Energy Convers. Manag.* **2015**, *99*, 210–220. [[CrossRef](#)]
28. Vázquez González, A.; Meana-Fernández, A.; Fernández, J.M. Stator-rotor interaction in the tip leakage flow of an inlet vaned low-speed axial fan. *Int. J. Numer. Methods Heat Fluid Flow* **2020**, *30*, 4425–4452. [[CrossRef](#)]
29. Zhao, W.; Wasala, S.; Persoons, T. Large eddy simulation for aeroacoustic prediction and characterization of electronics cooling axial fans under various operating conditions. *Appl. Acoust.* **2025**, *231*, 110530. [[CrossRef](#)]
30. Mo, J.; Choi, J. Numerical investigation of unsteady flow and aerodynamic noise characteristics of an automotive axial cooling fan. *Appl. Sci.* **2020**, *10*, 5432. [[CrossRef](#)]

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.