

Post-Print: Design of Multi-Blade Centrifugal Fan Based on Group Model and Bionic Volute Tongue

Authors: Wang, Ke, Ju Yaping, Zhang Chuhua

Date: 2017-10-30T00:00:00+00:00

Abstract

To improve the flow efficiency of multi-blade centrifugal fans, a group design model tailored to the characteristics of this fan type is proposed. Employing an analysis method that integrates flow field computation with experimental design, the impeller, volute, and volute tongue of the multi-blade centrifugal fan undergo sequential group design. The results demonstrate that: the aerodynamic performance of double-arc blades, which offer greater design freedom, is significantly superior to that of single-arc blades; while maintaining constant volute structural dimensions, the volute designed via the equilateral base method slightly outperforms that designed with three circular arcs; a bionic volute tongue based on the leading edge of the long-eared owl airfoil effectively attenuates flow separation phenomena; the design efficiency of the improved multi-blade centrifugal fan utilizing the proposed method increases by 4.33%, with significantly improved off-design performance. This research holds both academic significance and engineering application value for deepening the understanding of complex internal flow mechanisms in multi-blade centrifugal fans and advancing optimization design methodologies.

Full Text

1. Introduction

This study investigates the aerodynamic performance optimization of a multi-blade centrifugal fan through computational fluid dynamics (CFD) analysis. The research focuses on improving impeller efficiency by optimizing key geometric parameters, including blade angles and curvature, as well as enhancing volute flow characteristics.

[Figure 1: see original paper] Structure of original multi-blade centrifugal fan

The key structural parameters of original multi-blade centrifugal fan

2. Numerical Simulation Methods

The computational domain of the fan was established for CFD analysis, as shown in Figure 2. The numerical simulations employed the Multiple Reference Frame (MRF) approach to model the rotating impeller region. Grid generation was performed using ANSYS-ICEM for the entire domain and Turbo Grid for the impeller region, ensuring adequate resolution of the flow field.

[Figure 2: see original paper] Computational domain of fan

The Reynolds-averaged Navier-Stokes equations were solved using ANSYS-CFX with appropriate turbulence modeling. Boundary conditions were specified to match the experimental setup, with a design flow rate of $Q_v = 8537 \text{ m}^3/\text{h}$ and rotational speed of $n = 1050 \text{ rpm}$. The numerical results were validated against experimental data, as presented in Table 2.

Comparisons of experimental and CFD results at the design flow rate

3. Optimization of Impeller Parameters

3.1 Blade Angle and Curvature Optimization

The impeller efficiency is significantly influenced by the blade inlet angle (β_1) and outlet angle (β_2). Numerical experiments were conducted to evaluate the performance impact of varying these parameters. The optimization process revealed that a blade inlet angle of $\beta_1 = 90^\circ$ and outlet angle of $\beta_2 = 120^\circ$ yields substantial efficiency improvements.

[Figure 3: see original paper] Geometric structure of double-arc blade

Further analysis examined the effect of the impeller radius parameter R_f on overall performance. The results demonstrate that efficiency increases with R_f up to an optimal point. As shown in Figure 4, the maximum efficiency of 73.28% is achieved at $R_f = 180 \text{ mm}$, representing a 1.87% improvement over the baseline design. The corresponding pressure coefficient increases by 2.33%.

[Figure 4: see original paper] Influence of β_2 and R_f on impeller efficiency

The flow field analysis reveals that optimized blade geometry reduces flow separation and improves pressure distribution. Figure 5 illustrates the relative flow streamlines within the impeller at the mid-span plane ($z = 81 \text{ mm}$), showing smoother flow patterns for the optimized configuration.

[Figure 5: see original paper] Relative flow streamlines in impeller at $z=81 \text{ mm}$

The optimization process employed a response surface methodology based on CFD simulations. The mathematical relationship between the design parameters and efficiency was approximated by the following correlation:

§ “(cid:216)” (cid:217) “(cid:218)” §

where β_2 represents the blade outlet angle and R_f denotes the impeller radius parameter. This surrogate model enabled efficient exploration of the design space.

3.2 Volute Optimization

The volute geometry was subsequently optimized to complement the improved impeller design. The cross-sectional area distribution and tongue geometry were modified to reduce circumferential pressure distortion and minimize losses. Figure 6 presents the absolute flow streamlines in the volute at the mid-span plane, demonstrating more uniform flow distribution in the optimized configuration.

[Figure 6: see original paper] Absolute flow streamlines in volute at $z=81$ mm

The combined optimization of impeller and volute components resulted in a overall efficiency improvement of 92.83% for the NEW IMP configuration, with $\beta_1 = 81.72^\circ$, $\beta_2 = 95.17^\circ$, and $R_f = 187.76$ mm representing the optimal parameter set.

References

- [7] Adachi. Optimization methodology for centrifugal fan impellers [J]. Journal of Engineering Thermophysics, 2017, 38(8): 598-605.
- [8] LIAO Genghua, YANG Ying, HU Qinchao, et al. Three-dimensional Modeling of Long-eared Wing [J]. Journal of Jilin University: Engineering and Technology Edition, 2012, 42(S1): 423-427.
- [9] JU Yaping, ZHANG Chuhua. Optimal Design Method for Wind Turbine Airfoil Based on Artificial Neural Network Model and Genetic Algorithm [J]. Proceedings of the CSEE, 2009, 29(20): 106-111.
- [10] Klän S, Bachmann T, Klaas M, et al. Experimental Analysis of the Flow Field over a Novel Owl Based Airfoil [J]. Experiments in Fluids, 2009, 46(5): 975-989.
- [11] Additional references on CFD methodology and turbomachinery optimization.

Note: Figure translations are in progress. See original paper for figures.

Source: ChinaXiv – Machine translation. Verify with original.