STUDY ON THE INFLUENCE OF VOLUTE WIDTH-DIAMETER RATIO ON THE PERFORMANCE OF MULTI-BLADE CLEANING FANS

/

蜗壳宽径比对多翼清选风机性能的影响研究

Yichen DONG[1,2,3], Zheng MA[1,2,3,*], Jie YU[1,2,3], Shuai WANG[1,2,3], Hongyan GAO[1,2,3], Yaoming LI[1,2,3]

[1] School of Agricultural Engineering, Jiangsu University, Zhenjiang 212013, Jiangsu, China

[2] Key Laboratory for Theory and Technology of Intelligent Agricultural Machinery and Equipment, Jiangsu University, Zhenjiang 212013, Jiangsu, China

[3] Faculty of Agricultural Engineering, Jiangsu University, Zhenjiang 212013, Jiangsu, China

Tel: 18796087291; E-mail: mazheng123@ujs.edu.cn

DOI: https://doi.org/10.35633/inmateh-76-55

***Keywords:** combine harvester, multi-blade centrifugal fan, volute width-diameter ratio, flow field characteristics, CFD*

ABSTRACT

Multi-blade centrifugal fans are regarded as an important direction for agricultural cleaning fans due to the advantages of compact structure, low noise and high efficiency. In view of the insufficient aerodynamic performance of traditional centrifugal fans in combine harvester cleaning devices caused by straight-blade structures, as well as the lack of design theory for multi-blade centrifugal fans, this study focused on a multi-blade centrifugal fan with wing-shaped blades. Using a combination of CFD numerical simulation and bench tests, the influence of the volute width-to-diameter ratio on fan performance and the internal flow field was investigated. The results showed that at a rated speed of 1000 r/min, the optimal volute width-to-diameter ratio was 1.3. Compared with the prototype fan, the optimized fan achieved a 7.13% increase in efficiency, a 6.53% increase in air volume, and a 16.63% improvement in air distribution uniformity. In addition, the internal flow of the optimized fan was improved, with reduced turbulence intensity in the tongue region of the volute. Furthermore, a volute width-to-diameter ratio–speed–air volume function model was established using MATLAB, providing a theoretical basis for the selection and design of high-performance multi-blade centrifugal fans.

摘要

多翼离心风机因结构紧凑、低噪高效等优势，被视为农业清选风机的重要方向。针对联合收获机清选装置中传统离心风机因平直叶片结构导致气动性能不足，以及多翼离心风机设计理论缺乏的问题，本研究以翼型叶片的多翼离心风机为研究对象，采用 CFD 数值模拟与台架试验相结合，探究了蜗壳宽径比对风机性能和内部流场的影响。结果表明：在额定转速 1000 r/min 下，最优风机蜗壳宽径比为 1.3；相较于原型风机，效率提升 7.13%，风量提升 6.53%，出风均匀性提升 16.63%；优化后的风机内部流动改善，蜗舌区域湍流强度减小。基于 MATLAB 建立蜗壳宽径比-转速-风量函数模型，本研究可为高性能多翼离心风机设计提供选型依据。

INTRODUCTION

With the increase of combine harvester feeding capacity and the popularization of multi-purpose mode, the agricultural production of combine harvester cleaning performance has put forward more stringent requirements. As the combine harvester's “digestive system”, the performance of the cleaning device directly determines the seed impurity rate and loss rate and other key indicators *(Liang et al., 2020; Almoslh et al., 2021)*. Currently, on the market, about 93% of combine harvester cleaning device use wind sieve type cleaning systems *(Ma et al., 2024; Wei et al., 2020)*. These systems utilize the synergistic effect of the centrifugal fan volute flow channel and impeller to generate an airflow field that meets the cleaning requirements. In close coordination with the sieve plate, they enable the precise and effective separation of grain from glumes, stalks, and other impurities *(Badretdinov et al., 2019; Zhang et al, 2025)*. The efficiency of the fan directly influences gas flow, pressure, and airflow stability. A highly efficient fan can deliver a more stable airflow, thereby enhancing cleaning performance. With the continuous growth of crop production and the increasing cutting width of combine harvesters, there is an urgent need to improve the efficiency of existing centrifugal fans to better meet the demands of modern, high-efficiency agricultural production.

Yichen Dong, Master's degree student; Zheng Ma, Professor; Jie Yu, Assistant Researcher; Shuai Wang, Assistant Researcher, Hongyan Gao, Assistant Researcher; Yaoming Li, Professor

In recent years, many scholars have utilized CFD numerical simulation methods to thoroughly study the influence of centrifugal fan structure on fan performance. *Varun et al, (2023),* designed a centrifugal fan model with 11 backward curved blades. The study found that when the number of blades increased to 14, the total pressure increased by 21.77% and the efficiency increased by 5.74%. In order to reduce the impact of uneven airflow distribution on the cleaning performance, *Chai et al, (2020),* optimized the blade parameters of the fan of the combine harvester using the response surface method, revealing the mechanism of the airflow distribution at the air outlet.

Wang et al, (2017), carried out numerical simulations of four agricultural fan models and discussed the effects of fan blade number and inclination angle on fan characteristics. *Liang and Wada, (2023),* studied the influence of the fan structure in the grain combine harvester on the airflow characteristics, explored the impact of the cleaning load on the flow rate of each air outlet of the fan, and optimized the fan structure.

Multi-blade centrifugal fan is regarded as an important development direction of future cleaning fan because of its advantages of large number of blades, low noise, large air volume and high pressure coefficient *(Yang et al., 2024; Ottersten et al., 2022; Acarer et al., 2020)*. At present, research on the structure of multi - blade centrifugal fans mainly focus on optimizing their impeller structure, while studies on the influence of volute structure on cleaning performance are relatively scarce. The geometric shape, dimensional parameters of the volute, and its matching relationship with the impeller directly affect the air flow distribution, pressure field, and velocity field, thereby determining the fan's efficiency and air flow uniformity (*Schäfer et al., 2020; Shim et al, 2020, Babubhai et al,2023)*. How to improve the fan's ability for efficient air flow separation and guidance through volute structure optimization is a technical problem urgently needing to be solved in the current fan design field.

This paper focuses on the optimization of the volute structure of multi-blade centrifugal fans, with the ratio of the volute width to the outer diameter of the impeller as the research variable. By setting four groups of values of the volute width and dimensionless processing of this parameter, the universality of the conclusions is enhanced, and the conversion according to the law of fan similarity can be applied to the prediction of the performance of fans with different specifications. CFD numerical simulation and bench test are combined to investigate the influence of the volute width-diameter ratio on the performance of the multi-blade centrifugal fan, focusing on the analysis of the parameter on the uniformity of the outlet airflow, air volume and the internal flow field of the mechanism, and design and build a performance test rig to verify the simulation results, with the aim of providing a basis for the subsequent multi-blade centrifugal fan design and optimization of the selection of the basis.

MATERIALS AND METHODS

Multi-blade centrifugal fan model construction

This paper takes the multi-blade centrifugal fan as the research object, and the structural model is shown in Fig. 1. The impeller consists of 16 wing fan-shaped blades, and the two sets of blades are back-to-back symmetrically mounted on the two sides of the center disk; the unequal basis element method is used to determine the size of the volute casing profile of the multi-blade centrifugal fan. The main structural dimensions of the fan are shown in Table 1.

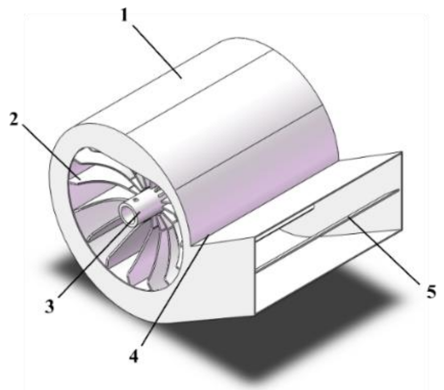


Fig. 1 - Schematic structure of multi-blade centrifugal fan

1 - Volute shell; 2 – Impeller; 3 - Impeller shaft; 4 - Volute tongue; 5 - Dividing plate

Table 1

Main structural parameters of the fan dimensions

Name	Sizes	Name	Sizes
Impeller inner diameter D_1	48 mm	Outer diameter of impeller D_2	262 mm
Number of blades Z	16	Inlet Diameter D_0	240 mm
Width of volute case B_k	314 mm	Height of air vents S	157 mm
Extended dimensions of the volute housing A_f	56 mm	Distance between impeller end face and housing f	5 mm
volute outlet length C	220 mm	Air outlet diffusion angle α	2°

Wind turbine similarity theory is an important theoretical basis used in the field of fluid mechanics to analyze and predict the performance of wind turbines. The theory provides a scientific basis for the design of wind turbines by establishing the geometrical, kinematic and dynamic similarity relationship between the model and the prototype *(Wang et al., 2012)*. This means that when the conclusions drawn in this study are applied to the actual design optimization of combine harvester fans, the parameters can be converted according to the similarity criterion to match the working condition requirements of different models. The relevant formulas of the similarity law for fans are shown in Table 2, where the formula labeled "" indicates the known parameters.

Table 2

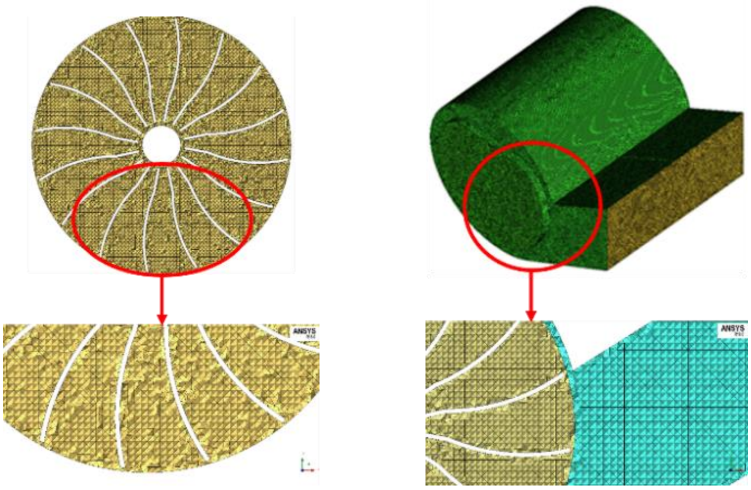
Calculation formulas based on wind turbine similarity theory

scientific law	calculation formula
Similarity law for flow rate	$\frac{Q}{Q'}=\frac{n}{n'}(\frac{D_2}{D_2'})^3$
Similarity law for full pressure	$\frac{P_{tF}}{P'_{tF}}=\frac{\rho}{\rho'}(\frac{n}{n'})^2(\frac{D_2}{D_2'})^2$
Similarity law for power	$\frac{N_i}{N'_i}=\frac{\rho}{\rho'}(\frac{n}{n'})^3(\frac{D_2}{D_2'})^5$
Similarity law for efficiency	$\eta_i=\eta'_i$

Where: Qv is the flow rate, m^3/s; n is the rotational speed, r/min; D_2 is the outer diameter of the impeller, mm; P_{tF} is the total pressure, Pa; ρ is the density of the gas, kg/m^3; P_{sh} is the shaft power, kW; η is the efficiency of the fan, %.

Grid Division

The flow channel model meshing is based on ICEM CFD in ANSYS 18.0. Structured meshes are used for the impeller channels and unstructured meshes are used for the volute shell channels. And the interface is set between the different flow channels to ensure the continuity of the flow at the interface. The Merge function is used to merge the two meshes, and the mesh schematic is shown in Fig. 2.



(a) Impeller meshing (b) Overall meshing

Fig. 2 - Effect of mesh division

Numerical Calculation Simulation Setup

Numerical simulations are performed based on Fluent in ANSYS 18.0, with steady state calculations first, and transient calculations after convergence of the calculations using the results of the constant calculations as initial values. The turbulence model is the RNG k-epsilon model, and the velocity-pressure coupling is done by SIMPLE algorithm with no-slip wall condition and enhanced wall function near the wall. The diffusion term, convection term and turbulent viscosity coefficients in the momentum equations are in second-order windward differential format. The inlet boundary is the pressure inlet with a value of 220 Pa, and the outlet boundary is the pressure outlet with a value of 0 Pa. The impeller speed is 1000 r/min, the maximum number of iteration steps is set to 20, the convergence residual is 10^{-3}, and the number of time steps is set to 250.

Numerical simulation validation

In order to verify the accuracy of the simulation results, a multi-blade centrifugal fan performance testing bench was built, as shown in Fig. 3. It mainly consists of multi-blade centrifugal fan, testing duct, sensor, motor, variable frequency controller, data acquisition system, and upper computer.

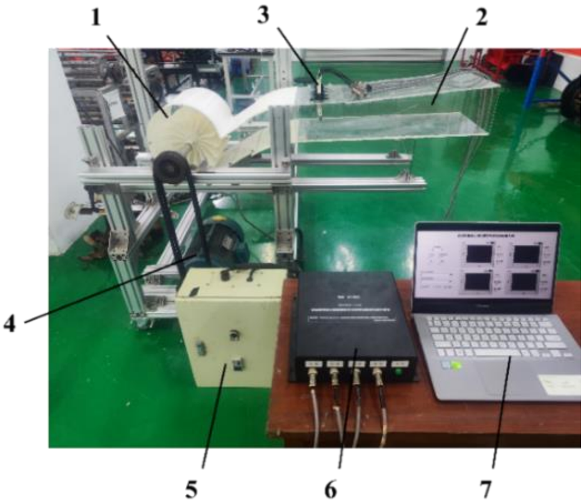


Fig. 3 - Multi-blade centrifugal fan performance testing bench

1 - Multi-blade centrifugal fan; 2 - Detection duct; 3 – Sensor; 4 – Motor; 5 - Frequency conversion controller; 6 - Data acquisition system; 7 - Upper computer

By numerical simulation of the fan under different working conditions, the simulation and test comparison curves of full pressure and efficiency are obtained as shown in Fig. 4. The results show that the numerical simulation results are in good agreement with the test data, with an error of no more than 10%, which verifies the reliability of the numerical simulation and can be used for subsequent simulation analysis.

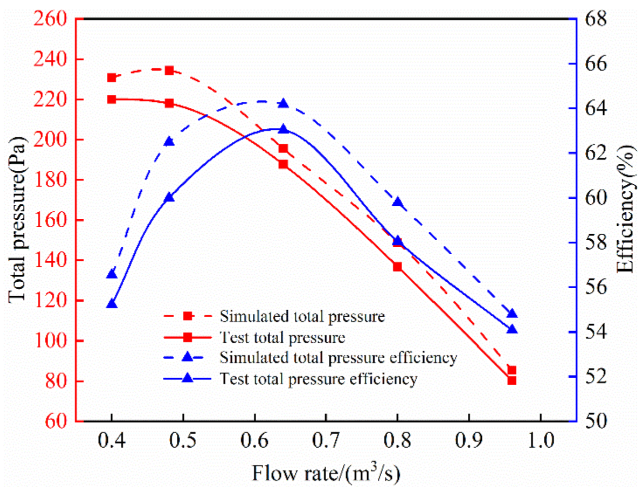


Fig. 4 - Comparison of numerical simulation and test results for prototype fan

Design of Matching Schemes for the Width-Diameter Ratio of Fan Volutes

Fig. 5 shows the axial structural schematic diagram of the volute and impeller of the multi-blade centrifugal fan. During the design of the scheme, the outer diameter and width of the impeller are kept unchanged, and only the width of the volute is changed to alter the axial matching between the volute and the impeller.

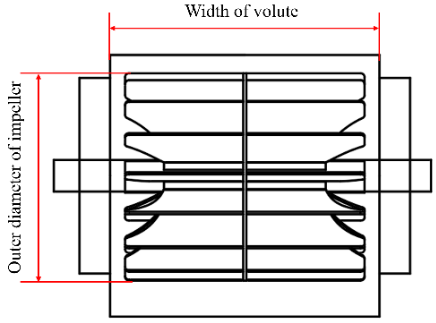


Fig. 5 - Schematic diagram of axial structure matching between volute and impeller

Since the size of the centrifugal fan will change with the change of the feeding amount of the combine harvester, in order to ensure the universality of the test results, the width of the volute is made dimensionless. The width-diameter ratio λ of the volute is defined as the ratio of the width of the volute to the outer diameter of the impeller, and the calculation formula is as follows:

$$\lambda = B_k / D_2 \tag{1}$$

where: B_k is the width of the volute, mm; D_2 is the outer diameter of the impeller, mm.

Four groups of test schemes are set up for grouped tests. The specific schemes are shown in Table 3.

Table 3

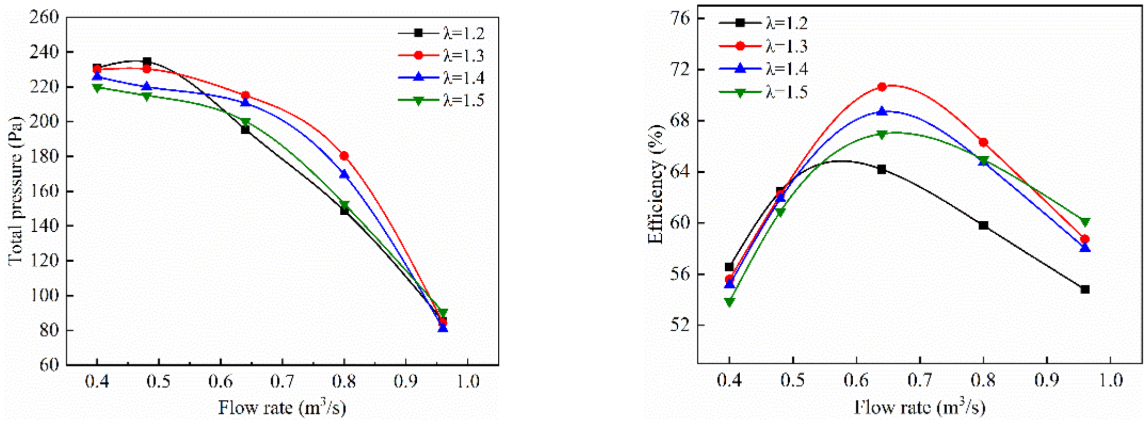
Experimental program for different volute width-to-diameter ratios

Experimental group	Outer diameter of impeller D_2 [mm]	Width of volute B_k [mm]	Volute width-diameter ratio λ /
1	262	314	1.2
2		340	1.3
3		366	1.4
4		393	1.5

RESULTS AND ANALYSIS

Performance Curves of Multi-blade Centrifugal Fans

The fan performance curve is used to characterize the relationship between the main performance parameters of the fan, such as air volume, air pressure, power and efficiency, etc., under a certain rotational speed and inlet conditions, reflecting the operating trend of the fan. In order to investigate the effect of different casing aspect ratios on the performance of centrifugal fans, this study uses CFD to numerically simulate the centrifugal fans with four casing designs at a rated speed of 1000 r/min, and obtains the total-pressure and efficiency performance curves of multi-blade centrifugal fans as shown in Fig. 6.



(a) Flow-Total Pressure Performance Curve (b) Flow-Efficiency Performance Curve

Fig. 6 - Schematic structure of multi-blade centrifugal fan

Fig. 6(a) illustrates that the total pressure of all four fan configurations generally decreases with increasing flow rate. The λ=1.3 fan exhibits superior total pressure across most flow ranges compared to other designs. The λ=1.2 configuration achieves peak total pressure at 0.48 m³/s through optimized kinetic-diffusion balance, yet its narrow flow path causes severe frictional losses at higher flows. Conversely, the λ=1.5 fan demonstrates higher pressure recovery in high-flow regimes (0.86~0.96 m³/s) due to reduced flow blockage from its enlarged cross-section.

The efficiency curves in Fig. 6(b) reveal parabolic trends with distinct peaks. The λ=1.3 configuration achieves maximum efficiency improvement (6.43% over the prototype), indicating optimal flow matching and minimal impact losses in the 1.3~1.4 width-diameter ratio range. The λ=1.2 fan shows left-shifted efficiency peaks, though efficiency plummets at higher flows due to axial spacing limitations. The λ=1.5 design excels in high-flow scenarios but suffers from vortex-induced losses at low flows due to incomplete diffusion.

Comprehensive analysis demonstrates that λ=1.3 provides balanced aerodynamic performance across operational ranges, making it ideal for general applications. The λ=1.2 configuration suits specialized low-flow/high-pressure scenarios, while λ=1.5 adapts to high-flow/low-loss requirements. Optimal width-diameter ratio selection requires balancing efficiency enhancement against potential aerodynamic losses, with 1.3~1.4 identified as the preferred range for multi-blade centrifugal fans in agricultural cleaning systems. These findings establish quantitative guidelines for fan geometry optimization under varying operational demands.

Comparative analysis of air velocity uniformity of outlet airflow

In grain cleaning operations, the airflow distribution characteristics of a multi-wing centrifugal fan directly affect the cleaning quality and operational efficiency. Studies have shown that when the airflow parameters (including flow rate, dynamic pressure and static pressure) at the air outlet of the fan show a uniform distribution along the impeller width direction, the grain cleanliness can be significantly improved and the sorting loss can be reduced *(Tang et al., 2007)*. The uniformity of the lateral distribution of wind speed at the air outlet is measured by the wind speed variance V, which is calculated as follows:

$$V = \frac{\sum_{i=1}^{n} (a_i - a)^2 \Big/ N-1}{a} \tag{2}$$

where: a_i is the wind speed value of the i-th measurement point, m/s; a is the average value of wind speed of the turbine, m/s; N is the number of measurement points of the turbine.

Under the rotational speed of 1000 r/min, three monitoring areas are divided at equal spacing in the vertical direction of the fan outlet, which are the upper L_1, middle L_2, and lower L_3, and 10 monitoring points are arranged at equal spacing on each monitoring line, and the distribution of the monitoring points is shown in the schematic diagram of Fig. 7.

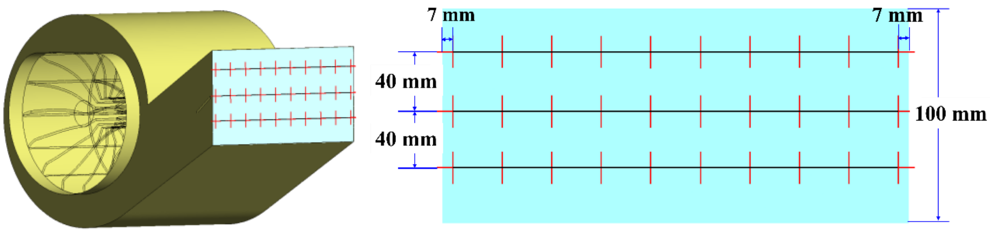
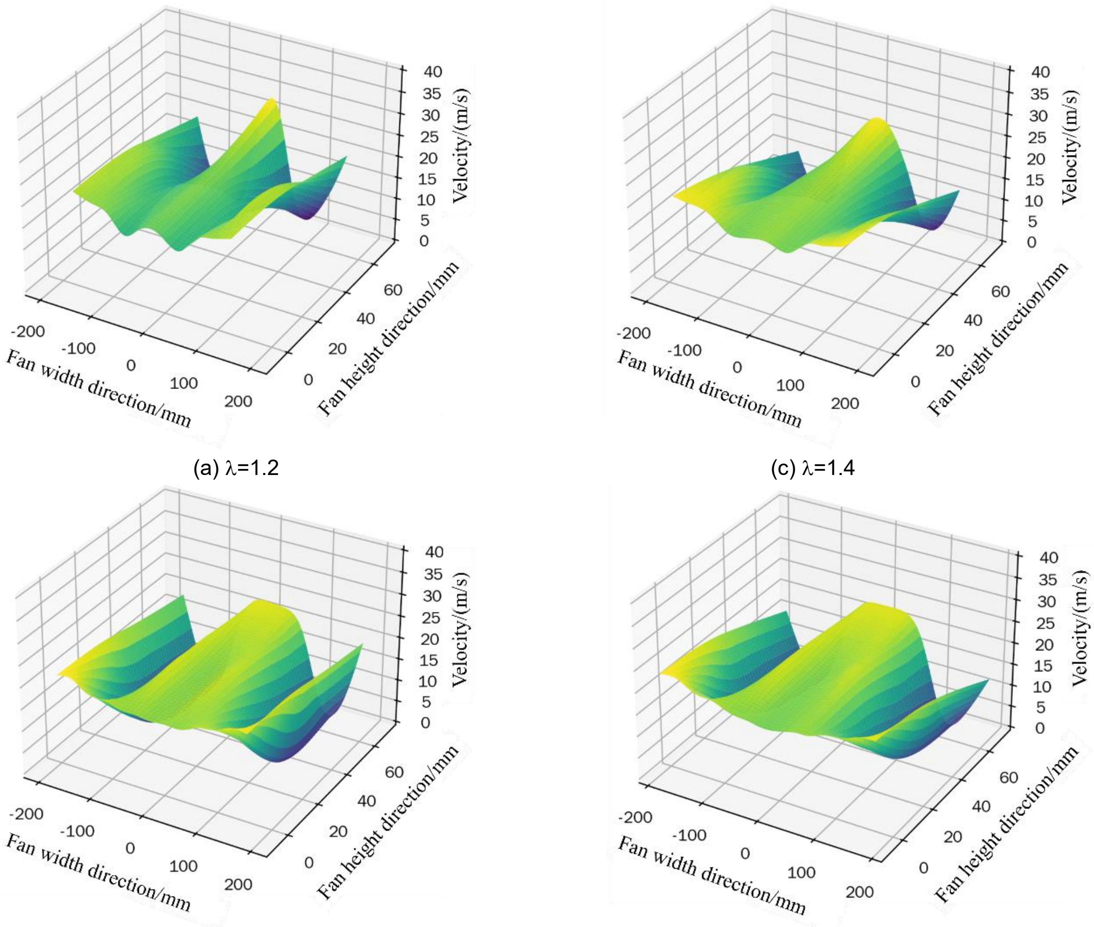


Fig. 7 - Schematic distribution of monitoring points of centrifugal fan outlet cross-section

According to the value of wind speed at the measurement point, the three-dimensional curve of wind speed at the air outlet is plotted, as shown in Fig. 8. It can be seen that the wind speed in the upper and middle layers is lower than that in the lower layer of the outlet, and the fluctuation is larger. The upper airflow curve shows a typical "W" distribution, with a high-speed jet protruding from the center axis region. The symmetric double-vortex structure is generated by the flow separation effect of the volute shell gap near the two sides of the volute shell wall, resulting in a sudden drop of the wind speed below 5 m/s. The wind speed of the upper layer is lower than that of the lower layer of the outlet, and the fluctuation is large. When λ increases from 1.2 to 1.5, the width share of the high-speed core area expands from 32% to 48%, but the peak wind speed decreases from 24.350 m/s to 22.175 m/s, which is a decrease of 8.93%, indicating that the increase of the width-to-diameter ratio weakens the centrifugal acceleration effect. The lower layer of the outlet is approximately "M" type distribution, the wind speed is higher overall, and the distribution is smoother.



(a) λ=1.2 (c) λ=1.4

(b) λ=1.3 (d) λ=1.5

Fig. 8 - Three-dimensional curves of wind speed distribution at the outlet of fans with different fans

Calculating the wind speed variation values of the four fans is shown in Table 4, and the results show that the wind speed variation value of the fan with a λ of 1.3 is the smallest in the outlet section, which is 2.426, and is 16.63% lower than that of the prototype fan, which is 2.910, and the wind speed variation value of the fan with a λ of 1.5 is the largest, which is 3.496, namely 44.11% higher than that of the optimal structure, indicating that its wind speed distribution uniformity is the poorest. This suggests that the over-expanded volute flow channel exacerbates the flow separation and energy dissipation.

Table 4

Variability of air velocity at the outlet of four types of fans

Volute width-diameter ratio λ	Wind speed variability V
1.2	2.910
1.3	2.426
1.4	3.830
1.5	3.496

Comprehensive analysis of the above data shows that the change of the volute width-diameter ratio has a significant effect on the uniformity of the wind speed distribution at the fan outlet, and a suitable volute casing aspect ratio can effectively enhance the qualitative characteristics of the fan and improve the airflow uniformity inside the impeller. λ=1.3 is the optimal solution under the current design parameters, and the synergistic optimization of low turbulence dissipation and high uniformity is achieved by balancing the centrifugal effect and the flow channel geometry.

Modeling of volute width-diameter ratio-speed-air volume function based on MATLAB

According to the results of the numerical simulation, as the rotational speed of the fan changes, the relationship between the width-diameter ratio of the volute and the air volume will also change accordingly.

However, most of the existing studies focus on the influence of a single parameter on the performance of the fan, and there is a lack of quantitative characterization of the multi-dimensional correlation among λ, n, and Qv. This makes it difficult to precisely match the requirements of dynamic working conditions. Therefore, constructing an accurate function model to describe the interaction among the width-diameter ratio of the volute, the rotational speed, and the air volume can not only improve the working efficiency of the fan but also provide a theoretical basis for the design of the fan.

In order to achieve efficient model fitting and validation, the MATLAB R2022a platform is used, whose advantages are: MATLAB's library of nonlinear regression algorithms supports the parameter identification of complex physical models, its multivariate statistical toolbox function can synchronously assess the fitting excellence of polynomial regression and power law models, and with its powerful numerical calculation and visualization functions, it can intuitively reveal the interactive effects of each parameter. The multivariate fitting analysis of wind turbine parameters is efficiently completed. The fitted multivariate function model is:

$$Q_v = f\left(\lambda, n\right) = k \cdot \lambda^a \cdot n^b \tag{3}$$

where: k is a constant coefficient; a and b are power indices to be determined; and C is a constant term.

The discrete data were fitted using nonlinear least squares and the best k=0.0217, a=0.7948, b=0.4621 where Residual Sum of Squares=0.0053, Adj.R-Square=0.9444, the residual sum of squares is small, and the corrected coefficient of determination shows the success of the fit. The results of the fit are as follows:

$$Q_v = 0.0217 \cdot \lambda^{0.7948} \cdot n^{0.4621} \tag{4}$$

This functional relationship equation has the aspect ratio index a = 0.7948, which indicates that increasing the width-diameter ratio can enhance the airflow, but the gain decays with increasing λ. The three-dimensional surface plot of λ and n as independent variables and Q as dependent variable comprehensively shows the interrelationship between λ, n and Q as shown in Fig. 9.

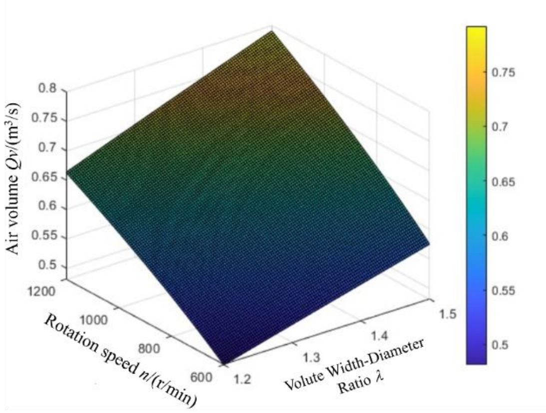
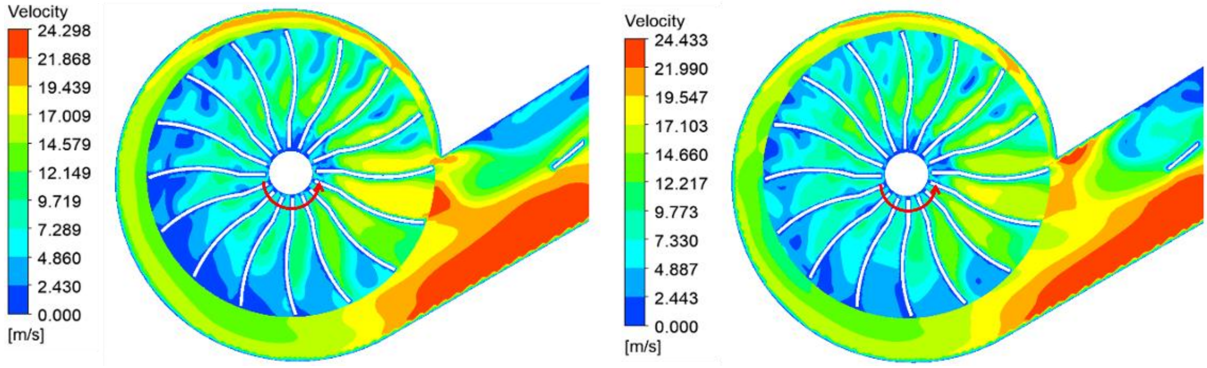


Fig. 9 - Three-dimensional surface diagram of the relationship between volute width-to-diameter ratio-speed-air volume

Simulation analysis of flow field of multi-blade centrifugal fan before and after optimization

Numerical simulation results show that the efficiency of the optimized fan with λ=1.3 reaches 70.61%, which is 6.43% higher than the 64.18% of the original fan. The air volume increases to 0.636 m³/s with a growth rate of 6.53%. This indicates that the optimization of the width-to-diameter ratio of the volute has achieved preliminary results, and the performance of the fan has been improved. The simulation results of multiblade centrifugal fan with λ=1.2 (original model) and λ=1.3 (optimized model) are selected for comparative analysis. Fig. 10 shows the comparison of the velocity clouds of the two centrifugal fans in the middle section.

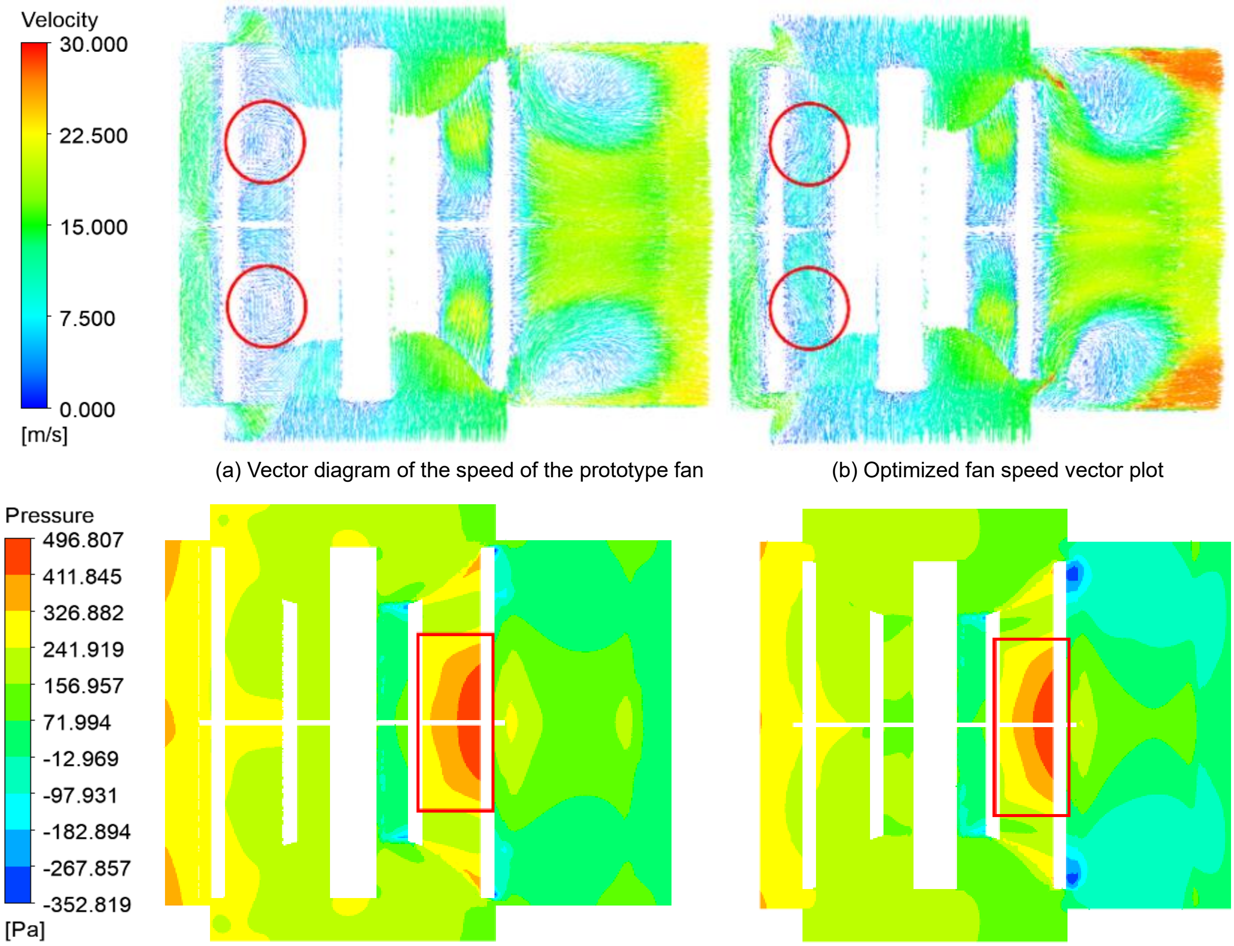
Through comparison, it can be found that the optimized velocity distribution has been significantly improved. The area of the low-velocity recirculation zone on the upper wall surface of the air outlet has been reduced, especially the low-velocity zone inside the impeller and the flow uniformity have been greatly improved. This improvement essentially stems from the fact that after optimizing the axial matching degree between the volute width and the impeller, the secondary flow of the airflow in the meridian plane has been effectively suppressed. Specifically, increasing the volute width reduces the radial velocity gradient and improves the matching between the airflow angle at the impeller outlet and the volute, thereby decreasing airflow impact losses. The phenomenon that the high-speed flow region with (v>21 m/s) concentrates towards the volute outlet confirms the reconstruction effect of the optimized width-diameter ratio of the volute on the jet-wake structure, effectively improving the conversion rate of the airflow kinetic energy.



(a) Prototype fan (b) Optimizing fan

Fig. 10 - Velocity clouds on the middle section of the turbine before and after optimization of the volute width-to-diameter ratios

Fig. 11(a) and (b) present the velocity vector diagrams of the fan on the Y-Y plane. It can be clearly seen that the outlet velocity of the optimized fan has increased. The maximum velocity has increased from 24.18 m/s to 30.048 m/s, an increase of approximately 24%. In Fig. 11(a), the symmetrical vortex structure that existed in the middle of the suction surface of the original impeller has completely disappeared after optimization. The pressure nephograms in Fig. 11(c) and (d) show that the total pressure decreases to a certain extent along the direction of the volute outlet, indicating that kinetic energy dissipation occurs when the airflow flows in the volute. The reasons for this energy loss mainly come from two aspects: on one hand, there is frictional loss between the airflow and the volute wall and internal structures; on the other hand, the sudden change in the volute tongue structure leads to airflow separation and turbulent pulsation.



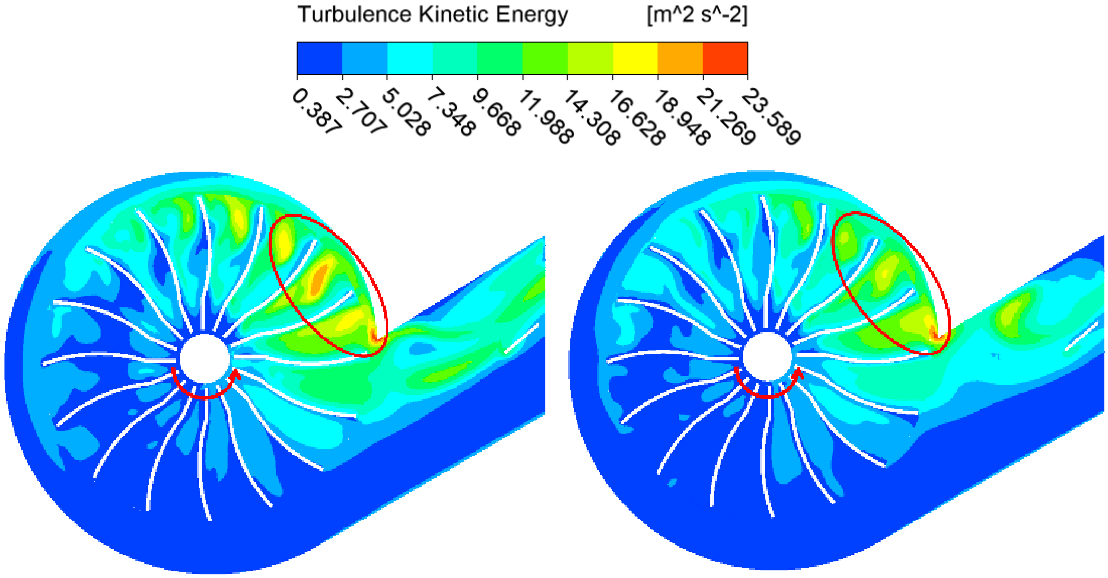
(a) Vector diagram of the speed of the prototype fan (b) Optimized fan speed vector plot

(c) Pressure distribution of the prototype fan (d) Optimization of fan pressure distribution

Fig. 11 - Y-Y plane velocity vector map and pressure cloud before and after optimization of volute width-to-diameter ratios

Turbulent Kinetic Energy (TKE) expresses the kinetic energy per unit mass of gas associated with the vortex in a turbulent flow. To some extent, it can reflect the stability of the airflow. Fig. 12 extracts the turbulent kinetic energy cloud for the middle section of the impeller of the multibladed centrifugal fan before and after optimization. The cloud diagram shows that the high energy consumption zone is mainly concentrated near the volute tongue, including the transition zone near the impeller front disk and downstream of the volute tongue, indicating that the fluid flow in this region is relatively complex.

The turbulence kinetic energy of the optimized volute casing decreases significantly compared with that of the prototype fan, and the turbulence of the airflow in the impeller channel and at the front end of the flow path improves significantly, indicating that the optimization of the volute casing width-to-diameter ratio makes the internal flow of the fan more smooth, and improves the stability and uniformity of the airflow, which is conducive to the generation of a more uniform airflow, and facilitates the effect of the cleaning action.

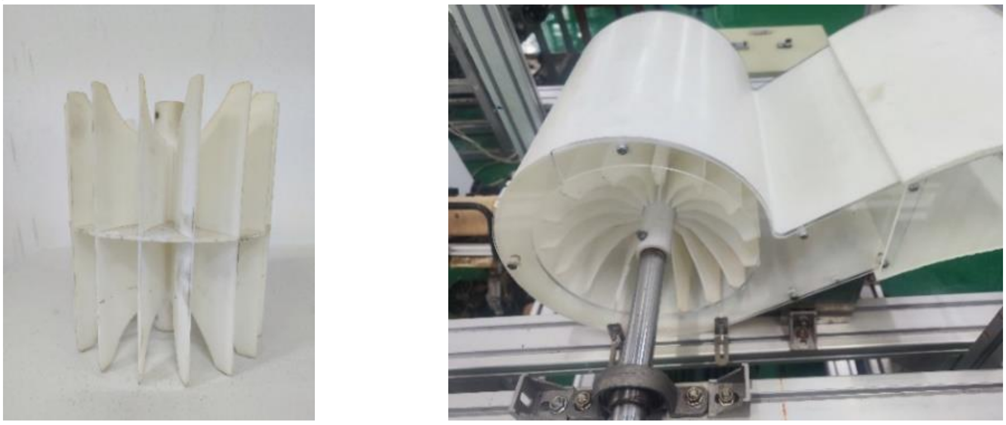


(a) Prototype fan (b) Optimizing fan

Fig. 12 - Turbulent kinetic energy distribution in the middle section of the fan before and after optimization

Experimental verification

To ensure the reliability of the optimized results, the optimized volute was machined and tested using 3D printing. All test conditions were the same as for the prototype. The physical diagram of the optimal fan is shown in Fig. 13.



(a) Fan impeller (b) Multi-blade centrifugal fan

Fig. 13 - 3D printed optimal fan physical drawing

Table 5 shows the comparison between the bench test data of the optimized solution for the multi-blade centrifugal fan volute structure and the data of the prototype fan.

As shown by the data in Table 5, the optimal fan achieved a 14.6% increase in air volume, an 11.8% increase in total pressure, and a 9.22% increase in efficiency compared with the prototype fan.

Although the shaft power increases by 3.8%, the increase in airflow, total pressure and efficiency confirms that the structural optimization not only improves the performance of the working conditions, but also expands the efficient working range of the fan, which indicates that the energy utilization is more reasonable. Moreover, the results of the bench test are similar to those of the numerical simulation, which verifies the reliability of the numerical simulation. The increase in air volume and full pressure of the optimal fan is due to the smoother flow of air in the fan after the optimization of the volute casing structure, which reduces the energy loss and enables the gas to be transported and pressurized more efficiently.

Table 5

Bench test comparison test data

Program	Air volume [m^3/s]	Total pressure [Pa]	Efficiency [%]	Effective power [KW]	Shaft power [KW]
Prototype fan	0.658	209.73	71.36	0.138	0.164
Optimizing fan	0.574	187.66	62.14	0.105	0.158

CONCLUSIONS

Aiming at the current situation of low efficiency of fans with flat blade structure in the agricultural field and less research and application of multi-blade centrifugal fans, this study analyzes the influence of the volute width-to-diameter ratio on the performance of agricultural cleaning centrifugal fans, aiming to provide a selection basis for the design and optimization of high-performance multi-blade centrifugal fans. At a rated speed of 1000 r/min, the multi-blade centrifugal fan with an aspect ratio of 1.3 has the best overall performance. Compared with the prototype fan, the efficiency is increased by 7.13%, the total pressure is increased by 10%, and the air volume is increased by 6.53%. At the same time, the fan has the smallest variance value of 2.426 for the outlet air velocity, which is 16.63% lower than that of 2.910 for the prototype fan, indicating that the optimization of the width of the volute casing improves the uniformity of the outlet airflow, which is conducive to the cleaning effect of the airflow. Based on the MATLAB platform, a multi-dimensional function model of the volute casing width-to-diameter ratio-speed-to-airflow is established, which fills the gap of the existing research focusing on a single parameter to quantitatively characterize the performance of the turbine. The validation results of the bench test show that the error between simulation and test is within 10%, which verifies the reliability of numerical simulation. According to the fan similarity theory, the universal law of the relationship between the width of the shell and the performance of the fan can be used for the optimization design of the shell of the combine harvester fan, which can be adapted to match different specifications of the models.

ACKNOWLEDGEMENT

This research was financially supported by National Natural Science Foundation of China (52375249), National Natural Science Foundation of China (32401690).

REFERENCES

[1] Acarer, S., Uyulan, Ç., & Karadeniz, Z. H. (2020). Optimization of radial inflow wind turbines for urban wind energy harvesting. *Energy*, *202*, 117772. https://doi.org/10.1016/j.energy.2020.117772

[2] Almoslh, A., Alobaid, F., Heinze, C., & Epple, B. (2021). Experimental Study of the Influence of Gas Flow Rate on Hydrodynamic Characteristics of Sieve Trays and Their Effect on CO2 Absorption. *Applied Sciences*, *11*(22), 10708. https://doi.org/10.3390/app112210708

[3] Babubhai, V. K. P., Chaudhari, A., Sharma, A., & Diwakar, V. (2023). Experimental and numerical study on the influence of flow passages in centrifugal fan using computational fluid dynamics. *Engineering Research Express*, *5*(2), 025030. https://doi.org/10.1088/2631-8695/accf00

[4] , I., Mudarisov, S., Lukmanov, R., Permyakov, V., Ibragimov, R., & Nasyrov, R. (2019). Mathematical modeling and research of the work of the grain combine harvester cleaning system. *Computers and Electronics in Agriculture*, *165*, 104966. https://doi.org/10.1016/j.compag.2019.104966

[5] Chai, X., Xu, L., Sun, Y., Liang, Z., Lu, E., & Li, Y. (2020). Development of a cleaning fan for a rice combine harvester using computational fluid dynamics and response surface methodology to optimise outlet airflow distribution. *Biosystems Engineering*, *192*, 232-244. https://doi.org/10.1016/j.biosystemseng.2019.12.016

[6] Liang, Z., & Wada, M. E. (2023). Development of cleaning systems for combine harvesters: A review. *Biosystems Engineering*, *236*, 79-102. https://doi.org/10.1016/j.biosystemseng.2023.10.018

[7] Liang, Z., Li, D., Li, J., & Tian, K. (2020). Effects of fan volute structure on airflow characteristics in rice combine harvesters. *Spanish Journal of Agricultural Research*, *18*(4), e0209-e0209. https://doi.org/10.5424/sjar/2020184-15426

[8] Ma, Z., Zhu, Y., Li, J., Song, Z., Yu, J., Li, Y., & Xu, L. (2024). Study on the performance of detection air duct and evaluation index of agricultural cleaning centrifugal fans. *Biosystems Engineering*, *239*, 81-97. https://doi.org/10.1016/j.biosystemseng.2024.01.015

[9] Ottersten, M., Yao, H. D., & Davidson, L. (2022). Inlet gap influence on low-frequency flow unsteadiness in a centrifugal fan. *Aerospace*, *9*(12), 846. https://doi.org/10.3390/aerospace9120846

[10] Schäfer, R., & Böhle, M. (2020). September. Validation of the lattice Boltzmann method for simulation of aerodynamics and aeroacoustics in a centrifugal fan. *In Acoustics*, *2*(4), 735-752. https://doi.org/10.3390/acoustics2040040

[11] Shim, H. S., & Kim, K. Y. (2020). Design optimization of the impeller and volute of a centrifugal pump to improve the hydraulic performance and flow stability. *Journal of Fluids Engineering*, *142*(10), 101211. https://doi.org/10.1115/1.4047539

[12] Sun Wen-sheng, Yi Shu-juan, Qi Hai-long, Li Yi-fei, Dai Zhi-bo, Zhang Yu-peng, Yuan Jia-sha, Wang Song. (2025). Design and Optimization of Air-Assisted Spiral Seed-Supply Device for High-Speed Dense Planting Maize Seeder. *INMATEH - Agricultural Engineering*, *75*(1), pp.13-25. https://doi.org/10.35633/inmateh-75-01

[13] Tang, C., Zhang, Ouyang, Z., Luo, H., Fang, Y., & Wu, B. (2007). Experimental research on the combined centrifuge-aerofoil grain cleaning fan *(离心—轴流组合式清粮风机的试验研究)*. *Transactions of the Chinese Society of Agricultural Engineering*, *23*(10), 117-120. https://doi.org/10.3321/j.issn:1002-6819.2007.10.020 *(In Chinese)*

[14] Varun Ch. S., Anantharaman, K., & Rajasekaran, G. (2023). Effect of blade number on the performance of centrifugal fan. *Materials Today: Proceedings*, *72*, 1143-1152. https://doi.org/10.1016/j.matpr.2022.09.185

[15] Wang, L., Gu, B., & Wang, T. (2012). Fast Calculation Model of Fan Performance Based on Similarity Theory *(基于相似理论的风机性能快速计算模型)*. *Fluid Machinery*, *40*(07), 24-28. https://doi.org/10.3969/j.issn.1005-0329.2012.07.006 *(In Chinese)*

[16] Wang, L., Jiao, Z., & Zhao, G. (2017). Simulation Analysis of Centrifugal Fan in Air Screen Type Cleaning Device *(风筛式清选装置离心风机气流场仿真分析)*. *Agricultural Equipment & Vehicle Engineering*, *55*(08), 10-14. https://doi.org/10.3969/j.issn.1673-3142.2017.08.003 *(In Chinese)*

[17] Wei, Y., Zhu, L., Zhang, W., & Wang, Z. (2020). Numerical and experimental investigations on the flow and noise characteristics in a centrifugal fan with step tongue volutes. *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science*, *234*(15), 2979-2993. https://doi.org/10.1177/0954406219890920

[18] Yang, L., Yaoming, L., Liya, L., & Maolin, S. (2024). Modeling and control methods of a multi-parameter system for threshing and cleaning in grain combine harvesters. *Computers and Electronics in Agriculture*, *225*, 109251. https://doi.org/10.1016/j.compag.2024.109251