Simulation and optimum design of small fan for animal house by CFD

Zhaoxia Liu¹,a, Yingqing Xiong¹,b, Yan Chen¹,c*

¹ College of Mechanical and Electronic Engineering, Wuhan Donghu University, Wuhan 430017

*a-mail: zhaoxia9980@163.com, b-mail: Yingqingxiong@163.com

*c-mail: chenyangwhpu.edu.cn

* Corresponding author’s e-mail: chenyang@whpu.edu.cn

Abstract: In order to improve the aerodynamics performance of the fan in animal house, the extractor for animal house (model: 400-4-2) has been taken as the optimization object, and the Computational Fluid Dynamics (CFD) method was used to analyze the airflow nearby the whole extractor. The FreescaX5 3D scanner was used to collect data and reconstruct 3D model of the prototype fan. Then, the prototype fan model was optimized by CFD simulation to investigate the performance and defects of the prototype fan, and thereby to propose the improvement scheme. The simulation results of flow field near the blade showed that the performance of fan was influenced by the impeller numbers, the bending direction and the blade shape. In the numerical simulation, when the wind speed was used as the evaluation standard, the parameters of the optimized fan are as follows, the blade number is 7, the blade bending direction is forward bending, the blade tip angle and root angle is 9.5° and 58.0° respectively, and the blade cross-section shape is streamlined with small airfoil curvature. The air volume of the optimized fan increased by 3.86% when the optimized fan is compared with the prototype fan at the same power.

1. Introduction

In recent years, the scale of the livestock and poultry husbandry in China has increased rapidly, and large-scale breeding has become a development trend. The temperature and humidity in the livestock and poultry house are the most important factors, and the way to adjust the environment in livestock and poultry house is mechanical ventilation. Ventilation can increase the air circulation and keep the appropriate temperature and humidity. Meanwhile, ventilation can effectively eliminate toxic and harmful gases, dust and pathogenic microorganisms, and improve the environmental quality of livestock and poultry house. The amount of ventilation is the key to maintain the air quality, animal comfort and harmful gas emissions in the house. However, strong winds may cause a sudden drop in the temperature of the house, which will lead to livestock and poultry cough, cold and other diseases. Therefore, improving the performance of the ventilation system is of great significance for the environment of livestock and poultry house and preventing a variety of diseases in high-density livestock and poultry houses.

The fan is the core equipment in the mechanical ventilation system, and it directly affects the ventilation performance of the livestock and poultry house. The influencing factors of fan performance include the number of blades, section shape, installation angle and bending direction. In recent
research on agricultural fans, it has been found that the fan with more blades have greater total pressure, but the efficiency shows a trend of first increasing and then decreasing. The section shape and thickness of blade can also affect the performance of the fan. With the increase of blade section radian, the noise and flow rate of the fan first increase and then decrease. The outlet flow increases with the increase of chord length at the blade tip and blade root, while decrease with increase of chord length in the blade. The forward-curved-blade fan (FCF) can reduce the energy loss of the low speed axial converter and improve the total pressure loss. When other structural parameters of the impeller remain unchanged, the air volume of the impeller increases with the increase of the blade installation angle, but the efficiency shows a trend of first increasing and then decreasing. These studies provide guidance for setting parameter and optimizing structure of fan in existing livestock and poultry house. However, there is no effective solution to improve the aerodynamic performance of fan in the house.

In view of the above problems, the numerical simulation method has been adopted to reconstruct the common fans of livestock house. The CFD model based on ANSYS Workbench platform was selected to analyze the airflow field nearby the whole fan and the blade, in order to investigate the performance and defects of the prototype fan. The flow velocity, static pressure value and the flow rate was selected as evaluation indexes to optimize the impeller structure of fan. The results can provide guidance for increasing fan ventilation and reducing loss rate.

2. Methods

2.1 Reverse reconstruction of prototype fan

In this paper, the fan produced by Zhongsheng Farming Equipment Factory (Shandong, China) was selected as the research object, the parameters of the fan were shown in Table 1.

| Parameters                      | Values |
|---------------------------------|--------|
| Speed (r/min)                   | 1500   |
| Air Volume (m³/h)               | 6650   |
| Capacity (W)                    | 1400   |
| Number of blades                | 8      |
| Diameter of impeller (mm)       | 460    |
| Diameter of hub (mm)            | 130    |

The fan model mainly includes three parts: the fan shell, electric motor and the impeller. In the simulation calculation, the fan shell is simplified as the fan rotation domain, and the motor is replaced by the impeller rotation setting and the boundary setting of the air inlet. The mark point was pasted and then the point cloud data was collected. The working distance of the scanner is 300 mm.

The point cloud data of impeller collected by 3D scanner is shown in figure 1(a). The data was converted into accurate polygon model by point processing and polygon processing methods, and then the impeller hub and blade were reconstructed, the obtained model was shown in figure 1 (b).

Figure 1 Impellers point cloud data diagram (a) and Physical model of impellers (b)

In order to analyze the performance and defect of the original fan in detail, ANSYS software was used to analyze the flow field near the fan blade. A multi-region calculation model was adopted, and the model was divided into three regions: inlet region, rotating region and outlet region. The origin of
the model is set as the midpoint of the outlet surface of the rotation zone, and the air inlet direction is along the positive Z-axis.

Considering the impeller structure and calculation accuracy, unstructured grid was used to calculate the impeller. Due to the flow field in the rotating region changed greatly, and the impeller structure was very complicated, the region was divided into 1437225 grids, and the grid size of the inlet and outlet regions was set to 10mm, and the number of grids is 1336369. The final elements of the fan is 2773594. The results of fan meshing and impeller meshing were shown in figure 2. The average element quality, the average aspect ratio, the average Jacobian ratio and the minimum volume is 0.84, 1.84, 1.04 and \(7.0188569 \times 10^{-12}\), respectively. The method accorded with the requirement of calculation precision.

![Figure 2 Grid division of model fan (a) and impeller (b)](image)

The flow field of the impeller was a turbulent model, and the Reynolds number \(R_e\) in the passage is \(3.88 \times 10^5\). RNG k - \(\varepsilon\) model and the second order upwind scheme were used to calculate. In order to further ensure the accuracy of the calculation, the solver adopts.

The boundary conditions were as follows: the inlet was set as mass-flow-inlet, and the flow rate is set to 1.5329kg/s. The inlet direction was set as perpendicular to the boundary. The outlet was set as free flow outlet. The impeller speed was set as the rated speed of the fan, and the rotation direction was along the Z-axis. The section at 2/3 distance from the impeller radius was taken as the research object, and the section shape and calculation domain are shown in figure 3.

![Figure 3 Section of blade and computational domain](image)

c - chord length, mm, A, B, C, D - the boundary points of computational domain

3. Result and discussion

3.1 Flow field analysis

The streamline distribution of fan flow field was shown in figure 4. The streamline in the inlet area is straight, and the velocity direction of each particle is parallel to the rotation axis (along the positive Z-axis). There is no separation of the airflow, and it belongs to laminar flow. After the airflow passed through the rotating area of the impeller, there was an airflow in negative direction of Z-axis, so the airflow formed a backflow and showed a spiral state. When the gas flowed away from the rotating region, the air flow no longer produced backflow, and continued to flow along the Z-axis, but the number of streamlines at the outlet decreased.
The blade tip angle and the root angle of the original fan blade were 6.0° and 58.0°, respectively. Large eddies were generated on both the pressure surface and the suction surface near the leading edge of the blade, which may lead to the decrease of blade lift and increase of blade resistance. According to Figure 5, the velocity of the airflow near impeller surface increased gradually from the root to the tip. The velocity at the hub is the minimum, while the velocity at the tip is the maximum. There is a backflow phenomenon in the diagram of x = 0 section. The cusp of leading edge and trailing edge of blade can lead to the decrease of the lift, while, the increase of the drag. This may cause the air flow separation around the blade, resulting in a certain amount of energy loss.

In order to analyze the performance of the fan before and after the structure optimization, the total pressure of the fan is calculated. The calculation formula is as follows:

$$P_{out} - P_{in} = (P_2 + \rho_2 \times \frac{v_2^2}{2}) - (P_1 + \rho_1 \times \frac{v_1^2}{2})$$

Where, $P_1$ and $P_2$ were the static pressure of the fan inlet and outlet sections, $P_A$; $\rho_1$ and $\rho_2$ were the air density of the fan inlet and outlet sections, kg/m$^3$; $v_1$ and $v_2$ were the average air velocity of the fan inlet and outlet sections, m/s.

The static pressure nephogram of impeller suction surface and outlet surface was shown in figure 6. For the suction surface of impeller, there was a part of negative pressure region in the leading edge of blade, and the static pressure value of other areas was positive. The maximum static pressure appeared at 1/3 near the leading edge of blade. On the outlet side, the static pressure of the blade was mostly negative, and the static pressure of the trailing edge was larger than that of the leading edge. The velocity and static pressure of four sections were analyzed, and the results showed that the air flow velocity at the interface between rotation region and outlet region increased first, and then decreased along the radial direction. The flow velocity was maximum near the wall while the flow velocity was more uniform away from the impeller.
3.2 Optimization and performance analysis of fan

According to the simulation results of the flow field of the original fan, the existing problems and optimization scheme of the original fan were summarized. The specific scheme was shown in Table 2.

| position                  | Defect                                                                 | Optimization scheme                                      |
|---------------------------|------------------------------------------------------------------------|----------------------------------------------------------|
| Number of blades          | The blade with symmetrical distribution is easy to produce resonance.   | Using asymmetric odd-number blades to avoid resonance.   |
| Bending direction of blade| The backward curved blade has a large kinetic energy loss.              | The forward curved blade is selected.                    |
| Section shape             | The cusp of leading edge and trailing edge of blade leads to the decrease of lift | Streamline blade is adopted.                            |
| Installing angle of blade | The installation angle affects the airflow velocity and increases the loss. | Adjust installation angle to reduce energy loss.         |

According to the literature, when the fan flow rate increases by 20%, the static pressure will increase by 44%, and the power will also increase by 72.8%. Therefore, it is necessary to consider the flow amount and the power at the same time. In this study, the asymmetric odd-number blade was selected to avoid resonance and reduce the fatigue damage of the blade. Meanwhile, the leading edge and trailing edge of the blade were adjusted to fillet to reduce the influence of flow separation near the blade edge. The specific scheme was shown in Table 3.

| Scheme                        | Parameters                      |
|-------------------------------|---------------------------------|
| Blade number/°                | 3, 5, 7, 9                      |
| Tip angle of the blade/°       | 3.5 (FCF1), 6.5 (FCF2), 9.5 (FCF3), 12.5 (FCF4) |
| Root angle of the blade/°      | 54.2 (FCF1), 54.2 (FCF2), 58.0 (FCF3), 58.0 (FCF4) |
| Bending of blade               | 3%c, 4%c                        |

Taking the flow rate generated by the impeller under the same power as the evaluation index, the flow rate curves of different blades were obtained by simulation, and the results are shown in figure 7. It can be seen that the flow rate increased with the increase of fan power. The flow rate of 7-blade fan was the largest. This may be due to the larger area of each blade and the higher working speed of the impeller when there were fewer blades, which leads to the greater wind pressure on the blades and the great loss. As the number of blades increased, the wind pressure on a single blade decreased, so the loss of the fan was larger caused by the increasing number of collisions and separations.

The velocity increases gradually from the hub to the blade top when the impeller works. The difference of radial velocity can be compensated by changing the installation angle. It can be seen from figure 8b, the impeller flow increased with the increase of the impeller installation angle at the same power value, while the flow decreased when the installation continued to increase. In this paper, the installation angle to obtain the highest flow rate was as follows: the tip angle was 9.5°, and the root angle was 58.0°.

The curvature of impeller blade also affected the performance of impeller. According to the calculation results of the optimization scheme, the aerofoil 2 with blade bending 3%c showed higher efficiency and larger flow rate. This is mainly due to the optimized airfoil with streamline can delay the boundary layer separation and reduce eddy current loss. When air flowed around the curved surface, there will be a separation of airflow, and the pressure changed with the section thickness. From the leading edge of the blade, the air flow cross section became narrow, the flow velocity increased and the pressure decreased gradually. When the air flow reached to the thickest section of the impeller, the flow section became larger and the flow velocity decreased rapidly, and eventually may result in eddy current loss. The air volume of prototype impeller is 6618.69 m³/h at the rated
power 1400W, while the air volume of optimized impeller is 6874.17 m$^3$/h at the same power, which is 3.86% higher than that of the original fan.

![Air flow rate curve under different power](image)

**Figure 7** Air flow rate curve under different power

a - air flow curves of different number of blades, b - air flow curves of different installation angle, c - air flow curves of different blade bending

4. Conclusion

The numerical simulation results showed that the air flow rate and efficiency of optimized extractor were greatly improved. The flow rate was selected as evaluation index, the simulation results showed that the air volume of the optimized fan was 3.86% higher than that of the original fan. However, other factors such as noise also affected the performance of fan. The results of this study provide a reference for structure optimization of similar poultry house.

Acknowledgement

This work was supported by the Youth Fund Project of Wuhan Donghu University (2019dhzk001)

References

[1] Jung U H, Kim J H, Kim J H, Park C H, Jun S O, Choi Y S. (2016) Optimum design of diffuser in a small high-speed centrifugal fan using CFD & DOE. Journal of Mechanical Science & Technology, 30, 1171-1184.

[2] Ding T, Fang L, Ni J Q, Shi Z, Sun B, Wang Z, Yao C. (2018) Optimization of Diffuser Parameters for Mixing Fans in Agricultural Buildings. Applied Engineering in Agriculture. 2017, 34, 437–444.

[3] Nho Y C, Park J S, Lee Y J, et al. (2012) Effects of turbine blade tip shape on total pressure loss and secondary flow of a linear turbine cascade. International Journal of Heat & Fluid Flow, 33(1): 92-100.

[4] Kim K Y, & Seo S J. (2004). Shape optimization of forward-curved-blade centrifugal fan with navier-stokes analysis. Journal of Fluids Engineering, 126(5), 735-742.

[5] Pogorelov A, Meinke M, Schroeder, Wolfgang. (2016). Effects of tip-gap width on the flow field in an axial fan. International Journal of Heat & Fluid Flow, 61(OCT.), 466-481.

[6] Lee M H, Shiah Y C, Bai C J. Experiments and numerical simulations of the rotor-blade performance for a small-scale horizontal axis wind turbine. Journal of Wind Engineering and Industrial Aerodynamics, 2016, 149:17-29.
[7] Sarraf C, Nouri H, Ravelet F, & Bakir F. (2011). Experimental study of blade thickness effects on the global and local performances of a controlled vortex designed axial-flow fan. Experimental Thermal & Fluid Science, 35(4), 684-693.