ISSN: 2710-0588

# Performance Optimization of Small Centrifugal Fans Based on ANSYS

Junhao Yang, Shujun Tang

School of Civil Engineering, Central South University of Forestry and Technology, Changsha 410004, China

## Abstract

**Small centrifugal fans are the core components in quick-freezing refrigeration equipment. They operate through a process where an electric motor drives the impeller to rotate, converting mechanical energy into potential energy. This paper first calculates the range of blade quantities using theoretical formulas for blade number selection, then simulates the working state of small centrifugal fans using ANSYS Workbench. Finally, it compares and analyzes parameters such as pressure, flow rate, torque, and efficiency of centrifugal fans with different numbers of blades. By first calculating the range of centrifugal impeller quantities through theoretical methods and then combining it with Fluent simulation analysis, this paper obtains the optimal number of blades in a more scientific and reasonable manner.**

## Keywords

**Refrigeration equipment, small centrifugal fan, simulation, number of blades.**

## 1. Introduction

With the progress of the times and the improvement of people's living conditions, people have increasingly high requirements for food quality. Quick freezing is a method that can quickly cool food and maintain its quality for a long time [1,2]. According to statistics, the annual output of quick-frozen products in China is at least 15 million tons, with an annual growth rate of 10% to 30% [3]. Correspondingly, the production of quick-frozen equipment in China is also in a period of rapid development. Among refrigeration equipment, the strong cold air generated by the refrigeration system forms a stable horizontal annular low-temperature air flow under the action of fans and deflectors, which exchanges heat with the frozen food moving vertically, so that the frozen food can be quickly cooled and frozen, thus completing the freezing process. Among them, the fan is a key component in refrigeration equipment, and its performance will directly affect the refrigeration efficiency of the equipment.

Fans can be divided into large and small ones. Usually, a fan with a blade radius of less than 300 mm, an air volume of less than 15 $m^3$/min, and an input power of less than 750 W is called a small fan [4]. Small fans are widely used in refrigeration equipment, and they usually consist of an impeller group, a collector, and a volute. The flow field inside the fan mainly depends on the number of blades, the blade profile, and the blade thickness in the impeller. The pressure and velocity distribution in the impeller will directly affect the performance of the fan. A smaller number of blades can increase the flow area and reduce the processing accuracy, but it will weaken the impeller's ability to control the fluid, thereby affecting the pressure and efficiency of the fan; although an excessive number of blades can reduce the load on each blade, it will reduce the distance between adjacent blades, increase the probability of cavitation, and affect the fan performance [5].

In this paper, ANSYS Workbench is used to carry out numerical simulation and simulation analysis on the working state of the small centrifugal fan system in refrigeration equipment. It

ISSN: 2710-0588

explores the cross-sectional velocity, pressure, quick-freezing kinetic energy distribution under different numbers of blades, and draws relevant conclusions on the optimal number of blades.

## 2. Simulation and Modeling of Centrifugal Fans

### 2.1. Modeling of Centrifugal Fans

A small centrifugal fan is mainly composed of an impeller set, a collector, and a volute. Centrifugal fans used in refrigeration equipment on the market generally adopt a cylindrical collector. In addition to these three main components, there are also volute coamings, volute brackets, volute tongues, etc. In actual production, these structures are connected by processes such as threaded connection, welding, and riveting, which do not affect the flow field distribution inside the fan. Therefore, the final model has been simplified. Space Claim was used for modeling, and its system model is shown in Figure 1 and Table 1.

Formulas for selecting the number of blades:

$$Z = 2.1\sigma\frac{R1+R2}{R2-R1} \tag{1}$$

According to formula (1), the range of the number of blades is 51 to 80. Considering that an excessive number of blades will increase the installation accuracy requirements, impeller models with 52, 60 and 68 blades are established respectively to conduct simulation analysis on their working states.

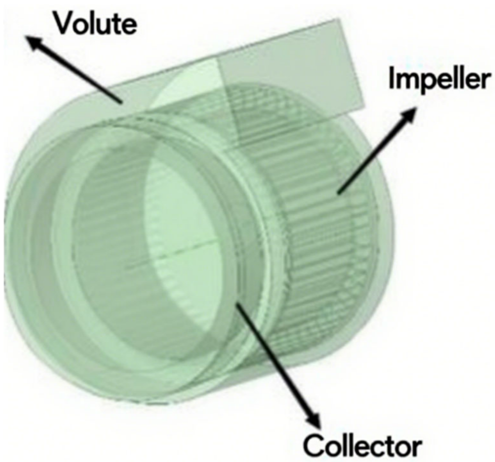


**Figure 1.** Two or more references

**Table 1.** Parameters of the centrifugal fan model

| rotational speed (rad/s) | inner diameter of the impeller (mm) | outer diameter of the impeller (mm) | height of the impeller (mm) | blade leading edge angle( °) | blade trailing edge angle ( ° ) |
|---|---|---|---|---|---|
| 80 | 105 | 126 | 140 | 60 | 120 |

### 2.2. Mesh generation

In ANSYS Workbench, the Space Claim module is used to perform Boolean operations on the model to remove the solid models in the impeller set, collector, and volute, extract the volute fluid domain (fluid1) and the impeller fluid domain (fluid2), and create the impeller set topological structure defined as the impeller area. Thus, the calculation area of the entire centrifugal volute is divided into the impeller area and the volute area. The model is then imported into Mesh for meshing, where the inlet and outlet, rotor wall, collector rotor wall, and other walls are defined, followed by regional meshing as shown in Figure 2.

For meshing the centrifugal fan, the minimum element size of the fluid domain is defined as 0.008. The impeller area is meshed with refined grids, with the minimum element size set to 0.002 mm and the growth rate at 1.2. By adopting the method of gradually refining the mesh, it is found that the quality of the computational mesh gradually stabilizes as the number of meshes increases. To ensure the accuracy of the calculation results while accelerating the solution speed, the total number of meshes is finally determined to be 3.08 million, with an average element quality of 0.831 and a minimum element quality of 0.159.

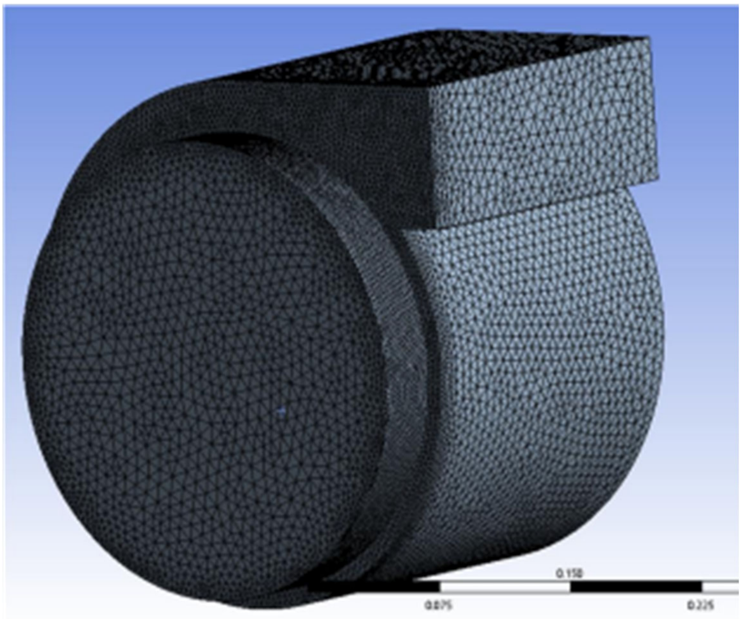

**Figure 2.** Mesh generation

## 2.3. Solution model and calculation method

In this paper, the flow simulation process of the established centrifugal fan system does not consider heat exchange. It is assumed that the flow of air in the volute is stable, and the effect of the rotating domain and the volute domain is approximately average, and only an approximate solution of the system is required. Therefore, the MRF (Multiple Reference Frame) method and SIMPLEC algorithm are selected. The discretization of the turbulent dissipation term, turbulent kinetic energy and momentum equation all adopt the second-order upwind scheme, and the RNG k-ε turbulence model with extensible avoidance function is used for the numerical simulation of the fan.

In the Fluent module of ANSYS, the iterative residual and outlet mass flow rate are monitored simultaneously. When observing the decreasing trend of the iterative residual and monitoring the outflow air mass at the fan outlet, the calculation is considered converged and terminated when the residual reaches $10^{-4}$ and the change of the outlet air mass is less than 0.1%, as shown in Figure 3 and Figure 4.

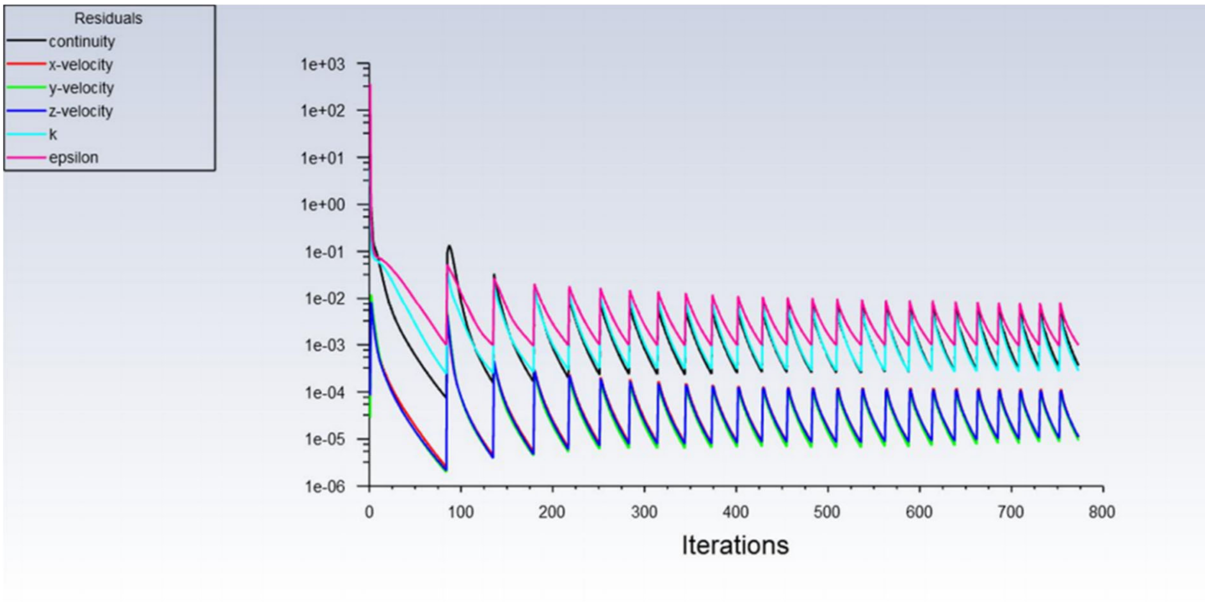


**Figure 3.** Residual convergence graph

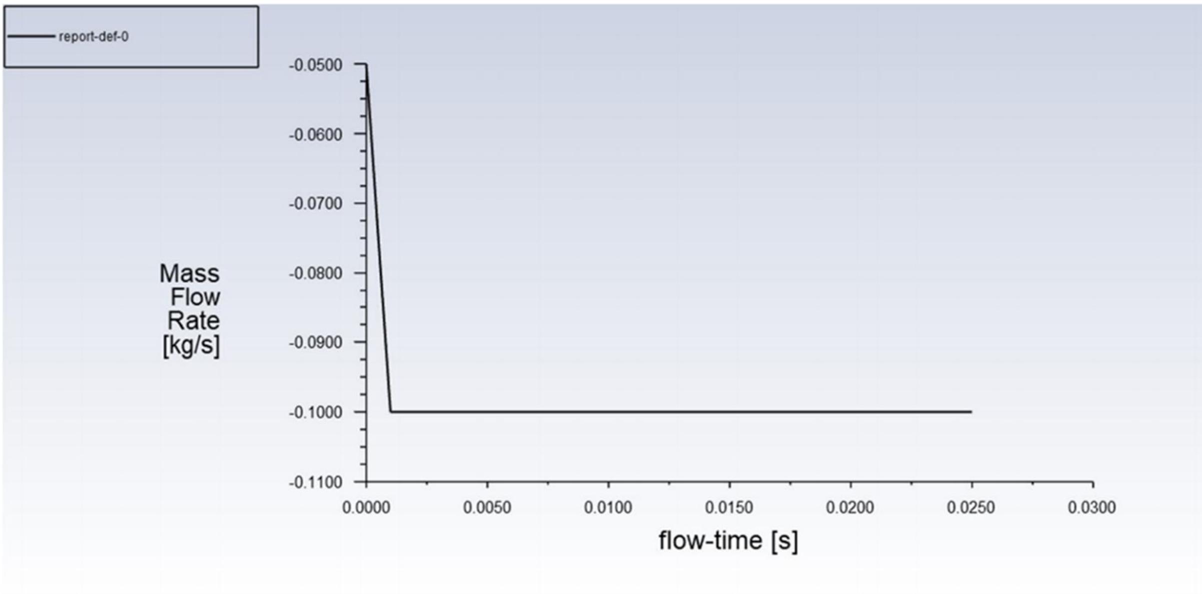


**Figure 4.** Curve of outlet mass variation

## 3. Simulation Results and Performance Analysis

### 3.1. Analysis of Simulation Results

Since the centrifugal volute system of this model adopts upper and lower air intake modes, a reference plane is established on the middle plane of the volute to obtain static pressure cloud diagrams under working conditions with different numbers of blades. From the static pressure cloud diagrams in Figure 5, it can be seen that the pressure inside the volute decreases sequentially along the impeller. This is because after the air flows through the blades, the flow velocity gradually decreases, and kinetic energy is converted into potential energy. When the air flow is continuously squeezed against the volute wall, the pressure on the volute wall keeps increasing. As the air flow moves along the wall to the volute outlet, the flow velocity starts to increase again, and the static pressure decreases.

From the static pressure distribution diagrams of different blade numbers in Figure 6, it can be seen that the static pressure generated by the impeller with Z=52 is relatively low. As the

ISSN: 2710-0588

number of blades increases, the static pressure generated by the blades also increases continuously. However, when the number of blades increases to 68, due to the reduction in the inlet volume of the flow channels between the blades, the work done by the blades on the air flow decreases, resulting in a drop in flow velocity and thus a decrease in the static pressure generated by the entire volute. The static pressure reaches the maximum value when the number of blades is around 60. It can be seen from Figure 6 that the maximum velocity of Z=60 is significantly higher than that of Z=52, and the velocity flow field distribution near the blades is more uniform. For Z=52, a large-scale backflow area appears near the volute tongue. It can be concluded that the centrifugal volute system with 60 blades can convert mechanical energy into potential energy more effectively.

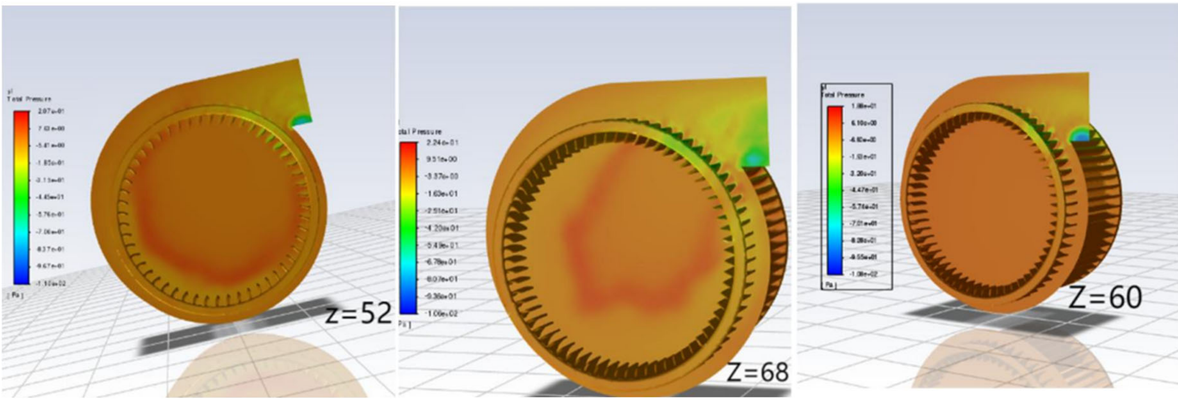

**Figure 5.** Static pressure cloud diagrams under different numbers of blades

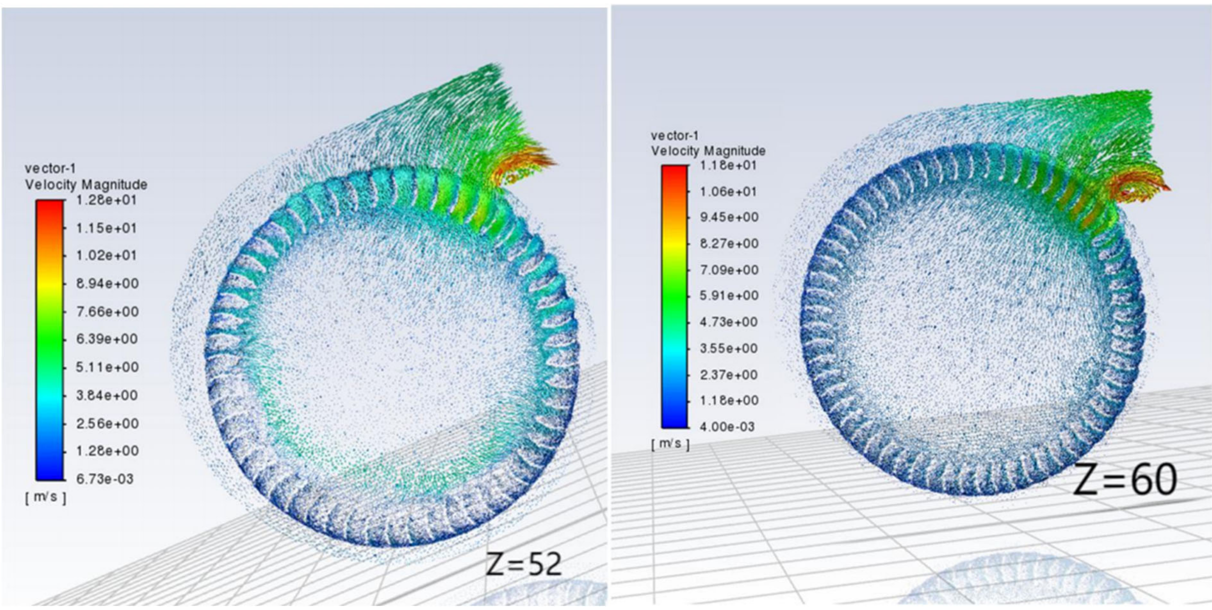

**Figure 6.** Velocity diagram

## 3.2. Performance Analysis

According to the calculation of centrifugal eddy current efficiency, the performance data and efficiency of the volute system with different numbers of blades at 80 rad/s can be obtained respectively, as shown in Table 2.

From the results in the above table, when the number of blades is less than 60, the total pressure generated inside the volute is relatively low, and a backflow area is formed near the volute tongue, which affects and leads to a small outlet flow rate. When the number of blades is more

ISSN: 2710-0588

than 60, the reduction in blade spacing results in a decrease in the volume of the flow channel, a reduction in blade torque, and a drop in flow velocity, leading to low system efficiency. When the number of blades Z=60, the torque, outlet flow rate, and efficiency reach the peak, which indicates that the performance of the centrifugal volute is at its optimal state.

**Table 2.** Values of volute flow rate, total pressure and torque for different numbers of blades

| Number of blades | Outlet flow rate($m^3$/kg) | Outlet flow rate(pa) | Torque (N×m) | η (%) |
|---|---|---|---|---|
| 52 | 0.56 | 57.6 | 1.81 | 22.2 |
| 60 | 0.66 | 72.5 | 2.33 | 25.7 |
| 68 | 0.51 | 51.1 | 1.63 | 19.9 |

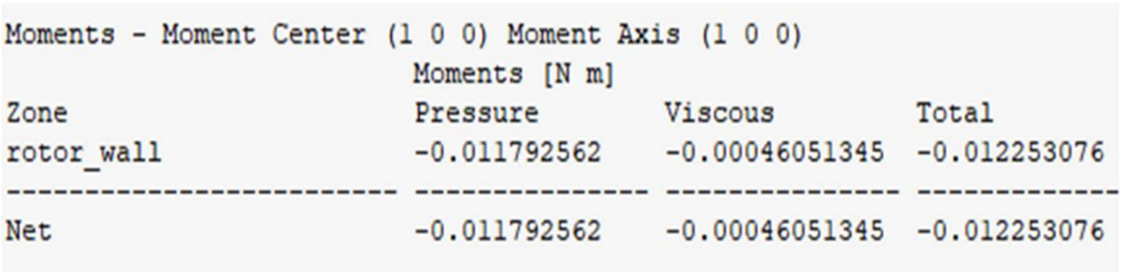


**Figure 6.** Impeller torque

## 4. Conclusion

In this paper, ANSYS is used to simulate the working state of the centrifugal fan in a certain type of refrigeration equipment. Aiming at the influence of impellers with different numbers of blades on the internal flow field of the fan, the following conclusions are drawn:

At the same rotational speed, with the increase in the number of blades, the internal total pressure and outlet flow rate of the centrifugal fan also increase, and increasing a certain number of blades enhances the ability to control the flow. However, as the number of blades continues to increase, the volume of the flow channels between the blades will decrease, reducing the ability of the impeller to do work on the air and increasing the cavitation rate. Therefore, there exists an optimal number of blades for the centrifugal impeller.

In the actual design process of a small centrifugal fan system, the reasonable range of the number of blades of the centrifugal fan can first be calculated according to theoretical formulas, and then combined with ANSYS for working simulation to obtain the optimal number of blades. This can provide great convenience for the research and development of centrifugal fans and improve the research and development efficiency of new centrifugal fans.

## References

[1] W. X. Guang, Z. Y. Wei: Overview of Research and Production of Small Fans in China, Micromotors, Vol. 1990 No.10, p.28-30.

[2] Q. J. Ye, M. Yu, Q. K. Zhang, et al.: Research Progress on Application of Quick-Freezing Technology in Food Industry, Agricultural Products Processing: Academic Journal, Vol. 2012 No.12, p.97-100.

[3] Z. H. Hu, Q. Sun, Y. Long, et al.: Research Progress on Quick-Freezing Preservation Technology for Fruits, Food and Fermentation Industries, Vol. 41 (2015) No.2, p.242-247.

ISSN: 2710-0588

[4] X. Y. Sun, L. F. Hou, J. X. Sui, et al.: Current Situation and Trend of Quick-Frozen Food Industry in China, Agricultural Machinery, Vol. 2012 No.21, p.69-72.

[5] J. Y. Zhou: Influence of Blade Number on Centrifugal Volute Performance Based on Fluent Simulation and Analysis, Household Appliances, Vol. 2022 No.201 (09), p.70-74.

[6] X. D. Cheng: Centrifugal Fans (Chemical Industry Press, Beijing, China 2018), p.138-141.