Effect of clearance jet on aerodynamic performance of centrifugal fan

G J Zhu¹, X Y Bo¹, J J Feng¹,² and G K Wu¹

¹Institute of Water Resources and Hydro-electric Engineering, Xi’an University of Technology, Xi’an 710048, China

E-mail: jianjunfeng@xaut.edu.cn

Abstract. The change of the depth of the inlet collector inserted into the impeller will change the gap shape between the collector and the impeller, and then affect the performance of the ventilator. Taking the small centrifugal ventilator as the object, the influence of the depth of the impeller inserted into the collector on the aerodynamic performance of the ventilator is studied in this paper. Firstly, the depth of the collector inserted into the front disk of the impeller is adjusted to 2 mm, 3 mm and 4 mm respectively to change the geometry of the clearance. Then, based on the computational fluid dynamics analysis method, the influence of the insertion depth of the above three different concentrators on the aerodynamic performance of the ventilator is analyzed. The analysis results show that the gap leakage is positively correlated with the depth of the impeller inserted into the collector, and the aerodynamic performance of the ventilator can be significantly improved by reducing the insertion depth of the collector under the conditions of design flow rate and large flow rate, but under the condition of small flow, the decrease of insertion depth weakens the effect of gap jet on the flow field in impeller, resulting in the decrease of fan efficiency and static pressure.

1. Introduction

Centrifugal fan is widely used in the field of ventilation and cooling in industrial production. The external rotor centrifugal fan usually includes three parts: the inlet collector, the impeller and the outer rotor motor. There is a gap between the inlet collector and the impeller to separate the dynamic and static parts. The volume loss caused by the gap is the main reason for the decrease of the efficiency of the ventilator. Therefore, it is of great significance to understand the clearance leakage mechanism of centrifugal fan and the influence of leakage on the aerodynamic performance of centrifugal fan to improve the aerodynamic performance of centrifugal fan.

The influence of the inlet clearance between the collector and the impeller on the aerodynamic performance of the centrifugal fan cannot be ignored. The internal flow of centrifugal fan is very complicated, and it is difficult to reflect the real characteristics of the internal flow of the centrifugal fan only for the numerical calculation of one blade passage or section of the centrifugal fan [1]. The air flow parameters near the front plate of the fan impeller vary greatly, and are affected by the vortex area on the back of the collector. It is easy to form a large vortex area, which varies with the relative axial position between the collector and the impeller and the flow rate of the fan, which constitutes a main source of fan loss and noise. Kind and Tobin [2] tested the flow field of the squirrel cage fan with a five-hole probe, the results show that the impeller flow state under low flow conditions is different from that of the impeller under high efficiency conditions. Fukutomi et al [3] and Wang et al [4] designed the inlet nozzle of a fan, and improved the aerodynamic performance of fan by affecting...
the effective flow area of impeller and weakening the flow vortex in blade channel at a specific position.

In centrifugal turbomachines, the leakage of impeller inlet clearance is one of the main reasons for its efficiency reduction. Dong et al [5] investigated the effect of gap leakage jet on the flow field structure of centrifugal ventilator under different relative length of inlet clearance. Bunker [6] has found that the leakage flow of the top clearance of the turbine blade is one of the important factors leading to the increase of the flow loss. In addition, other scholars have studied the effect of ring clearance on leakage loss and mechanical efficiency of centrifugal pump [7-9]. CFD has become a main method to calculate the internal flow field of turbomachine and analyze its internal flow mechanism [10].

The pressure difference flow caused by the pressure difference at both ends of the gap is the main cause of the formation of the gap jet. Rain [11] uses the leakage model to analyze the pressure difference flow in the compressor, and accurately forecasts the gap loss of the compressor. Baskharone et al [12] used finite element analysis method to simulate the leakage jet flow in the internal clearance of multistage pump, and compared it with the existing impeller leakage analysis model. The fan clearance has an influence on the adjacent flow channel. When the radial clearance is large enough at any flow rate, the influence can be extended to the whole flow channel area [13]. Therefore, it is very important to find a suitable inlet clearance to control the flow field at the inlet of the ventilator.

The depth change of the current collector into the front disc of the impeller will affect the geometry and size of the gap between the current collector and the impeller, thus affecting the performance of the ventilator. In order to explore the influence of the depth change of the collector inserted into the impeller front disk on the internal flow characteristics and energy external characteristics of the centrifugal fan. In this paper, different schemes by adjusting the depth of the collector inserted into the impeller front disk have been obtained, and then the aerodynamic external characteristics and internal flow field characteristics of the fan have been compared and analyzed under different schemes based on the computational fluid dynamics method. Finally, according to the analysis results, the variable influence of the insertion depth of the collector on the fan performance is revealed, which provides a reference for the relative position design between the collector and the centrifugal fan impeller in the future.

2. Calculation method

2.1. Geometry and grid generation

![Figure 1](image_url)

**Figure 1.** The depth of the collector inserted into the front disk of the impeller is $h$.

Taking a small centrifugal fan as the research object, the main parameters of the fan are as follows: the design speed is $n = 3300$ rpm, the design flow rate is $Q_v = 364$ m$^3$/s, the inner diameter of the impeller is $D_1 = 134.6$ mm, and the outer diameter of the impeller is $D_2 = 196$ mm, blade number $z = 6$. The 3D
modeling software centrifugal fan is used to model the fan. Figure 1 shows the profile of fan impeller, and the insertion depth $h$ is 2 mm, 3 mm and 4 mm, respectively. The model is divided into three parts: the inlet section, the impeller section and the exit section.

The whole fluid region is divided into a rotating impeller area and two static regions, and the fan impeller is inside the wind tunnel, as shown in figure 2. In order to fully simulate the flow field in the impeller, the inlet region of the model is hemispherical, and the outlet section is the modeling of the wind tunnel.

![Figure 2. Three-dimensional modeling of the ventilator.](image)

The full flow channel of the ventilator is divided by hexahedral structured grid, as shown in figure 3, the total grid number is verified by the static pressure and efficiency of the ventilator. When the total number of fan model grid is 5.29 million, the number of grid has no effect on the calculation results.

![Figure 3. Grid division of ventilator. (a) Grid view and (b) impeller grid.](image)

The aerodynamic performance of the prototype fan was tested by the large wind tunnel rear side method. The aerodynamic performance was tested on the test platform with reference to the international standard AMCA-210-2007 [14]. Figure 4 shows the sketches and photos of the experimental equipment.
Figure 4. Aerodynamic performance experimental device. (a) Fan experimental wind tunnel and (b) Inner impeller diagram of wind tunnel.

The static pressure $P_s$ and static pressure efficiency $\eta$ at the outlet of the test fan can be calculated according to the test data according to the following formula:

$$P_s = P_t - \frac{1}{2} \rho \left( \frac{Q_v}{A} \right)^2$$

(1)

$$\eta = \frac{Q_v P_s}{W} \times 100\%$$

(2)

Where $Q_v$ represents the volume flow rate of the ventilator, $A$ indicates the area of the outlet of the wind tunnel, and $\rho$ is the density of the air, $P_t$, $W$ is the total pressure and total power consumed by the test fan.

2.2. Boundary condition settings

In the numerical calculation, the inlet boundary condition of the calculation domain is set to the static pressure boundary condition and total pressure is 0, the outlet boundary is given the mass flow rate, and the solid wall is set to the non-slip wall. In the steady calculation, the interface between the rotating part and the static part is connected by the frozen rotor method. In the process of solving the N-S equation, the convective term in the governing equation is discretized by the second order difference scheme, and the SST $k-\omega$ turbulence model is used for numerical simulation. In order to verify the reliability of the numerical calculation, five working conditions are selected for steady calculation in and around the rated working condition of the unit, and the working conditions are shown in table 1.

Table 1. Working conditions and parameters of centrifugal ventilator.

| Working condition number | Airflow rate (/m$^3$/h$^{-1}$) |
|--------------------------|---------------------------------|
| Condition 1              | 150                             |
| Condition 2              | 313                             |
| Condition 3              | 364                             |
| Condition 4              | 500                             |
| Condition 5              | 626                             |

The experimental results of static pressure efficiency for the clearance depth of the prototype fan of 3 mm are compared with the results of numerical simulation, as shown in figure 5 below. It can be
seen that the results of numerical simulation are the same as the experimental results, and the agreement between them is very good, which indicates that the results of numerical simulation are reliable. The efficiency obtained by numerical simulation is slightly larger than the experimental value, which is due to the fact that the surface roughness of fan impeller is not taken into account in the process of numerical simulation, so the numerical efficiency is slightly higher than the actual efficiency.

Figure 5. Comparison between numerical simulation results and experimental values.

3. Results and discussion

According to the different insertion depth of the collector, five calculation points are calculated with the help of Ansys software CFX. Through the treatment and analysis of the numerical results, the external characteristic curve and the internal flow field structure of the centrifugal fan are obtained.

3.1. Analysis of external characteristic curves

The static pressure and efficiency of centrifugal fan are selected as the external characteristic parameters of centrifugal fan. Figure 6 shows the static pressure and efficiency curves of centrifugal fan at different depth of collector insertion. It can be seen that with the increase of the depth of the front disk of the impeller inserted into the collector, the static pressure shows a decreasing trend. In the small flow area, with the increase of the gap depth, the static pressure changes little, but the static pressure decreases obviously in the large flow area. As far as the efficiency curve is concerned, the efficiency change of different inlet clearance fans is basically the same, showing the trend of
increasing at first and then decreasing. When the clearance depth increases from 2 mm to 3 mm, the fan efficiency decreases by 2% under rated working conditions, and the fan efficiency decreases more obviously in large flow area. To sum up, when the gap insertion depth is 2 mm, the fan has obvious advantages near rated working conditions and in large flow area.

Figure 7 shows the variation of gap leakage in different sizes. With the increase of the depth of the front disk of the impeller inserted into the collector, the leakage of the gap increases, which is caused by the increase of the cross section area of the gap. With the increase of inlet flow rate, the total pressure obtained by the gas decreases gradually, and the pressure difference between the two sides of the clearance becomes smaller with the decrease of the total pressure of the fan, so the leakage loss will also decrease, and the trend is more obvious with the increase of the insertion depth. This shows that when the insertion depth of the front disk of the impeller is 2 mm, the leakage of the fan is reduced, the internal flow field of the fan is improved, and the efficiency of the fan is also improved.

![Figure 7. Leakage quantity of gap of different dimensions.](image)

### 3.2. Internal flow analysis

The axial cross-section of the flow field is selected, as shown in figure 8, there is a significant swirl flow in the space area between the air inlet, the casing and the front disc of the impeller, and the flow from the impeller outlet to the gap between the impeller and the air inlet experiences a process of generation, development and dissipation. The change of the insertion depth of the front disc of the impeller is bound to influence the flow field dynamics in the fan, which in turn leads to the change of the air performance of the fan.

![Figure 8. Axial profile of fan flow field.](image)

The flow rates of 150 m$^3$/h, 313 m$^3$/h and 626 m$^3$/h are selected as the small flow, design flow and large flow of centrifugal ventilator for detailed analysis. Figure 9 shows the three-dimensional flow distribution diagram at 3 different gaps under 3 working conditions. Under the condition of small flow...
rate, the internal flow pattern of the blade flow channel is disordered, and the effect of the gap jet on the internal flow pattern is not obvious. With the increase of the inlet flow rate of the fan, the fluid distribution in the single flow channel is more uniform, but the flow pattern in the fan impeller with different clearance is not much different.

![Figure 9. Three-dimensional streamline distribution map at different clearance of fan.](image)

Figure 10 shows the pressure distribution of blade airfoil under different working conditions. In order to investigate the effect of gap jet on the pressure distribution of the blade, the blade position near the gap, that is, the blade airfoil surface at 0.98 times the blade height, is selected for analysis. Where the transverse coordinate \( z/c \) denotes the relative chord length of the airfoil, \( z/c = 0, 1 \) represents the head of the airfoil, the tail position, and the longitudinal coordinate is the static pressure coefficient \( C_p \), which is defined as the static pressure coefficient.

\[
C_p = \frac{P_s}{\frac{1}{2} \rho v^2}
\]

Where \( P_s \) is static pressure, \( \rho \) is density, \( v \) is the circumferential velocity of the outer edge of impeller.

![Figure 10. Pressure distribution of airfoil on the front disk surface of fan impeller. (a) \( Q_v = 150 \) m\(^3\)/h, (b) \( Q_v = 362 \) m\(^3\)/h and (c) \( Q_v = 626 \) m\(^3\)/h.](image)
It can be seen from the figure that with the change of the size of the gap, the pressure change trends of the two ends of the wind wing type are basically the same. The pressure difference between the blades near the hub is the largest, along the chord length direction of the airfoil, the pressure difference between the two ends of the blade is first reduced and then increased. The pressure difference of the two ends of the airfoil at the strength of 0.2 times is the smallest, and the pressure distribution of the airfoil surface of the blade is found to have no significant difference from other cross-sectional analysis. The inlet flow rate of the fan has a great influence on the pressure distribution on the airfoil surface, especially under the condition of large flow rate, the pressure surface pressure of the blade near 0.2 times chord length is much smaller than that near the suction surface, which leads to the uneven force on the blade and the serious decrease of the fan efficiency. Therefore, the fan should avoid running under the condition of large flow rate.

![Figure 11. Distribution of kinetic energy.](image)

Figure 11 shows the kinetic energy distribution of three clearance impellers at 73% of height of the fan under the condition of small flow. It can be seen from the diagram that when the clearance depth of the impeller is 2 mm, the internal flow is unstable, the turbulent kinetic energy at the inlet of the impeller and the outlet of the impeller is slightly larger. With the increase of the insertion depth of the collector, the influence of the gap jet on the inner flow channel of the impeller is enhanced, which makes the flow in part of the impeller smooth and the turbulent kinetic energy in the impeller decreases.

4. Conclusions

- In the small flow area, with the increase of the gap, the static pressure changes slightly. However under the design flow and large flow conditions, the pneumatic performance of the ventilator can be significantly improved by the decrease of the insertion depth of the collector, and the operation advantage of the fan with the clearance depth of 2 mm is obvious.
- With the increase of gap depth, the amount of gap leakage increases. Under the condition of small flow rate, the clearance jet can improve the flow field in the impeller obviously. When the clearance depth is 3 mm, the turbulent kinetic energy in the impeller decreases obviously, and the static pressure and efficiency of the fan increase.
- The inlet flow rate of the fan has a great influence on the surface pressure distribution of the airfoil, and the pressure difference between the two sides of the airfoil is the smallest at 20% chord length. Under the condition of large flow rate, the pressure surface pressure of the blade near 20% chord length is much smaller than that of the suction surface, which leads to the uneven force of the blade and the serious decrease of fan efficiency.

Acknowledgments

This work was partially supported by Natural Science Basic Research Plan in Shaanxi Province of China (Program Nos. 2019JQ-044 and 2018JM5102) and Scientific Research Plan Projects of Shaanxi Education Department (Program No. 19JK0570), and also be funded by the National Natural Science
References

[1] Li X H, He H W et al 2002 Numerical analysis of whole steady flow field for a centrifugal fan J. Eng. Therm. 23 453-6

[2] Kind R J and Tobin M G 1990 Flow in a centrifugal fan of the squirrel-cage type J. Turbomach. 112 84-90

[3] Fukutomi J, Kuwauchi T, Itabashi A et al 1999 A study of sirocco fan with eccentric inlet nozzle Trans. JSME: Ser. B 65 2023-9

[4] Wang J B, Ou Y D and Wu K Q 2005 Analysis on the vortices flow in the blade passages of the fan for air-conditioner and the influence of the eccentric fan inlet J. Eng. Thermophys 26 951-3

[5] Dong Y S, Yang A L, Chen E Y et al 2016 Influence of clearance leakage jet flow in wear-rings on flow characteristics of a centrifugal fan Chin. J. Power Eng. 36 286-93

[6] Bunker R S 2006 Axial turbine blade tips: Function, design, and durability J. Propul. Power. 22 271-85

[7] Tam L T, Przekwas A J, Muszynska A et al 1987 Numerical and analytical study of fluid dynamic forces in seals and bearings J. Vib. Acoust. 110 112-9

[8] Childs D W 1991 Centrifugal-acceleration modes for incompressible fluid in the leakage annulus between a shrouded pump impeller and its housing J. Vib. Acoust 113 209-18

[9] Wang Y and Zhang X 2008 Effects of clearance of impeller wear rings on efficiency of low specific speed centrifugal pump J. Drain. Irrig. Mach. Eng. 26 27-30

[10] Coutierdelghosa O, Régiane F P, Reboud J L et al 2003 Experimental and numerical studies of a centrifugal pump in cavitating conditions J. Fluid Eng. 125 492-7

[11] Rains D A 1954 Tip clearance flows in axial compressors and pumps (Pasadena: California Institute of Technology)

[12] Baskharones E A and Wyman 1999 Primary/leakage flow interaction in a pump stage J. Fluid Eng 121 133-8

[13] Xi G and Wang S J 1997 Numerical analysis of the three-dimensional turbulence flow in an unshrouded impeller of centrifugal compressors J. Xi’an Jiaotong Univ. 1997 93,98

[14] AMCA/ASHRAE 2007 ANSI/AMCA standard 210-07/ANSI/ASHRAE standard 51-07, laboratory methods of testing fans for certified aerodynamic performance rating (Arlington Heights, IL: Air Movement and Control Association and Atlanta: American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc.)