ORIGINAL RESEARCH ARTICLE

Hydrodynamic Analysis of Gas Flow in Centrifugal Ventilator

Yue Yang,Changnian Zhu,Jiafeng Wan,Bingkun Dong

Institute of Vehicle and Power Engineering, Jiuquan University of Technology, Gansu, China

ABSTRACT

The present study conducted a detailed analytical research from the aspect of hydrodynamics, on the air flow of an exhaust fan model selection influenced by model selection. The effects of friction, local resistance, duct diameter and radius of curvature of elbow on air flow and pressure of a pipeline system are discussed. The characteristics of centrifugal fan and pipe network of exhaust system, and the relationship between actual resistance and rated flow/ pressure are also discussed. Using FLUENT, acomputational fluid dynamics software, the 3D gas flow inside 4-73 №10D centrifugal ventilator was numerically simulated and analyzed, and the pressure and velocity distribution of each part are analyzed emphatically.

KEYWORDS: Pipe network characteristics; centrifugal fan; three-dimensional numerical simulation; pressure field; flow field

1. Introduction

In view of the difficulty in the measurement of the flow field of the ventilator, the numerical simulation is therefore an important method to study the mechanical flow field of the impeller. With the rapid advancement in computational fluid dynamics, the internal flow field research of fluid machinery has made a big leap, from two-dimensional, quasi-three-dimensional flow to full three-dimensional flow. Guo and Kim analyzed the flow of the forward centrifugal fan with a steady and unsteady three-dimensional RANS method. Carolus and Stremel analyzed the turbulence at the inlet of the fan by CFX to obtain the relationship between pressure and noise. Meakhail et al. used PIV test method and CFX simulation method to analyze the impeller area. However, many researchers choose a flow or unit as a research object, thus overlooking the non-symmetry of the volute caused by the flow of non-axis symmetry, or the actual fan model cannot get a real internal flow field. In this paper, the commercial software FLUENT6.3, the 4-73 № 10D centrifugal fan in the design conditions under constant three-dimensional flow numerical simulation, to capture the internal flow phenomenon, reveal the actual situation of the fan flow for the fan to further improve the expansion of the operation Provide a theoretical basis.

2. Hydrodynamics Analysis of 2 Ventilation System

2.1. Influence of Friction Resistance on Air Volume and Wind Pressure

Air flow along the ventilation pipe will produce two types of resistance: First type, friction resistance caused by the friction between air and pipe wall (also known as frictional drag); Second type, local resistance caused when there is a change of direction and speed or the presence of eddy current as the air flows through some parts of the duct (such as elbow, Three-way, suction hood and butterfly valve) . The frictional resistance of the unit length of the duct is calculated as follows:

$$P_{m} = \frac{\lambda}{D}\frac{\rho}{2}$$

Where: P mr - frictional resistance per unit length of round duct, Pa / m;

Λ - frictional resistance coefficient;

N - average velocity of air in ducts, m / s;

P - air density, kg / m3;

D – Internal diameter of circular duct, m.

In the calculation of these two types of resistance, usually in accordance with the laminar flow state to take the friction coefficient λ, then, along the pressure loss and the air flow rate is proportional to the first time, when the flow rate increases beyond the critical flow rate Re = 2 300 , The air flow inside the duct becomes turbulent, and the pressure loss along the pipe is proportional to the air flow rate of 1.75 to 2.0 times, that is, the resistance increases by nearly 1 times.

Usually the duct wall as a hydraulic smooth tube, that is, the absolute wall roughness K = 0.1 mm to calculate, and in fact, after a period of time, the fan impeller, duct, elbow, umbrella exhaust hood, The inner wall of the duct has become a hydraulic rough pipe (or resistance square area), the absolute roughness of the wall value of the value of K ≈ 0.9 ~ 3.0 mm; this is the wall of the air pipe, , The actual frictional resistance P 'mr per unit length should be calculated by multiplying the frictional resistance per unit length by the correction coefficient β,

P 'mr = β × P mr

Assuming that the air flow rate in the duct is 10 m / s and the absolute roughness value is K = 0.1 mm,

Correction coefficient β = (Kν) 0.25 = (0.1 × 10) 0.25 = 1 (1)

Where: K - duct internal wall absolute roughness, mm;

N - air flow rate inside the duct, m / s.

Assuming that the air flow rate in the duct is constant, still 10 m / s, but the absolute roughness value K = 1 mm, then:

Correction coefficient β = (Kν) 0.25 = (1 × 10) 0.25 = 1.78 (2) In other words, when the unit length duct friction resistance is 1.78 times the original.

Assuming that the air flow rate in the duct is 10 m / s and the absolute roughness value is K = 2 mm, then:

Correction coefficient β = (Kν) 0.25 = (2 × 10) 0.25 = 2.114 (3) At this time, the frictional resistance in the unit length duct is 2.114 times the original.

Another important factor is that many manufacturers either do not add paint mist coagulant/add in accordance to the required amount or time, or do not remove the paint residues on time when using a water curtain spray booth. The paint residues in the circulating water retain on the baffle, the watertight plate, the scroll plate, the inner wall of the duct, resulting in the increase of the inner wall roughness, which leads to the increase of the friction resistance. This is one of the reasons why after a period of time, the air pressure drops, the amount of ventilation decreases, and the paint mist spills. In our opinion, the air pressure is not recommended to be set at 10% to 20% of the margin, but the best to increase it to about 80% of the margin; to regularly clean fan impeller, volute, duct, baffle and other ventilation system Within the paint mud, and this is a lot of manufacturers do not pay attention to, should be used for the operation and maintenance of the operator training.

2.2. The impact of local resistance

The flow of fluid in the duct, through the elbow, valves, reducer, etc., the direction and the size of the area changes, there may be eddy current loss or collision losses, these are called local resistance.

The local resistance of the duct components can be calculated as follows:

$$\Delta P = \xi \frac{\nu^2 \rho}{2}$$

Where: Δ P - local resistance of duct components, Pa;

Ξ - local resistance coefficient;

N - average velocity of air in ducts, m / s;

P - air density, kg / m 3.

In the general ventilation system, due to the different parts of the air duct, the local resistance coefficient is difficult to calculate, usually through the test, and then look up the table to determine. The actual duct system due to the diameter, flow rate, medium, radius of curvature, the angle of expansion and other sizes, once there is a parameter changes, the pipeline system, the actual local resistance is also changing. For example, the baffle soda separator in the use of a period of time, the surface will be stained with lacquer, making the local resistance increases, the air flow rate decreased; cross-sectional area becomes smaller, but also make the air flow rate, the air containing mist particles, The air density increases, the local resistance coefficient will become larger, in this situation, the gas will produce whirlpool, air flow into a turbulent state, these factors will lead to the local resistance of the baffle increases, the amount of ventilation decreased.

The local resistance coefficient ξ is for the average flow rate of a certain cross-section, but the local resistance loss of the various fittings does not occur in a section of the flow, but in a length of flow, if 2 The parts are too close, then they will interact with each other, then the flow of the situation is complicated, and cannot use the manual given ξ to calculate. Because the value of the ξ in the manual is measured without any other influence. For example, in order to reduce the height of the spray booth, at the top of the spray booth, it is often an umbrella suction hood and butterfly valve, elbow and fan suction port directly connected, there is little straight pipe transition, The partial resistance of the first class is not as simple as the resistance of several parts. The resistance coefficient ξ will change, the pipeline will produce whirlpool, the mainstream by compression or diffusion, the flow rate distribution will be quickly reorganized, viscous resistance and inertia resistance will be significantly increased.

2.3. The effect of duct diameter on wind speed

A flow rate of 6 ~ 14 m / s is optimal, preferably not more than 10m / s. In order to cut down material costs, some manufacturers tend reduce the diameter of the duct to increase the wind speed up to 24 m / s, making a drastic increase in air resistance, when the shaft power is certain, the amount of ventilation will decline, resulting in paint mist cannot be taken out. For example, a company for the production of a factory in the field of two spray booth, the ventilation effect has been poor, severe mist spray, 2 times after the replacement of the fan, still cannot solve the problem, the author found to the scene, the fan duct design is too small, The ductile frictional resistance and local resistance are suddenly increased, resulting in a serious decline in the amount of ventilation, the results only replace the large diameter of the duct to completely solve the problem.

Also, the fan outlet to the duct outlet length, the general should be out of the exhaust pipe outside the roof of more than 2 m height to take advantage of atmospheric pressure. The current common problem is that some designers often only consider the design of the fan into the wind section of the resistance of the problem, regardless of the fan out of the wind section of the wind resistance, not to consider the height of the outdoor exhaust pipe, which is defective.

The influence of the curvature radius of the elbow on the local resistance coefficient

90 ° of the air duct elbow its local resistance coefficient ξ and duct bending radius of curvature and duct diameter R / D inversely proportional to the greater the R / D, ξ value is smaller; such as: R / D is 1 Ξ is 0.23, when R / D is 2, ξ is 0.15, R / D is 2.5, ξ is 0.13, and when R / D is greater than 2.5, the effect is less obvious. Generally should be used R / D 2.0 ~ 2.5, so that the local resistance coefficient ξ can be smaller. It should be noted that the wind pipe elbow referred to here refers to the smooth round duct, in the manufacture, are generally divided into five sections of the production, lofting, rolling round, and then biting or welding into a whole (commonly known as shrimp curved) , And such a 90 ° shrimp elbow, the resistance coefficient than the smooth garden duct elbow, such as: R / D is 1, the shrimp curvature ξ value of 0.33, R / D is 2, Ξ is 0.19, which is where designers usually overlook. More manufacturers to reduce the cost, the use of R / D is 1, it is not desirable. These places accumulated, the pressure loss of the pipe network system is big. The local resistance coefficient ξ of the air duct elbow is also proportional to the bending angle. For example, the greater the bending angle, the greater the resistance coefficient, the general should use 45 °, 60 ° and 90 ° elbow.

3. Analysis of Pipe Network Characteristics and Operating Point of 3 Ventilation System

3.1. Centrifugal fan characteristics

Centrifugal fans even in the same speed, it will transport the air volume may vary. The pressure loss of the system is small, the required wind pressure ΔP = ξ2ν 2 ρ is small, then the air volume is large; the other hand, the system pressure loss is large, the required wind pressure is large, the air volume is small The

The characteristic curve of the fan is shown in Fig. It can be seen that the fan can work in a variety of different air volume. In the ventilation system, the fan will work according to its characteristic curve at a point, at this point, the fan air volume and the system pressure to balance, which also determines the fan air volume. But it is the fan of this automatic balance of performance, resulting in sometimes the actual situation, the fan air volume and wind pressure cannot meet the design requirements.

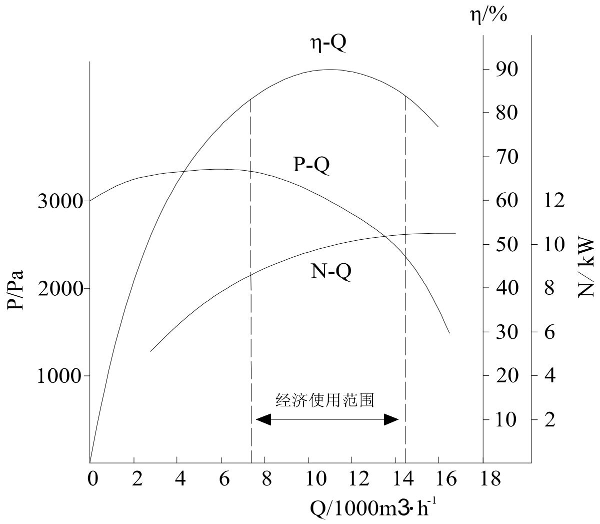
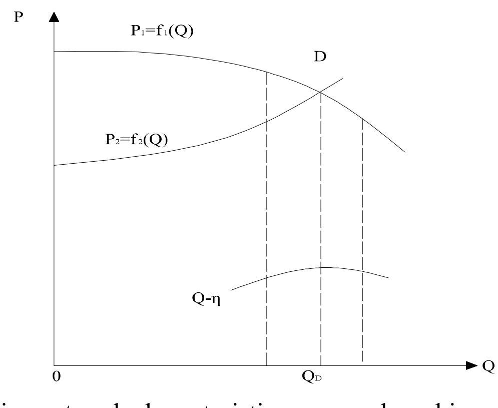


Figure 1 fan characteristics curve

3.2. Characteristics of Pipe network in ventilation system

When the fan is working in the ventilation pipeline system, its air volume, wind pressure and other parameters depend not only on the performance of the fan itself, but also on the characteristics of the whole pipe network system (pipe network characteristic curve and working point see Figure 2). The total resistance of the piping system consists of the sum of the various pressure losses in the system, the pressure of the suction gas and the pressure of the exhaust gas (when the gas is drawn from the atmosphere and the atmosphere is discharged, the pressure difference is equal to 0) The dynamic pressure at the time of discharge is three parts, that is, the P 2 = f 2 (Q) curve shown in the figure. In more cases, the pipe characteristic curve depends only on the total resistance of the piping system and the dynamic pressure at the time of pipe discharge, and both are proportional to the square of the flow Q; the pipe characteristic curve P 2 = f 2 (Q) And the fan performance curve P1 = f 1 (Q) of the intersection point D is the operating point of the fan. When the actual resistance in the pipe network is greater than the rated wind pressure of the fan, the air flow will be reduced. On the other hand, when the actual resistance in the pipe network is less than the rated wind pressure of the fan, the air flow will increase (the relationship between the characteristic curve and the fan performance is shown in Fig. 3).



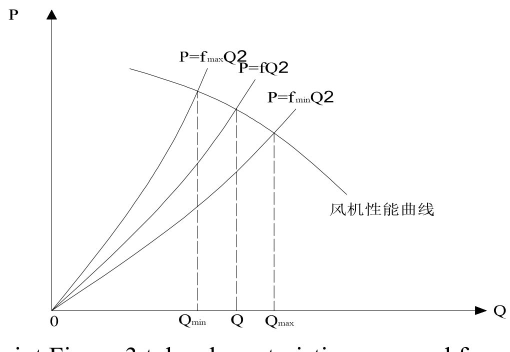


Figure 2 pipe network characteristic curve and working point Figure 3 tube characteristic curve and fan performance curve

As mentioned above, after a period of time the spray booth is used, due to the resistance of the pipe network system gradually become larger, the fan gradually unable to overcome the system pressure loss, resulting in gradual reduction in the amount of exhausted air. As a result, the paint mist spray organic solvent cannot be removed and spill out into the workshop; the same time, spray paint the surface of the workpiece near the air filled with particle size ranging from paint mist particles, a lot of sticky on the surface of the workpiece, affecting the surface spray quality.

It is also pointed out that the local resistance calculation in the general duct system is based on an ideal pipe network structure and a static model, but in fact a variety of structural design itself and the dynamic changes in the use of the process. Making the calculated local resistance and the actual use of the wind resistance is very different, which is now some of the problems of the painting room.

4. Flow field control equation establishment

In the ventilator, the flow velocity is low and can be regarded as incompressible flow. In the impeller with constant angular velocity rotation, the relative motion in the impeller can be considered to be constant when the relative motion is used to describe the relative motion with the non-inertial coordinate system that rotates with the impeller. So the impeller is incompressible, homogeneous, the density of the continuity of the continuity equation and the equation of motion

Where W - relative velocity;

P - pressure

F - mass force;

M - viscosity coefficient;

R - radius;

-2ω × W - Coriolis force;

- ω × (ω × R) - centrifugal force.

Where C1, C2, σ K, σ ε, C μ - empirical constants;

U i, U j - i, j direction of the speed;

X i, X j - i, j direction of the node coordinates;

P - fluid density;

P - pressure;

F i - body force;

H, η t - laminar and turbulent viscosity coefficients;

K - turbulent kinetic energy;

E - turbulent kinetic energy dissipation rate.

5. Calculate object and boundary conditions

5.1. Fan model parameters

Analysis of the object for the 4 -73 № 10D centrifugal fan, by the intake chamber, the collector, impeller and volute composition. In the Pro / E to establish the model, in order to solve the problem of convenience, in the machine assembly so that the absolute coordinates and relative coordinates in the same position, the origin is located on the outer wall of the rear wheel center, X axis negative direction for the volute exit, Y Axis negative direction for the volute inlet direction, Z axis positive direction for the inlet chamber inlet direction. The blade inlet width is 250mm, the inlet width of the blade is 250mm, the suction chamber of the inlet chamber is 1300mm × 600mm, the volute width is 650mm, the width of the volute is 650mm, The outlet is 900mm × 650mm, leaves 12, turn 1200r / min.

5.2. Meshing

In GAMBIT the flow path area is shown in Fig. As the fan structure is more complex, take tetrahedron and hexahedral mesh combined with the way, the grid total of 676045. Impeller flow area using rotary reference system MRF coordinate method; leaf, front and rear plate with relatively static reference system; intake chamber, current collector and volute using absolute static reference system.

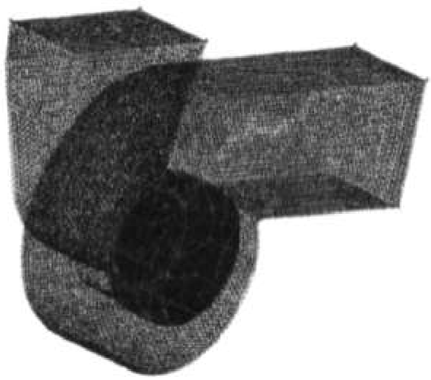


Figure 4 4-73 № 10D Fan overall grid

5.3. Calculation methods and assumptions

(1) Assume that the flow is stable, sticky, incompressible; flow process to ignore the role of mass force;

(2) There is a gap between the impeller inlet and the current collector, but in the calculation of 0, to avoid the gap area pressure gradient is too large;

(3) The discrete equation is used to solve the SIMPLE algorithm with pressure velocity. The turbulence model uses the standard k-ε equation and uses the standard wall function method.

5.4. Boundary conditions

Import: calculated according to the volume flow, the use of uniform imports, the speed of 12.6m / s. Outlet: set pressure outlet static pressure for atmospheric pressure, air density of 1. 2kg / m 3.

6. Results analysis

6.1. Static pressure analysis

As can be seen from Fig. 5, the static pressure gradually changes from the inlet to the outlet, reaching the maximum in the outer wall of the volute, and the static pressure decreases due to the flow loss at the outlet, which is consistent with the conclusion [6]. It can be seen from Figure 5a, due to the non-axis symmetry of the volute, the volute lower static pressure and the impeller center is not on the same axis; Figure 5b shows that the inlet chamber and the volute, because of this The shape of the two changes, resulting in lower static pressure.

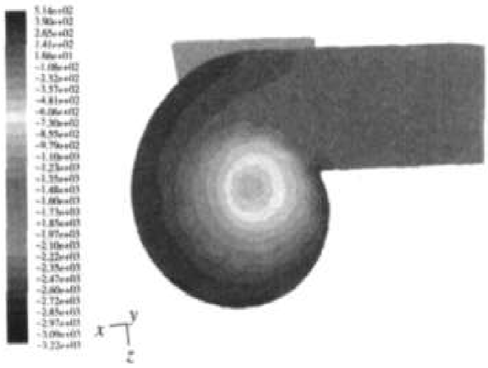


Figure 5 (a) Overall spiral case wall surface static pressure distribution

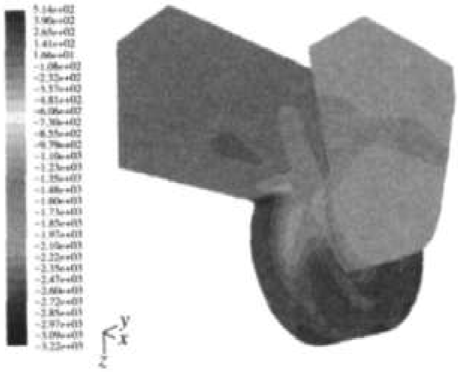


Figure 5 (b) the whole volute, the intake chamber before the static pressure distribution

6.2. Y-axis static pressure analysis

Y = 150mm as shown in Figure 6b; Y = 250mm as shown in Figure 6c; Y = 150mm as shown in Figure 6b; Y = 250mm as shown in Figure 6c; Y = 350mm as shown in Figure 6d. It can be seen from the four graphs that the impeller pressure distribution is not symmetrical due to the axisymmetry of the impeller. The spiral spiral spiral case is non-axisymmetric and the static pressure at the inlet of the impeller is the lowest. The static pressure center in the impeller tends to expand the volute, and the static pressure gradually increases in the outer part of the volute. Due to the presence of flow loss, static pressure gradually decreases along the volute exit.

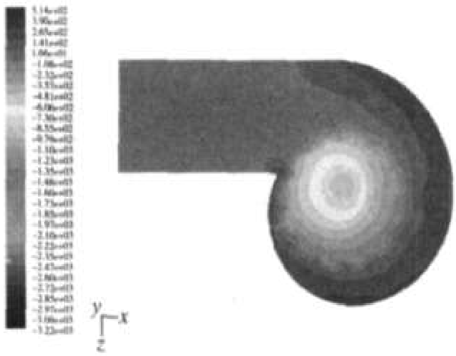


Figure 6 (a) Y = -20mm rear panel and volute outer wall gap between the static pressure

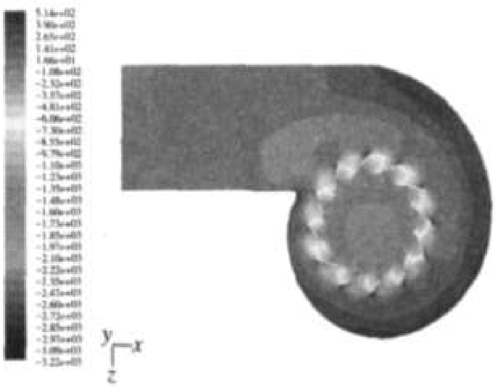


Figure 6 (b) Y = 150mm impeller axial intermediate surface static pressure

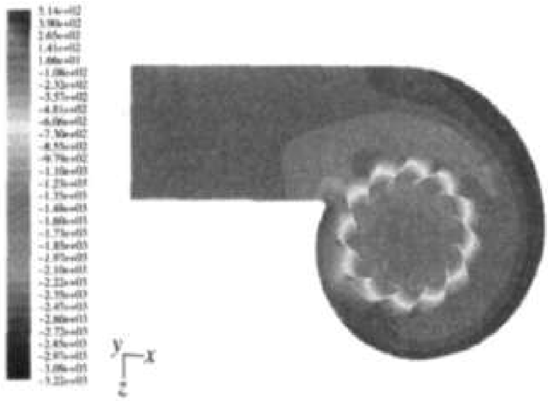


Figure 6 (c) Y = 250mm impeller outlet contact with the front panel axial surface static pressure

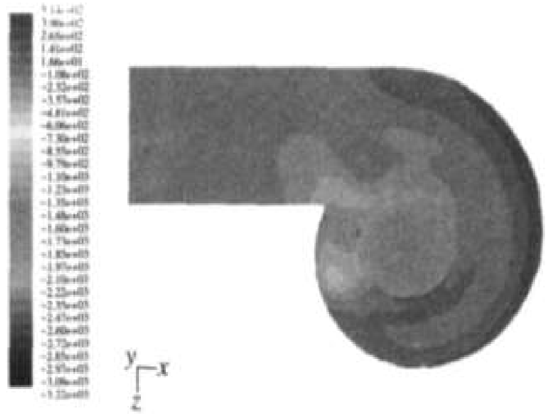


Figure 6 (d) Y = 350mm impeller inlet contact with the front panel axial surface static pressure

6.3. Impeller area static pressure analysis

The static pressure distribution of the impeller area is shown in Fig.

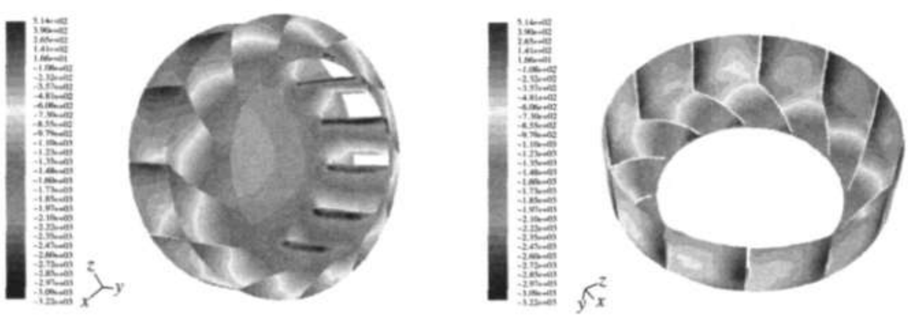


Figure 7 (a) impeller wall static pressure (b) impeller area front and impeller outlet static pressure

In the vicinity of the non-working face and the front plate, especially in the intersection area of the two, a low energy fluid area is accumulated, and the static pressure and the relative velocity are low. The trailing area is formed here, but the trailing area is not completely ' There is fluid passing through only the lower speed. Leaf surface and the front plate near the hydrostatic pressure, the relative speed are high, where the formation of the jet area. Fisher and Thpo-ma are used as the display test in the centrifugal pump impeller, and Cao Shuzhen uses the PIV method for three-dimensional flow measurement, and the presence of this area is verified by the flow photograph. This is later Wu Yulin and other scholars said the jet - wake flow structure.

6.4. Analysis of blade hydrostatic pressure

The static pressure on the working surface of Fig. 8a is obviously different from that of non-working surface in Fig. 8b: the static pressure distribution on the blade face is uneven, and more than 85% of the work is seen from the distribution. Surface static pressure distribution is more uniform, from the root of the blade to the top gradually increased. In a single leaf, the velocity of the airflow in the side walls of the two side walls is relatively low, and the gas flows from the high pressure zone to the low pressure zone by the pressure difference, which flows perpendicularly to the main air flow to produce a secondary flow.

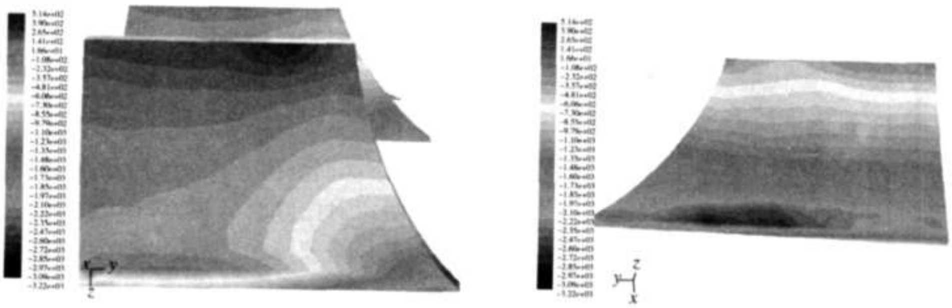


Figure 8 (a) blade working surface static pressure distribution (b) leaf non-working surface static pressure distribution

6.5. Full pressure analysis of the machine

Including all the outer wall cannot see the internal full pressure distribution, as shown in Figure 9 analysis.

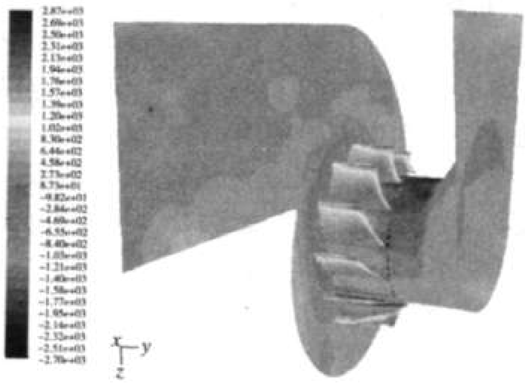


Figure 9 Wall full pressure side view

It is clear from Figure 9 that the change in the distribution of full pressure, especially in the collector and impeller area changes the most obvious. The low pressure at the collector is due to the gradual change in air flow from the axial direction to the radial direction. Impeller area depends on the impeller to do work, the total pressure in the leaf channel gradually increased, and in the leaf surface to reach the maximum exit, into the volute after the flow loss and gradually reduced. The flow of these two areas is very different, which is due to the different position of the flow channel and volute caused by the asymmetry. So for the whole fan, it is not possible to study a part or a flow path, because it is not only difficult to reflect the actual situation of the overall flow field, and the calculated boundary conditions are difficult to determine, which is accurate for the calculation Sex, reasonableness difficult.

6.6. Flow path area velocity analysis

The velocity of the region varies little, but the rotation of the rear panel and the stationary volute wall make the part of the fluid distorted, the center of the fluid rotation near the volute and the flow inside the impeller is not on the same axis , This is the machine simulation is another important phenomenon. The velocity of the fluid between the front and the volute changes obviously, and the airflow suddenly expands at the exit of the impeller, resulting in a decrease in the velocity of the air and the impulse disturbance of the main gas, resulting in the cross section of the secondary flow fan impeller at the volute Velocity distribution, from which to see the fluid from the impeller inlet to the impeller exit direction gradually increased, the impeller after the speed gradually reduced. The speed within the impeller channel near the exit of the volute is smaller than the velocity in the impeller passage of the other part, so it is assumed that each impeller channel is the same or not when calculating the flow path of the fan wheel passage.

It was observed that at the lower right corner of the volute outlet, the phenomenon of secondary flow was observed, located in the volute expansion side of the volcano near the volute, is the volcano outlet flow channel and the volute at the mouth of the common The role of the resulting. From the vortex position, the upper vortex is closer to the front wall of the volute, the lower vortex is stronger than the upper vortex, and near the bottom of the volute. For a comprehensive analysis of the graph, it can be concluded that the fluid does not flow out of the volcano in the volute, but in the twist-like swirl state.

It was observed that there was a return in the volute outlet channel, clearly that the fluid is distorted with swirling flow, indicating that the air flow in the intake chamber changes little, and into the impeller center after the flow is relatively uniform, the flow rate with the direction of the blade gradually increased in the front plate of the volute expansion of the secondary flow generated and more obvious. At the same time, it was observed that the secondary flow is stronger at the gap between the rear disc and the volute wall, so the noise here is relatively large, which can provide the theoretical basis for the analysis of noise. The extension of the volute exit is evident in the strong flow of shares, which was suggested by Wu Yulin et al. wake-jet structures.

7. Conclusions

In this paper, the three-dimensional numerical simulation of the turbulent flow inside the centrifugal fan was carried out, and the internal flow of the centrifugal fan was observed. The pressure and flow field of each part of the flow channel are analyzed emphatically.

(1) The non-axis symmetry of the pressure and velocity of the fluid region is found due to the non-axis symmetry of the machine. The centers of the fluid flow and f the flow field in the impeller area are not in the direction of the central axis, but to deviate from the central axis. The whole movement of the volute is like a twisted twist-like swirling flow;

(2) The results show that there is a wake-jet phenomenon between the blade and the front disk and the volute exit;

(3) Near the impeller front of the blade by the total pressure is higher than the pressure at the root of the root. The pressure on the working surface is greater than the pressure on the non-working face blades, resulting in the occurrence of the pressure difference, so that the fluid from high pressure to low pressure flow generated axial secondary flow phenomenon.

References

1. Cao Shu-zhen, Qi Da-tong, Zhang Yi-yun, et al. Three-dimensional flow measurement of centrifugal fan volute under low flow conditions [J]. Journal of Xi'an Jiaotong University, 2002.
2. Wu Yulin, Chen Qingguang, Liu Shuhong. Ventilator and Compressor [M]. Tsinghua University Press, 2005. 1.
3. Xu Baoren. Frequency conversion pump characteristics and energy saving [J]. Agricultural Equipment Technology, 2008.
4. Sun Hongyan. High-voltage frequency conversion technology in the production of water systems in the application [J]. A heavy technology, 2008.
5. Guo Lijun. Pumps and fans [M]. Beijing China Electric Power Press, 2004.
6. Yang Naiqiu. Hydraulic speed control and energy saving [J]. Energy Conservation and Safety, 2008.
7. Guan F. Modern pump technical manual [M]. Beijing: Beijing Aerospace Publishing House, 1995.
8. Ministry of Machinery Industry, Fourth Design Institute. Paint shop equipment design [M]. Beijing: Machinery Industry Press, 1985.
9. Lu Yaoqing. Heating and ventilation design manual [M]. Beijing: China Construction Industry Press, 1987.
10. Changchun Metallurgical Architecture School. Ventilation Engineering [M]. Beijing: China Construction Industry Press, 1981.
11. Lu Yaoqing. Heating and ventilation design manual [M]. Beijing: China Construction Industry Press, 1987.
12. Su Fulin, Deng Huqiu. Fluid mechanics pumps and fans [M]. Beijing: China Construction Industry Press, 1985.
13. WANG Jia-bing, QUAN Ying-da. Study on incoming flow characteristics and noise of multi-wing centrifugal fan [J]. Fluid Mechanics, 2004
14. Liu Lu. The structure optimization of the centrifugal fan impeller [J]; Zhejiang Hangzhou; Zhejiang University of Technology;
15. Wang Weibin. Numerical simulation of pressure pulsation and aerodynamic noise in the whole flow field of the rotary fan [D], Shandong Qingdao: Shandong University of Science and Technology, 2009.
16. Chen Huaixiu, Li Song. Three-dimensional numerical simulation to improve the design of centrifugal fan [J]. Fan Technology, 2003.
17. Zhang Li, Wang Qijie, Chen Hanping. Numerical calculation of unsteady flow in centrifugal impeller machinery [J]. Fan Technology, 2004.

18. WANG Fu-jun, ZHANG Ling, ZHANG Zhi-min. Study on pressure pulsation characteristics of unsteady flow field of axial flow pump [J].
19. Shao Jie, Liu Shuhong, Wu Sufeng, Wu Yulin. Axial flow model hydraulic turbine pressure pulsation test and numerical calculation prediction [J]. Journal of Engineering Thermophysics, 2008.
20. Zhang Liang, Wu Weizhang, Wu Yulin, Tao Xingming, Liu Shuhong. Prediction of Pressure Fluctuation in Francis Turbine [J]. Large Electric Machine Technology, 2002.
21. Wu Sufeng, Wu Yulin, Liu Shuhong. Effect of Axial Flow Turbine Swing on Pressure Fluctuation [J]. Journal of Engineering Thermophysics, 2007.
22. Wang Fujun. Computational fluid dynamics analysis. Beijing: Tsinghua University Press, 2004.
23. Tong Binggang. Kong Xiangyan, Deng Guohua, et al. Gas Dynamics, Beijing: Higher Education Press, 1995.
24. Espina P I, Piomelli U. Study of the gas jet in a close-coupled gas-metal atomizer. Technical Report, American Institute of Aeronautics and Astronautics, 98-0959. 1998.
25. Singh D D, Dangwal S. Effects of process parameters on surface morphology of metalwave produced by free fall gas atomization. J Mater Sci, 2006, 41 (12): 3853-3860.
26. Jeyakumar M, Gupta G S, Kumar S. Modeling of gas flow inside and outside the nozzle used in spray deposition. J Mater Process Tech, 2008, 203 (1-3): 471-479.
27. Meakhail, T., Zhang, L., Du, Z. H., Chen, H. P., Jansen, W., 2001. The Application of PIV in the Study of Impeller Diffuser Interaction in Centrifugal Fan. Part II-Impeller-Vaned Diffuser Interaction, 'Proceedings of The ASM E Fluid Engineering Division - IM ECE2001 / FED-24953 November 11-16, 2001, New York, USA.
28. Fisher K, Thpoma D. Investigation of the flow condition in a centrifugal puma. Transition of ASME, HYD, 1932, 54 (8): 45-56.