

Numerical investigation on the effect of inlet acceleration on inlet flow distortion in a centrifugal fan

Wang Yan^{1, a}, Dong Quanlin¹, Liu Xiaomeng¹

¹School of Instrumentation Science and Opto-electronics Engineering,
Beihang University, Beijing, 100191, China

^awybuaawork@163.com

Keywords: Inlet flow distortion, Numerical investigation, Centrifugal fan, Inlet acceleration

Abstract. Centrifugal fan is most widely used as an important component in ventilation system. The performance of centrifugal fan is sensitive to the inlet environment; therefore inlet flow distortion is one of the most attention problems. Numerical investigation has been carried out for the comparison of two kinds of inlet, one is standard straight duct and the other is a 90° curved duct. The results show that the distortion of inlet will seriously affect the impeller. Based on the theory of internal flow, a reasonable pressure gradient is added between the impeller and the curved duct, which is caused by inlet accelerator. Numerical simulation results show that acceleration can effectively suppress the inlet flow distortion; meanwhile it cannot affect the regular work of impeller.

Introduction

There are many factors affecting the performance of centrifugal fan, most researches consider how to reduce the energy loss during air movement. But as a part of ventilation system, it will also be affected by the design of the inlet duct. In the industrial field, such as air conditioning and building ventilation, a ventilation system is usually restricted by the weight and space requirements of installation environment, therefore the curved duct will be installed before the inlet of centrifugal fan, which causes air is unevenly distributed in front of the impeller inlet. The inlet flow distortion has a significant influence on the performance of the whole machine, the stable operating range and the aerodynamic noise.

Three kinds of inlet distortion effects on performance and surge margin were studied by Ariga et al. [1]. The results showed that the influence of different inlet distortion on the performance of the whole machine is different. Wright et al.[2] attempted to quantify the inlet flow distortion effects on the performance of centrifugal fan. The test results showed that the inlet flow distortion can lead to the reduction of the efficiency and the pressure of centrifugal fan, the performance can be reduced to 10%~15%, and the range of rotating stall will be affected as well. Zemp [3] used fast response aerodynamic probe (FRAP) for experimental testing based on different forms of inlet flow distortion. Bayomi et al. [4] added inlet straighteners to change the internal flow environment to reduce the inlet distortion and secondary flow, which can avoid reducing the efficiency and pressure. Wang Leilei et al. [5] evaluated the influence of inlet pipe on the aerodynamic performance of centrifugal compressor at different positions. In order to qualitatively analyze the influence of inlet environment on the performance of centrifugal compressor and the mechanism of internal flow, Kim et al. [6] installed a straight pipe and a 90 degree bend in front of centrifugal compressor. The results showed that the influence of inlet environment on the performance of the whole machine is very significant. Then on the basis of their own work, they added guide vanes in the bend, the results showed that the efficiency is improved by 3% [7].

In this paper, a common 90 degree curved duct is taken as an example, the mechanism of air flow in the duct and the influence on the flow field in the meridional plane are also taken into account. According to the mechanism analysis of inlet flow distortion and internal flow theory, in design condition, the distortion is mainly due to the increase of reverse pressure gradient caused by

the centrifugal force, and the momentum is not enough to overcome the viscous and pressure difference. Therefore we can reduce the influence of boundary layer separation by reducing the pressure gradient. The purpose of the optimization is to improve the performance of the inlet and do not spend too much in this paper.

Tested Models

There are different inlet flow distortion forms when using different kinds of ducts in centrifugal fan. The main research work of this paper is that the inlet flow distortion is caused by 90 degree curved duct. The main parameters of the centrifugal fan are shown in Table 1. In order to better understand the internal flow of the machine, it is necessary to record the flow characteristics at each important position; each position is as shown in Fig.1a. The geometric parameters of 90 degree curved duct are shown in Fig.1b, where $D_0=480\text{mm}$, $A_0/D_0\approx 1.021$, $R_4/D_0\approx 0.888$, $L_4/D_0\approx 4.438$, $L_3/D_0\approx 0.95$. Before the inlet structure is optimized, the inlet nozzle is assumed to be a straight duct type, $R_3\rightarrow\infty$.

Table 1. The main parameters of the centrifugal fan.

Parameters	Value	Parameters	Value
Specific speed N_s	42	Blade inlet angle $\beta_1/^\circ$	33
Pressure coefficient ψ_t	1.05	Inlet diameter of impeller D_1/mm	432
Flow coefficient φ	0.1	Inlet width of impeller B_1/mm	250
Specific diameter D_s	3.184	Blade outlet angle $\beta_2/^\circ$	60
Blade number Z	12	Outlet diameter of impeller D_2/mm	1020
Rotational speed N/rpm	1480	Outlet width of impeller B_2/mm	90

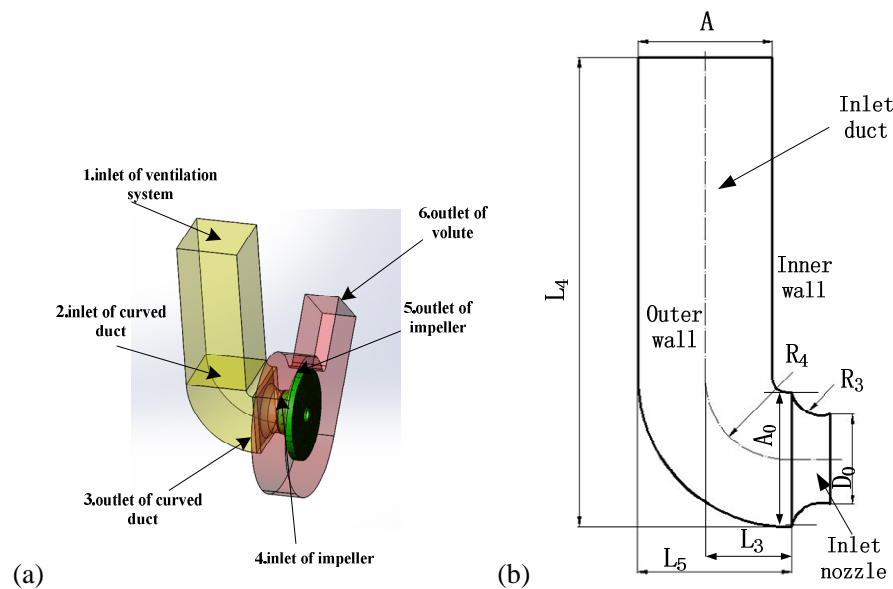


Fig. 1 Test model and geometric parameters of inlet duct.

In the theory of internal flow, the main function of the nozzle is to reduce the pressure in the flow direction, which can keep the smaller boundary layer thickness and weaken the boundary layer separation. Therefore, the inlet flow distortion is optimized by the nozzle structure in this paper, where $D_0=480\text{mm}$, $R_3/D_0\approx 0.336$, $L_0/D_0\approx 0.425$, $A_0/D_0\approx 1.479$. In order to achieve the appropriate acceleration effect, the value of A_0 is increased.

Numerical Simulation and Verification

The test model is meshed by the commercial software ICEM. Unstructured grids are used in inlet,

impeller and volute. The number of grids is 3.13×10^6 before optimization and 3.19×10^6 after optimization. The numerical simulation is calculated by CFX software, and the standard $k-\varepsilon$ model is used as a turbulent model. This model has been successfully applied in many types of fluid machinery, which proves that the standard $k-\varepsilon$ model can predict the performance and the velocity and pressure distribution of centrifugal fan [8, 9].

After introducing the standard $k-\varepsilon$ model, the continuity and momentum equations are expressed in CFX as follow:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j} (\rho U_j) = 0 \quad (1)$$

$$\frac{\partial \rho U_i}{\partial t} + \frac{\partial}{\partial x_j} (\rho U_i U_j) = -\frac{\partial p'}{\partial x_i} + \frac{\partial}{\partial x_j} (\mu_{eff} (\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i})) + S_M \quad (2)$$

$$\frac{\partial \rho h_{tot}}{\partial t} - \frac{\partial p}{\partial t} + \frac{\partial}{\partial x_j} (\rho U_j h_{tot}) = \frac{\partial}{\partial x_j} (\lambda \frac{\partial T}{\partial x_j} - \rho \overline{u_j h}) + \frac{\partial}{\partial x_j} [U_i (\tau_{ij} - \rho \overline{u_i u_j})] + S_E \quad (3)$$

where, U_i represents the mean velocity, $-\rho \overline{u_i u_j}$ represents the Reynolds stress and h_{tot} represents the total enthalpy, $\mu_{eff} = \mu + \mu_t$, μ_t is turbulent viscosity and its expression is

$$\mu_t = C_\mu \rho \frac{k^2}{\varepsilon} \quad (4)$$

p' is pressure, the expression is

$$p' = p + \frac{2}{3} \rho k + \frac{2}{3} \mu_{eff} \frac{\partial U_k}{\partial x_k} \quad (5)$$

Turbulent kinetic energy k and turbulent dispersion ε are solved directly by the discrete transport equation:

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial}{\partial x_j} (\rho U_j k) = \frac{\partial}{\partial x_j} ((\mu + \frac{\mu_t}{\sigma_k}) \frac{\partial k}{\partial x_j}) + P_k - \rho \varepsilon + P_{kb} \quad (6)$$

$$\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial}{\partial x_j} (\rho U_j \varepsilon) = \frac{\partial}{\partial x_j} ((\mu + \frac{\mu_t}{\sigma_\varepsilon}) \frac{\partial \varepsilon}{\partial x_j}) + \frac{\varepsilon}{k} (C_{\varepsilon 1} P_k - C_{\varepsilon 2} \rho \varepsilon + C_{\varepsilon 1} P_{\varepsilon b}) \quad (7)$$

where, $C_\mu=0.09$, $C_{\varepsilon 1}=1.44$, $C_{\varepsilon 2}=1.92$, $\sigma_k=1.0$, $\sigma_\varepsilon=1.3$.

The mesh of the centrifugal fan is composed of four zones: inlet duct, inlet nozzle, impeller and volute. Curved duct, inlet and volute are considered as stationary frame, whereas impeller zone is considered as rotating frame. Frozen rotor is used to define the interface between impeller and inlet and volute. Mass flow rate and opening pressure are considered as the inlet and outlet boundary conditions. The turbulence intensity is assumed to be 5%. The discretization algorithm for advection terms is calculated by high resolution scheme. Inlet mass flow and outlet mass flow are monitored by monitor. When convergence criteria, RMS is less than 1×10^{-6} or the mass flow difference between inlet and outlet is less than 1×10^{-2} , is achieved, the calculation will end.

In order to verify the accuracy and reliability of the numerical simulation method, it is necessary to verify the numerical simulation method. Experimental equipment and methods are carried out according to GB/T 1236-2000. The type of installation uses the pipe inlet and free outlet.

The test data included the critical parameter of centrifugal fan, such as volume flow, pressure, efficiency, noise and so on at a given speed of rotation. The test and simulation results of pressure and efficiency are shown in Fig. 2. The results show that the errors of the efficiency and pressure are 3.972% and 4.44% at the design point. The distortion of the simulation results will get worse as the test point is far away from the design point, such as, the errors of the efficiency and pressure increase to 5.463% and 9.337% when the volume flow is $31644 \text{ m}^3/\text{h}$. It is concluded that this numerical simulation method can accurately predict the aerodynamic performance of the centrifugal fan at the design point within an acceptable margin of error.

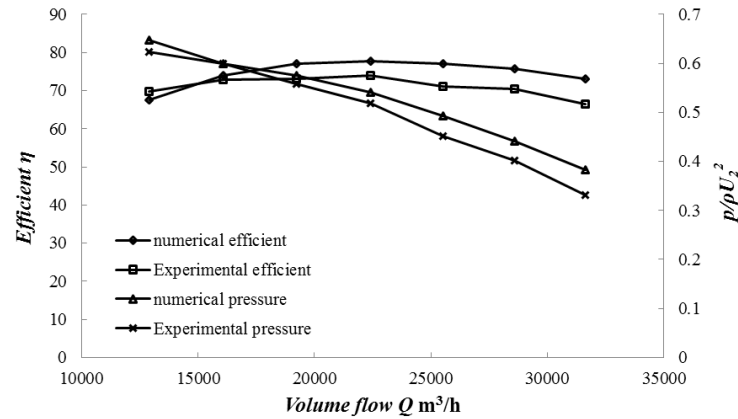


Fig. 2 Performance curves of the prototype.

Simulation Results Analysis

When the inlet is straight duct, the inlet of impeller is considered as axial inflow which means the inlet flow without distortion. When the inlet is curved duct, the flow field of impeller inlet is no longer uniform which means the inlet flow with distortion. Fig. 3 shows the velocity profile in the cross-section at the position 4. Since the value of the A_0 is changed after optimization of the inlet, the two different bend inlets are compared with the straight inlets, respectively. It can be seen that the thickness of boundary layer near the inner wall of bend is increasing, which lead to the decrease of the flow area in the cross-section before optimization as shown in Fig. 3a. Therefore the velocity distribution is distorted in the cross-section. The low velocity zone and flow separation zone will generate coupling distortion. When the inlet flow is accelerated after optimization, although the curved duct also generates a flow distortion, the velocity distortion zone has been reduced to a small extent as shown in Fig. 3b. The velocity distribution of bend inlet in most areas of the cross section is consistent with the velocity distribution of the straight inlet.

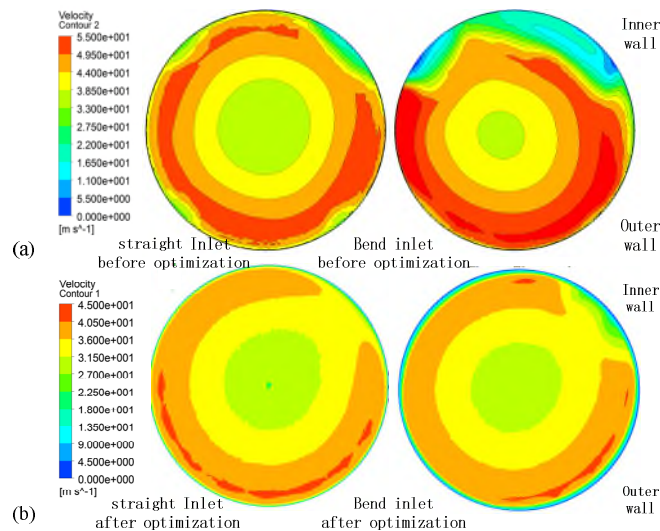


Fig. 3 Velocity profile in the cross-section at the position 4:
(a) before optimization, (b) after optimization.

Due to presence of inlet flow distortion, the relative airflow angle of inlet airflow is changed. According to the simulation data, the change of relative airflow angle caused by inlet distortion is analyzed. According to the Eq. (8), the expression of the relative flow angle is shown as follows:

$$\beta = \arctan(c_m / w_u) \quad (8)$$

The impeller rotation direction is defined as negative direction. The distributions of the flow angle along the blade height are shown in Fig. 4, where curve 1 in the figure represents the flow angle at the blade leading edge near the outer wall, curve 2 represents the flow angle at the blade leading edge near the inner wall. Because of the boundary separation at the shroud of centrifugal impeller, the distributions of the flow angle of straight inlet is different with that of bend inlet at the inner wall, but it is the same at the outer wall. According to the Fig. 4a, it can be seen that the inlet distortion can affect the zone up to 60% blade height at the inner wall. The influence of blade leading edge will inevitably affect the tail of blade. Fig. 4b shows the effect of inlet after optimization on the inlet flow distortion at the inner wall. It can be seen that the distortion of airflow angle is weakened to a great extent compared with that before optimization, and the distortion zone is reduced within 30% blade height.

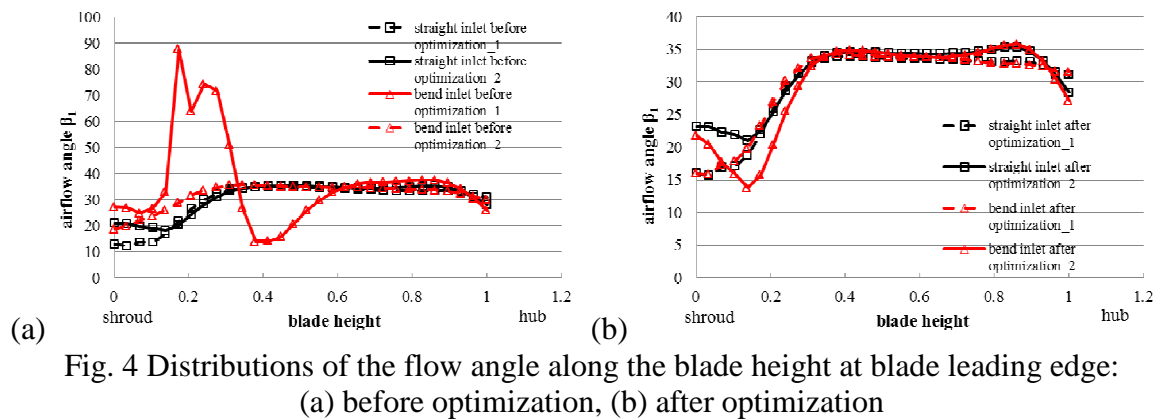


Fig. 4 Distributions of the flow angle along the blade height at blade leading edge:
(a) before optimization, (b) after optimization

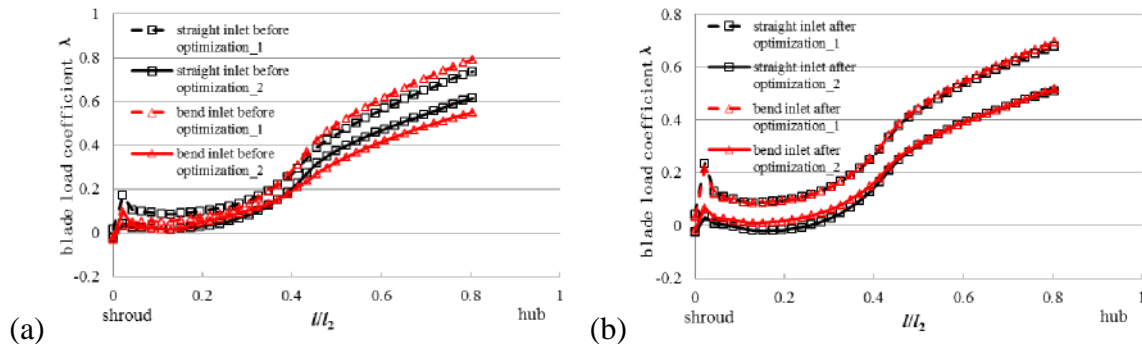


Fig. 5 Blade load coefficient distribution along the blade:
(a) before optimization, (b) after optimization.

Due to the centrifugal impeller is eccentric installed in the volute; therefore the blade will be subjected to different static pressure load. The distribution of blade load is related to the blade shape. Fig. 5 shows the blade load coefficient distribution along the blade. Similarly, when inlet flow distortion occurs, the blade load will change at the inner wall and outer wall as shown in Fig. 5a. The influence zone of distortion is mainly in the tail of blade section. Fig. 5b gives the distribution of the blade load coefficient after optimization. Although there are still some effects in the front of blade section, the difference is not obvious. It can be seen that, after the optimized inlet accelerates the distortion flow structure to a certain extent, the distortion flow in the bend inlet can return as the standard flow in the straight inlet.

Table 2 gives the pressure coefficient and efficiency of the centrifugal fan before and after optimization. It can be found that the influence of inlet flow distortion on the performance of centrifugal fan has been weakened to a great extent at the design condition after optimization. Therefore, acceleration can effectively suppress the inlet flow distortion.

Table 2. The pressure coefficient and efficiency of the centrifugal fan.

	Straight inlet before optimization	Bend inlet before optimization	Error before optimization	Straight inlet after optimization	Bend inlet after optimization	Error after optimization
Pressure coefficient	1.077	1.041	3.3%	1.134	1.128	0.5%
efficiency	73.698%	71.825%	2.5%	76.917%	76.609%	0.4%

Conclusion

In practical engineering applications, there is often insufficient space to replace the straight inlet into the bend inlet for installation. It is very easy to cause inlet flow distortion, which results in the non-uniform velocity distribution at the inlet of the impeller. It will be unable to achieve the design performance. In this paper, numerical simulation is used to test the flow characteristics of the impeller under the influence of inlet flow distortion. Test results show that the velocity and airflow angle in the impeller is unevenly distributed, which can have a very negative impact on the performance of centrifugal fan. In order to improve the effect of inlet flow distortion and save space, this paper designs an inlet which can effectively weaken the inlet flow distortion. According to the results of numerical simulation, after the acceleration of inlet, the distortion of the flow field will return to no distortion of the flow field. Compared with straight inlet and bend inlet after optimization, the pressure coefficient error and efficiency error are about 0.5% and 0.4%, respectively.

References

- [1] I. Ariga, N. Kasai, S. Masuda, et al. The effect of inlet distortion on the performance characteristics of a centrifugal compressor, *J. Eng. Gas Turb. Power*, 105(2) (1983) 223-230.
- [2] T. Wright, S. Madhavan, J. Dire, Centrifugal fan performance with distorted inflows, *J. Eng. Gas Turb. Power*, 106(4) (1984) 895-900.
- [3] A. Zemp, CFD investigation on inlet flow distortion in a centrifugal compressor, Zurich: Swiss Federal Institute of Technology, 2007.
- [4] N. N. Bayomi, A. A. Hafiz, A. M. Osman, Effect of inlet straighteners on centrifugal fan performance, *Energy Conv. Manag.* 47(18) (2006) 3307-3318.
- [5] L. L. Wang, D. Z. Lao, Liu H, et al. The influence of different installation angles of the 180° inlet bend pipe on the centrifugal compressor performance, *J. Eng. Thermophys.* 35(9) (2014) 1717-1721.
- [6] Y. Kim, A. Engeda, R. Aungier, et al. The influence of inlet flow distortion on the performance of a centrifugal compressor and the development of improved inlet using numerical simulations, *Proc. Inst. Mech. Eng. Part A J. Power Energy*, 215(3) (2001) 323-338.
- [7] A. Engeda, Y. Kim, R. Aungier, et al. The inlet flow structure of a centrifugal compressor stage and its influence on the compressor performance, *J. Fluids Eng.* 125(5) (2003) 779-785.
- [8] A. Behzadmehr, Y. Mercadier, N. Galanis, Sensitivity analysis of entrance design parameters of a backward-inclined centrifugal fan using doe method and cfd calculations, *J. Fluids Eng.* 128(3) (2006) 446-453.
- [9] M. J. Zhang, M. J. Pomfret, C. M. Wong, Three-dimensional viscous flow simulation in a backswept centrifugal impeller at the design point, *Comput. Fluid.* 25(5) (1996) 497-507.