Employing rotating vaneless diffuser to enhance the performance of plenum fan

Hua-Shu Dou¹, Lin Wu², Yikun Wei¹, Yongning Chen³, Wenbin Cao³, Cunlie Ying³

¹Faculty of Mechanical Engineering and Automation,
Zhejiang Sci-Tech University
Hangzhou, Zhejiang Province 310018, China
huashudou@yahoo.com, ykun-wei@sina.com
²Acon Biotech (Hangzhou) CO., Ltd
Hangzhou, Zhejiang Province 310030, China
wulin_1991@yahoo.com
³Zhejiang Yilida Ventilator CO., Ltd
Taizhou, Zhejiang Province 318056, China
cyn121@yilida.com, caowenbin@yilida.com, yingcunlie@yilida.com

Abstract

Numerical simulation is carried out for flow characteristics in a plenum fan and the influence of the diameter ratio of the rotating vaneless diffuser on the performance of plenum fan is analyzed. The diameter ratio of the rotating vaneless diffuser employed is from 1.03 to 1.3. The research results show that the rotating vaneless diffuser is able to enhance the performance of plenum fan. It is found that there is significant improvement in static pressure and efficiency at the diameter ratio of 1.05 at high flow coefficients, while the optimal diameter ratio is 1.2 at rated and low flow coefficient.

Keywords: Plenum fan; rotating vaneless diffuser; static pressure; efficiency.

1. Introduction

Centrifugal fans were widely used in many fields such as industry, agriculture, mining, chemical engineering and building ventilation. A lot of energy is consumed by fans every day, and the energy used for fans, blowers and compressor occupies a large fraction of the electric energy generated [1]. Thus, it is significant to save energy with operation of high efficiency fans. Therefore, to design centrifugal fans with high efficiency and to operate fans in high efficiency ranges are important.

Plenum fan is a new type of centrifugal fan, which is widely used in air condition devices and roof ventilations [2]. With this type of fans, there is no volute after the impeller. The kinetic energy at the outlet of impeller can not be to large extent converted into static pressure, and some of the kinetic energy is lost. It is seen that the absolute velocity at the outlet of the backward impeller is the smaller than that with radial impeller or forward impeller, as shown in Fig. 1. As such, the kinetic energy loss of backward impeller is the minimum among these three types of impellers. Therefore, backward impeller is widely employed in plenum fans [3]. Compared with traditional centrifugal fans, the plenum fan is of good characteristics such as lower noise, smaller volume [4] and better airflow due to the absence of volute [5]. In addition, the discharging direction of plenum fan is optional, which makes the installation easier. On considering these, plenum fan has a large potential prospect [6].
There are several studies on centrifugal fans in literature. Kim et al [7] studied aerodynamic performance of centrifugal fan with splitter blade, and found that the reverse-flow regions in the blade passage can be reduced by controlling the main blade numbers with splitter blades. Hariharan et al [8] found that the clearance has a positive effect on the centrifugal fan performance at low mass flow rate, but at the higher mass flow rate, it has a negative effect. Dundi et al [9] found that the performance of centrifugal fan can be improved by Gurney flap. The effect of Gurney flap at lower Reynolds number is better than that at higher Reynolds number. The Traditional centrifugal fan converts the kinetic energy into static pressure in volute to reduce the pressure loss [10]. Diffusers are used in some centrifugal fans [11]. There are some achievements in this area in previous researches. Senoo et al [12] found that the flow in vaneless diffusers is strongly asymmetric. Inoue et al [13] pointed out that the flow in centrifugal fan with vane diffuser is more complex than with vaneless diffuser. Li [14] found that the dynamic energy loss is the main part of the diffuser loss. Han [15] investigated the flow field in vaneless and vane diffuser. The results show that distribution of velocity at the outlet of impeller in a centrifugal fan with rotating vaneless diffuser to be more uniform. Sato et al [16, 17] investigated the impeller–vaneless diffuser architecture and found that the architecture could operate over a wide range of flow rates with stable flows and good performance. Allos et al [18] studied the flow in a vaneless diffuser experimentally. The flow structures were found three-dimensional and the flow velocity is not uniform from hub to shroud. Gao et al [19] argued that the static pressure in plenum fan can be enlarged by rotating vaneless diffuser. Although the rotating vaneless diffuser could be applied in plenum fan, but only a few data about this area are found in the literature. More details about the influence of the rotating vaneless diffuser to the performance of plenum fan can be found.

In this paper, the effect of rotating vaneless diffuser on the performance of plenum fan is studied by numerical simulation. Static pressure and static efficiency from numerical simulation are well consistent with the experimental data. Then the rotating vaneless diffuser is established by extending shroud and hub, and the ratio of diffuser and impeller diameters is from 1.03 to 1.30. Meanwhile, simulation of each model is carried out and comparison with the original fan is performed. Measurements of static pressure, static efficiency, and velocity at different flow coefficients are conducted for each case. Then, the velocity and static pressure on the blade surfaces and at impeller outlet at rated and high flow coefficient is analyzed.

In the following, governing equations and numerical method will be briefly described at first. Then geometrical model and description of mesh generation are followed. After that, the detailed discussions of flow field in the plenum fan are presented. Finally, some concluding remarks are provided.

2. Governing Equation

The governing equations for static parts are three-dimensional Reynolds averaged Nervier-Stokes equation (RANS):

\[ \nabla \cdot \ddot{u} = 0 \]  
\[ \frac{\partial \ddot{u}}{\partial t} + (\ddot{u} \cdot \nabla) \ddot{u} = -\frac{1}{\rho} \nabla p + \frac{\mu + \mu_t}{\rho} \nabla^2 \ddot{u} \]  

where \( \ddot{u} \) is the fluid velocity, \( \rho \) is the fluid density, \( \mu \) and \( \mu_t \) are the fluid viscosity and the turbulent viscosity respectively, and \( p \) is the pressure.

In rotating part, for the relative velocity formulation, fluid velocities can be transformed from the stationary frame to the moving frame using the following relation:

\[ \ddot{V}_r = \ddot{V} - \ddot{u}_r \]  
\[ \ddot{u}_r = \ddot{\omega} \times \ddot{r} \]  

where \( \ddot{V}_r \) is the relative velocity (the velocity viewed from the rotating frame), \( \ddot{V} \) is the absolute velocity (the velocity viewed from the stationary frame), \( \ddot{u}_r \) is the velocity of the moving frame relative to the inertial reference frame, and \( \ddot{\omega} \) is the angular velocity.

Then the governing equations of fluid flow in a moving reference frame are as follows:
Conservation of mass:
\[ \Delta \cdot \rho \vec{v}_r = 0 \]  
(5)

Conservation of momentum:
\[ \Delta \cdot \left( \rho \vec{v}_r \vec{v}_r \right) + \rho \left( 2 \vec{a} \times \vec{v}_r + \vec{a} \times \vec{a} \times \vec{r} \right) = -\Delta p + \Delta \cdot \tau_r + \vec{F} \]  
(6)

where \( \tau_r \) is the viscous stress.

The realizable \( k - \varepsilon \) model is selected, and this model was presented by Shih in 1995 [20]. Thinking that the coefficient \( C_\mu \) in the calculation formula of the turbulent eddy viscosity coefficient \( \mu_t \) is related to the strain rate instead of constant. Compared to standard \( k - \varepsilon \) model, the equation of \( \varepsilon \) is changed a lot, and the production items in equation of \( \varepsilon \) no longer contain \( G_k \) as in traditional \( k - \varepsilon \) model. The constraint equations of turbulent kinetic energy \( k \) and turbulence dissipation rate \( \varepsilon \) are as follows:

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_i)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \mu + \frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_j} \right] + \rho (P_k - \varepsilon) 
\]
(7)

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon u_i)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \mu + \frac{\mu_t}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} \right] + \rho C_{\varepsilon_1} E \varepsilon - \rho C_{\varepsilon_2} \frac{\varepsilon^2}{k + \sqrt{\nu \varepsilon}} 
\]
(8)

where

\[
C_i = \max \left( \frac{0.43}{\eta + 5}, 1 \right) 
\]
\[
\eta = \left( 2 E_{ij} \cdot E_{ij} \right)^{1/2} \frac{k}{\varepsilon} 
\]
\[
E_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) 
\]
\[
\sigma_k = 1.0, \quad \sigma_\varepsilon = 1.2, \quad C_{\varepsilon_2} = 1.9 
\]

3. Numerical Investigation

3.1 Numerical methods

The finite volume method is adopted in this work, and the SIMPLEC algorithm is used for pressure and velocity coupling with secondary order upwind difference interpolation. It is necessary for a run to converge to satisfy the requirement of maximum residuals, or differential pressure between inlet and outlet to be stable.

The inlet boundary is velocity inlet and the outlet boundary is pressure outlet using atmosphere pressure. All solid walls are no-slip walls, and the rotating speed of the impeller is 1450 rpm as same as that in the experiment. The multiple rotating reference frames are employed in the calculation considering the static parts and the rotating part.

3.2 Geometrical model and mesh

Original model (named model A in this paper) is a plenum fan numbered with SYW-560 produced by Zhejiang Yilida Company, as shown in Fig. 2. The geometry showed in Fig. 3 is established, and the computational geometry is based on the experimental model with scale in 1:1. For convenience, assistant axial fan is removed which is used in experiment. The outer area of the computing domain is a cylinder whose diameter is 5.2 times the plenum fan diameter and the height is double of the width of the impeller. More details of the plenum fan are given in Table 1.
Table 1 Rated variables of plenum fan

<table>
<thead>
<tr>
<th>Variables</th>
<th>units</th>
<th>value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flow coefficient</td>
<td></td>
<td>0.198</td>
</tr>
<tr>
<td>Static pressure coefficient</td>
<td></td>
<td>0.316</td>
</tr>
<tr>
<td>Efficiency of static pressure</td>
<td>%</td>
<td>50.2</td>
</tr>
<tr>
<td>Revolving speed</td>
<td>rpm</td>
<td>1450</td>
</tr>
<tr>
<td>Impeller diameter</td>
<td>mm</td>
<td>570</td>
</tr>
<tr>
<td>Impeller outlet breadth</td>
<td>mm</td>
<td>133</td>
</tr>
<tr>
<td>Number of blades</td>
<td></td>
<td>12</td>
</tr>
</tbody>
</table>

where the equation of static pressure coefficient is as follows:

\[
\psi_{st} = \frac{\Delta p_{st}}{\rho \left( \frac{\pi D_2 n}{60} \right)^2}
\]

(9)

Table 2 Parameters of different model

<table>
<thead>
<tr>
<th>Name</th>
<th>B</th>
<th>C</th>
<th>D</th>
<th>E</th>
<th>F</th>
<th>G</th>
<th>H</th>
<th>I</th>
</tr>
</thead>
<tbody>
<tr>
<td>D_3/D_2</td>
<td>1.03</td>
<td>1.05</td>
<td>1.07</td>
<td>1.1</td>
<td>1.15</td>
<td>1.2</td>
<td>1.25</td>
<td>1.3</td>
</tr>
<tr>
<td>Diameter (mm)</td>
<td>588</td>
<td>598</td>
<td>610</td>
<td>627</td>
<td>656</td>
<td>684</td>
<td>712</td>
<td>741</td>
</tr>
</tbody>
</table>

The vaneless diffuser studied in this paper is extended the same size from shroud and hub. The ratio between diffuser outlet diameter and impeller outlet diameter is expressed with D_3/D_2. The values of D_3/D_2 are taken as 1.03, 1.05, 1.07, 1.1, 1.15, 1.2, 1.25 and 1.3 respectively, and they are denoted by model B, C, D, E, F, G, H, and I respectively. The sizes are list in Table 2.

Unstructured tetrahedral mesh is used in this research, and the total elements of the original model are about 2,270,000.

4. Numerical Results and Discussion

4.1 Mesh independence verification

Mesh independence verification for original model is performed to find an appropriate amount of mesh number. As is shown in Fig.4, a 2,270,000 mesh system is sufficiently meet requirements and is thus used in this paper (to satisfy the flow characteristics, and is thus utilized in present analysis).

![Fig. 4 Comparison of used meshes](image)

4.2 Numerical methodology verification

Figure 5 shows the comparison of static pressure of numerical simulation and experimental data. The error at small flow coefficients is bigger than that at rated and high flow coefficients. It is found that the flow is more stable at rated and high flow coefficients than that at low flow coefficients, and the calculation of flow field is more accurate.
Figure 6 shows the comparison of static pressure efficiency of numerical simulation and experimental data. The mechanical efficiency and the volumetric efficiency are included in numerical data by reference [21]. The maximum relative error percentage on pressure rise is 11.4% at the flow coefficient of 0.0578, and the minimum relative error percentage on pressure rise is 6% at the flow coefficient of 0.263. In terms of efficiency, the maximum relative error percentage on pressure rise is 7% at the flow coefficient of 0.0578, and the minimum relative error percentage on pressure rise is 3% at the flow coefficient of 0.263. In the whole range, the error is within permission, and the numerical methodology is reliable.

4.3 Impact of the rotating vaneless diffuser

Figure 7 shows the static pressure curves of plenum fans with different rotating vaneless diffuser. It can be seen in Fig. 7 (a) that at rated and small flow coefficients, with the increase of $D_3/D_2$, the static pressure increases gradually. But at high flow coefficients, this is not true, especially at the flow coefficient beyond 0.263. When the flow coefficient is larger than 0.263, the highest static pressure is obtained with model C. The static pressure of all these models is higher than that of the original model at flow coefficient beyond 0.231. When the flow coefficient is larger than 0.294, the static pressure with all these models is almost in the same except model C. The static pressure at high flow coefficients with model B and D are shown in Fig. 7 (b). It can be seen that the static pressure with model B and D is lower than model C at high flow coefficient.

In Fig. 8, the static pressure efficiency of plenum fans with different rotating vaneless diffuser is shown. It can be found that there is no significant difference among model G, H and I at the smallest flow coefficient. At the rated flow coefficient, model F has the highest efficiency. For flow coefficient higher than 0.263, efficiency of model C is higher than that of others, because the static pressure of model C is higher than that of others and the energy consumption is less. Fig. 8 (b) shows that the static pressure efficiency of model C is still the highest compared with model B and D.
At the rated flow coefficient, analysis of static pressure on suction surface and pressure surface of a blade are taken respectively for model A, C, E, F and G. The velocity near the blade surface is analyzed as well. The axial position is located at the middle of impeller meridian section. The static pressure and velocity are plotted in Fig. 9 and Fig. 10. The abscissa stands for the position along the direction of blade curve from impeller inlet.

It can be seen in Fig. 9 that the static pressure in both suction and pressure surfaces increases along the blade length, and the static pressure of two sides are almost same at the end of blade. For the pressure distribution on the pressure surface, there is no obvious distinction in these five models, but the distribution of static pressure of suction surface in model C, E, F and G are stable than model A (original). Therefore, the flow on the suction surface are improved for the models C, E, F and G.
It can be seen from Fig. 10 that the velocity near both suction and pressure surfaces increases along the blade length. At the end of blade, the velocity of model C, E, F and G is lower than that of model A on the suction surface, while it is almost same near the pressure surface. The velocity decreases sharply at the end of blade on the pressure surface of model A, while in other models, the velocity is uniform or decreases slowly at the same position. These results show that the flow in blade passages can be improved by rotating vaneless diffuser.

Velocity at the outlet of the passage in middle section of the impeller is analyzed and shown in Fig. 11. The abscissa expresses the position from suction surface to pressure surface. It can be found that there is an obvious jet-wake pattern in model A, and there is a peak in the velocity distribution, which leads to the flow more non-uniform. In other models, the velocity at the outlet of the passage decreases and becomes more uniform across the blade-to-blade channel at impeller outlet with the increase of D3/D2. This indicates that rotating vaneless diffuser makes the flow decelerated from impeller outlet to diffuser outlet, which leads to static pressure is increased. The bigger the diffuser diameter, the lower the velocity at outlet and the higher the static pressure of plenum fan.

At the larger flow coefficient of 0.294, analysis of static pressure and velocity on suction surface and pressure surface of a blade are taken respectively for model A, C, E, F and G as well. The axial position is located at the middle of impeller meridian section. The static pressure and velocity are plotted in Fig. 12 and Fig. 13. The abscissa indicates the position along the direction of blade curve from impeller inlet.
It can be seen in Fig. 12 that the increase of static pressure of model A, C and E in pressure surface are higher than that of others, but the static pressure at outlet is almost the same. In the suction surface, the increase of the static pressure of model C is the highest, while other four models is almost the same. Therefore, at this flow coefficient, the highest static pressure on blade is achieved with model C. This is the reason why model C has the highest static pressure and static efficiency at this flow coefficient. It also means that the flow in the passage can be improved by reasonable design of rotating vaneless diffuser. The rotating vaneless diffuser with diameter ratio of 1.05 has more favorable influence at this flow coefficient.

Figure 13 shows the velocity distribution at flow coefficient of 0.294. At the leading edge of blade, the velocity increases sharply on suction surface. This is due to the impact influence at this position when flow coefficient is higher than the flow coefficient at design condition. On pressure surface, the velocity is higher than that of suction surface in middle and rear part of the blade. At the end of the blade, the velocity in the suction surface is higher than that on the pressure surface. Figure 14 shows the velocity distribution at the outlet of the passage in the middle section of the impeller at the flow coefficient of 0.294. The abscissa indicates the position from suction surface to pressure surface. It can be seen that an obvious jet-
wake pattern exists in model A. The flow velocity of model C, F and G slows down after through the diffuser and then becomes more uniform. However, for model E, there is almost no improvement. Thus, at this flow coefficient, the diffusion role of model F and G are better, and model C is middle, and the diffusion role of model E isn’t obvious. Through the above analysis, it is realized that model E doesn’t perform well, and neither blade nor diffuser gives good performance.

Fig. 14 Velocity distribution on passage outlet at flow coefficient of 0.294

5. Conclusions
Numerical simulations of flow in a plenum fan with rotating vaneless diffuser with different diameter ratio $D_3/D_2$ are carried out in this paper. The finite volume method and SIMPLEC algorithm are used in the solution. Numerical results are in good agreement with the experimental data, thus proving the reliability of the numerical simulation. The performance of the plenum fan with various rotating vaneless diffuser is compared with the original model. The conclusions could be summarized as follow:

1. Rotating vaneless diffuser to some degree increases the static pressure in plenum fan. It is able to convert part of the kinetic energy at impeller outlet into static pressure efficiently.

2. Rotating vaneless diffuser has an influence on inner flow in the impeller, which lets velocity on the pressure surface at the outlet of blade more uniform, and the distribution of static pressure on the suction surface is more stable at rated flow coefficient.

3. The efficiency of models with diameter ratio 1.15~1.25 is higher than that of others at rated and low flow coefficients. Considering the size of impeller, the model F is the optimal model.

4. At high flow coefficient, the static pressure and efficiency of plenum fans with diffusers at $D_3/D_2$ of 1~1.1 are higher than other models. The model C ($D_3/D_2=1.05$) has the highest static pressure and static efficiency.

Acknowledgements
This work is supported by Zhejiang Province Key Science and Technology Innovation Team Project (2013TD18) and Natural Science Foundation of Zhejiang Province (LY14E060003) as well as the National Natural Science Foundation of China (51579224, 51536008).

Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$D_3$</td>
<td>Diameter of diffuser outlet</td>
</tr>
<tr>
<td>$D_2$</td>
<td>Diameter of impeller outlet</td>
</tr>
<tr>
<td>$\bar{u}$</td>
<td>fluid velocity</td>
</tr>
<tr>
<td>$\rho$</td>
<td>fluid density</td>
</tr>
<tr>
<td>$\mu$</td>
<td>fluid viscosity</td>
</tr>
<tr>
<td>$\mu_t$</td>
<td>turbulent viscosity</td>
</tr>
<tr>
<td>$\mu_r$</td>
<td>relative velocity</td>
</tr>
<tr>
<td>$\dot{\nu}$</td>
<td>absolute velocity</td>
</tr>
<tr>
<td>$\bar{u}_r$</td>
<td>velocity of the moving frame</td>
</tr>
<tr>
<td>$\omega$</td>
<td>angular velocity</td>
</tr>
<tr>
<td>$\bar{\rho}$</td>
<td>position vector</td>
</tr>
<tr>
<td>$\tau_r$</td>
<td>viscous stress</td>
</tr>
<tr>
<td>$k$</td>
<td>turbulent kinetic energy</td>
</tr>
<tr>
<td>$\varepsilon$</td>
<td>turbulence dissipation rate</td>
</tr>
</tbody>
</table>

References