Numerical and experimental investigations on non-axisymmetric D-type inlet nozzle for a squirrel-cage fan

Hui Liu, Boyan Jiang, Jun Wang, Xiaopei Yang and Qianhao Xiao

School of Energy and Power Engineering, Huazhong University of Science and Technology, Wuhan, People’s Republic of China

ABSTRACT
In this study, the unique reverse-flow around the inlet nozzle of a squirrel-cage fan was numerically studied by computational fluid dynamics (CFD), and a non-axisymmetric D-type inlet nozzle was proposed to inhibit it. It is shown that serious back flow develops at the volute tongue when the fan operates at a low flow coefficient that is lower than the best efficiency point. The airflow passing the blade passages in reverse further crosses the impeller, mixing the clearance leakage airflow, and finally blows into the intake chamber near the circumferential position of 150°. By installing the inlet nozzle along this direction, D-type inlet nozzle schemes with different blockage extent were calculated by CFD. The scheme with a cut distance of 70% of its exit-section radius was chosen as the best design by comprehensively comparing the back flow ratio and fan performance of different working conditions. CFD results illustrated that reverse blowing at the fan entrance was significantly constrained by this D-type inlet nozzle, and the vortex in the inlet chamber correspondingly disappeared. Experimental data illustrated that the new nozzle increased the fan pressure rise under low-flow conditions by about 6% and barely reduced performance at other working points.

ARTICLE HISTORY
Received 16 November 2020
Accepted 21 January 2021

KEYWORDS
Squirrel-cage fan; inlet nozzle; computational fluid dynamics; low-flow condition; backflow; aerodynamic performance

1. Introduction
Squirrel-cage fans are widely applied in heating, ventilation, and air conditioning (HVAC) devices due to their lower noise compared to other types of fans in the same size (Eck, 1973). As HVAC equipment appears so frequently in daily life, ensuring the squirrel-cage fan operates powerfully, efficiently, and quietly is the primary concern of researchers in this field. A full understanding of the internal flow structure in these fans will provide a foundation for improving their aerodynamic performance.

Squirrel-cage fans received their name due to their large diameter ratio, blade width, and blade quantity. Their blade channels are short, so the axial intake airflow cannot turn radial in time. Most of the airflow is concentrated in the rotor’s hub-side, which induces a large flow separation at the rotor shroud. Previous studies showed that the airflow passing through the front 1/4 of the blade channels was negligible; and that only about the hub-side 3/4 of the channels was useful for energy conversion (He & Sato, 2001; Raj & Swim, 1981; Yamazaki & Satoh, 1986, 1987). Non-uniform impeller outflow further induced a mass-flow imbalance along the axis-wise, result in the secondary vortex in the volute (Kitadume et al., 2007). This flow non-uniformity starts from the inlet, runs into the impeller, and develops in the volute. It is the most distinct flow structure in this type of centrifugal fan and is generally considered responsible for poor efficiency (Montazerin et al., 1998).

As this non-uniform flow starts from the fan entrance, a reasonably designed inlet nozzle is the key to improving the internal flow uniformity. Many researchers have investigated the effects of the nozzle’s geometric shape and installation location on the internal flow and aerodynamic performance. It has been shown that the inlet nozzle with an exit diameter smaller than the impeller’s inner diameter can reduce the impact of the clearance leakage on the mainstream and improve the airflow utilization of the impeller (J. B. Wang & Ou, 2004). In contrast, a reverse installed inlet nozzle with an exit diameter larger than the impeller’s inner diameter was found to provide a more uniform flow into the volute than the traditional one (Montazerin et al., 2000; J. B. Wang & Ou, 2004). Besides, other studies (Gholamian, Rao, & Bharamara, 2013; Gholamian, Rao, & Panitapu, 2013) have suggested that the inlet nozzle’s exit diameter should be equal to the impeller’s inner diameter. When the nozzle’s exit diameter is smaller than the rotor’s inner
diameter, appropriately increasing the axial gap between the two components can weaken the separation vortex near the impeller shroud (Gholamian, Rao, & Panitapu, 2013).

Squirrel-cage fans are usually installed in a compact structure unit (Akimoto et al., 1996) and tend to operate in an off-design condition with a relatively low flow rate. At this time, the fan internal flow exhibits extreme non-uniformity along the circumferential direction (Adachi et al., 2004; Raj & Swim, 1981). The measured fan flow field shows that the reverse flow easily penetrates through the rotor near the volute tongue as the blade channels are short (Kadota et al., 1994). The back flow will further form a cross-impeller flow structure (Gyeong et al., 2005; Kind & Tobin, 1990), the location and the scope of which obviously affect the fan performance and noise.

Under this low-flow operating condition, fan components designed by traditional methods may not perform the best and their aerodynamic shape require alterations. For instance, the traditional volute profile is usually designed based on the assumption of uniform impeller outflow, meaning the resulting profile should be a spiral profile with a constant spread angle (Eck, 1973). However, after using the real non-uniform outflow on the volute inlet circle to perform the inverse design iteration, the obtained spiral volute profile with a non-constant spread angle performed better (Qi et al., 1996). Furthermore, a non-continuous volute profile obtained by cutting the volute was even beneficial to the fan pressure under low-flow (Jiang et al., 2018). Inlet nozzles with a non-axisymmetric shape or that were eccentrically installed were also applied. It was found that the optimal eccentric direction of the nozzle was in the direction away from the volute outlet. At this point, the circulating air flow generated between the inlet nozzle and the impeller promoted a secondary energy conversion in the rotor, which improved the fan pressure (Fukutomi et al., 1999). Eccentric nozzles improved the flow separation near the rotor shroud in the part of the blade passages, and the fan flow rate correspondingly increased (J. Wang et al., 2005). Various eccentric and elliptical nozzles were further applied on a double-suction squirrel-cage fan, and corresponding optimal eccentric and shape parameters were also suggested (Wen et al., 2013).

This study focuses on the asymmetric design of the fan inlet nozzle so it is necessary to first observe the flow state near the fan’s entrance. As mentioned above, the internal flow of squirrel-cage fans has been measured and analyzed by many researchers. However, most have focused on the flow separation between blades (Akbari et al., 2012) or the secondary flow in the volute (Kitadume et al., 2007), and little attention has been paid to the inlet region. Besides, most of the squirrel-cage fans studied in the above literature were operated in an open intake environment. How an intake chamber affects the flow near the fan entrance remains unclear and requires further investigation. In recent years, computational fluid dynamics (CFD) has been widely applied in the engineering field to investigate fluid problems at different scales (Elfarra, 2019; Ramezanizadeh et al., 2019). By this method, the three-dimensional fan flow field at different operating conditions (K. Wang et al., 2018), and even the transient characteristics (Oro et al., 2013), can easily be simulated. The effects of geometry modifications on fan performance can also be conveniently predicted (Kim et al., 2015).

In this study, the reverse flow in a squirrel-cage fan was investigated by CFD, and a non-axisymmetric D-type inlet nozzle was correspondingly designed to inhibit it. The installation direction of the new nozzle was selected by referring to the specific position where the back flow developed, and the fan performance of D-type nozzle schemes with different blockage extent was numerically compared. The suppression effect of the best scheme on the back flow was analyzed in detail, and the performance improvement under low-flow conditions was experimentally verified.

2. Geometric model

2.1. Fan model

The object of the present study was a small-sized double-suction squirrel-cage fan used in a range hood. Figure 1(a) shows the internal structure of the entire hood, at the center of which the test fan is installed. The square intake chamber is divided into two paths by the fan itself: the front path and the rear one. The intake airflow comes from below, respectively passing the front and rear intake chambers and finally entering the fan through the corresponding inlet nozzle. Additional components (such as the baffle and filter) are installed at the intake chamber entrance to collect and filter the cooking lampblack. These structures induce a non-symmetrical intake condition, so the rear intake chamber is correspondingly designed to be narrower than the front one. Together with the flow resistance of the built-in motor and its brackets, the proportion of the flow that passes through the rear path is only about 20% of the fan total flow rate. Therefore, in the following text, the flow structures in the front flow passage are mainly investigated by the flow-field analyzes.

The main parameters of the test fan are illustrated in Figure 1(b,c), and the corresponding values are listed in Table 1. The inner and outer diameter of the rotor were 210 mm and 254 mm, respectively. A single-arc blade...
profile was adopted, and the inlet and outlet blade angles were 75.6° and 160.4°, respectively.

Figure 2 shows the measured performance curves. The rotation speed \( n \) of the AC motor was artificially controlled according to the fan load under different working conditions. The rotation speed gradually increased as the fan volume flow rate \( Q_v \) decreased, from 920 RPM at the maximum flow rate to 1372 RPM at zero flow rate. Both the static pressure rise \( P_{st} \) and total pressure rise \( P_{TF} \) of the fan increased gradually as the fan volume flow rate decreased, and the maximum pressure rise was about 430 Pa at the zero flow rate point. As the fan flow rate increased, the input power \( W_0 \) increased gradually, and the total pressure efficiency \( \eta \) of the entire machine rose first and then decreased. The operation condition with the highest efficiency was named the best efficiency point (BEP), \( Q_{BEP} = 13.42 \text{ m}^3\text{·min}^{-1} \).

Figure 3 illustrates the discharge environments and the corresponding changes in each fan’s operating condition. To reduce air pollution, the cooking fumes in modern residential buildings must be collected and purified by an exhaust system, connecting all the range hoods in the same building unit. At mealtime, the simultaneous operation of a large number of hoods will congest the main pipe and increase the pressure inside (see Figure 3(a)), which makes the fans in each range hood shift to operate at a low flow-rate. As shown in Figure 3(b), the typical resistance curve of the exhaust system and the fan pressure curve intersect at point 1, which is located near the BEP. However, once the main is congested, the resistance curve increases, and the fan actually operates at point 2. Experimental results show that the flow rate of the fan’s actual operating condition may only be 70% of \( Q_{BEP} \) or even lower. Therefore, in order to overcome the increasing back-pressure in the discharge main and keep kitchen air clean, increasing the fan pressure under such low-flow conditions is one of the primary performance goals of these hood use fans.

2.2. Inlet nozzle

Figure 4 shows the detailed configuration of the prototype inlet nozzles. Each air entrance (front and rear) contains a nozzle shaped inlet but they are constructed differently. The front nozzle is a dividable component made of plastic (see Figure 4(a)), while the rear one is shaped by flanging the edge of the casing shell and is inseparable. The exit-section of both two inlet nozzles are round, the diameter of which is equal to the impeller’s inner diameter. The clearance between the nozzle end and the rotor shroud or hub is \( \delta = 0.02D_2 \).

As mentioned before, the intake air comes from a specified direction, so the flow-rate shows an obvious uneven distribution when flowing through the nozzle section. In the present study, a non-axisymmetric inlet nozzle is designed, with the shape and the installation method as shown in Figure 5. Compared to the prototype nozzle
Changes in the operating environment and working conditions of the test range hood fan: (a) Pressure changes in exhaust main; (b) Condition shifting of each range hood.

Figure 4. The prototype axisymmetric inlet nozzles: (a) Shape of the front inlet nozzle; (b) Assembly relationship between nozzles and impeller.

shown in Figure 4(a), the nozzle’s exit-section is cut by a straight line, and part of the arc profile is replaced by the cut line segment. The crescent-shaped area between the replaced arc and the cut line is blocked and no longer flowable. The remaining cross-sectional shape is similar to the D-shape, so the modified nozzle is called a D-type inlet nozzle. The distance between the cut line and the nozzle center, \( l_D \), obviously determines the shape of the D-type nozzle. The smaller it is, the smaller the flowable area will be. At the same time, since the D-type nozzle is no longer an axisymmetric one, the direction of installation matters. As shown in Figure 5(b), the normal direction of the cut line \( \theta_D \) is defined as the D-type nozzle’s installation direction. Additionally, the nozzle of the fan rear air inlet is also modified into a D-type according to the front nozzle’s shape. The exit-section shapes of the front nozzle and the rear one are mirrored symmetrically along the rotation axis.

3. Numerical method

3.1. Governing equations and turbulence modeling

Three-dimensional CFD simulations were performed in Fluent to obtain the flow field of the test fan. The steady flow field was simulated by solving in-compressible Reynolds-averaged Navier-Stokes (RANS) equations, the governing equations of which are as follows:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho U_i) = 0
\]

\[
\rho \left( \frac{\partial U_j}{\partial t} + U_k \frac{\partial U_j}{\partial x_k} \right) = - \frac{\partial P}{\partial x_j} + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial U_i}{\partial x_j} \right) + \frac{\partial}{\partial x_j} \tau_{ij}
\]
where \( U_i \) are averaged velocity components in the \( x_i \) direction, respectively \((i, j, k = 1, 2, 3)\). \( P \) is the averaged pressure. \( \rho , \ t \), and \( \mu \) are the time, air density, and the dynamic viscosity of the fluid. \( u'_i \) represents the velocity fluctuation in the \( x_i \) direction. \(-\rho u'_i u'_j \) is the Reynolds stress term, which could be related to the mean flow by a turbulent eddy viscosity, \( \mu_t \):

\[
\tau_{ij} = -\rho u'_i u'_j = \mu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \left( \mu_t \frac{\partial U_k}{\partial x_k} + \rho k \right) \tag{3}
\]

Shear Stress Transport (SST) \( k-\omega \) model (Menter, 1994) was adopted to calculate the turbulent viscosity. The \( k-\omega \) formulation was used near the wall, while the \( k-\varepsilon \) formulation was applied in the mainstream away from the solid-walls. SST model achieves a more accurate and more robust performance than the \( k-\omega \) or \( k-\varepsilon \) standard models, and was rated the most accurate model for aerodynamic applications in a NASA Technical Memorandum (Bardina et al., 1997). It was also proved to be accurate both in 3D flow calculation and fan performance predicting in the previous studies (Darvish & Frank, 2012; K. Wang et al., 2018). The SST \( k-\omega \) model computes the turbulence viscosity as a function of the turbulence kinetic energy (\( k \)) and the turbulence dissipation rate (\( \omega \)), the transport equations are (Ansys, 2011):

\[
\begin{align*}
\frac{\partial \rho k}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j k) &= \frac{\partial}{\partial x_j} \left[ \mu + \sigma_k \mu_t \right] \frac{\partial k}{\partial x_j} + \tau_{ij} S_{ij} - \beta^* \rho \omega k \tag{4} \\
\frac{\partial \rho \omega}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j \omega) &= \frac{\partial}{\partial x_j} \left[ \mu + \sigma_\omega \mu_t \right] \frac{\partial \omega}{\partial x_j} + C_{\omega \rho} \frac{\partial}{\partial x_j} \tau_{ij} S_{ij} - \beta \rho \omega^2 + 2(1 - f_i) \frac{\rho \sigma_\omega}{ \omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \tag{5}
\end{align*}
\]

where \( t, \rho, \) and \( u_j \) are the time, air density, and velocity components in the \( x_j \) direction, respectively.

The coefficients of the SST turbulence model \( \beta, C_{\omega \rho}, \sigma_k, \) and \( \sigma_\omega \) are obtained by blending the coefficients of the \( k-\omega \) model, denoted as \( \phi_2 \) with those of the transformed \( k-\varepsilon \) model (\( \phi_2 \)). The corresponding relation reads:

\[
\phi = f_1 \phi_1 + (1 - f_1) \phi_2 \tag{6}
\]

The SIMPLE algorithm was used to perform the pressure-velocity coupling. Second-order upwind discretization was adopted for convection terms and central difference schemes for diffusion terms.

### 3.2. Grid model and independent test

The entire fan flow field was divided into several flow zones to build a computing grid one by one (shown in Figure 6). The complex components (such as the baffle and filter) were also omitted in order to simplify the grid modeling process. The model consisted of hexahedral structured grids. Although the generation process of hexahedral grids was cumbersome, their quality was higher compared to unstructured grids, which helped to improve the calculation accuracy and reduce the computation time. Additionally, the wall grid was refined with an appropriate boundary layer to meet the \( y^+ \) requirements of SST \( k-\omega \) model.

The inlet boundary condition was set as the Mass flow rate. As the inlet boundary was further apart enough from the fan, the inflow velocity was assumed to be evenly distributed and perpendicular to the inlet. The intake airflow was also assumed to be fully developed, so the default turbulence quantities were adopted for its turbulence intensity and turbulence viscosity ratio, and were 5% and 10%, respectively. The outlet boundary was also fully extended, and no back flow occurred, so an Outflow boundary was adopted to simulate the free outlet condition. In the simulation, the rotational speed under each flow rate was fixed at 1200 RPM, which was the rotation speed at the BEP. The multiple reference frame (MRF) was employed to deal with the rotating impeller and exchange the information at the rotor-stator interface, which achieved a frame change across the interface without a relative position change (Ansys, 2011). This processing method is commonly used in CFD calculations of rotational machines (Öztürk et al., 2019; Zhu et al., 2020).

In the present study, various grid models with an increasing grids quantity from 3 million to over 22 million were built for grid independence validation. The evaluation metrics were the total pressure rise at the design point \( P_{TF,BEP} \), and the maximum volume flow rate \( Q_{max} \). The former was calculated based on the boundary conditions mentioned above. The latter represented the volume flow rate when the fan operated in an open discharge condition, which was predicted by using an averaged zero relative total pressure and averaged zero static pressure on the inlet and outlet boundary, respectively. Figure 7 illustrates how each metric varies with the grid quantity \( N \). It can be seen that the predicted \( P_{TF,BEP} \) is small when the grid is coarse and then gradually increases with the increase of grid resolution. Conversely, the predicted \( Q_{max} \) is falsely high initially and gradually decreases as the grid resolution improves. The red bar represents the ratio of the difference between the performance index calculated by the current grid scheme
Figure 6. Computation zone and grid model of the test range hood: (a) Computation zone; (b) Entire grid; (c) Rotor.

Figure 7. Independent grid validation using: (a) Total pressure rise at the best efficiency point $P_{T,F,BEP}$; (b) Maximum volume flow rate at zero backpressure $Q_{max}$.

Table 2. Grid quantity of each flow region (unit: million).

| Flow regions     | Volute       | Rotor(front/rear) | Volute       | Outpipe | Total        |
|------------------|--------------|-------------------|--------------|---------|--------------|
| Grid quantity    | 2.67         | 2.17/1.5          | 5.316        | 1.1     | 12.87        |

and the performance index calculated by the previous scheme according to the result of the current scheme. It represents the convergence process of the index with the number of grids. The change rate of both $P_{T,F,BEP}$ and $Q_{max}$ starts to reduce to below 0.1% after the grid quantity reaches 12.87 million and changes only slightly as with the grid quantity continues to increase. Considering both the calculation accuracy and the computational burden caused by a large number of grids, the 12.87 million cells grid model was selected as the final adopted model. The cell quantity of each flow region is listed in Table 2.

3.3. Performance experimental and verification

The fan aerodynamic performance was tested on an experimental platform referenced to international standard ISO5801-2007: Industrial fans – Performance testing using standardized airways. Figure 8 shows a schematic and photograph of the experimental facility. The operating condition of the fan was controlled by a throttling device. The static pressure below the cell straightener was sensed by averaging four pressure taps located 90° apart around the measuring duct.

The volume flow rate of the test fan $Q_v$ is calculated by the following equation:

$$Q_v = \alpha \frac{\pi}{4} d^2 \sqrt{\frac{2\Delta p}{\rho}}$$

(7)

where the parameters $\Delta p$, $\rho$, $\alpha$, $d$ are the static pressure obtained by the pressure taps, air density, the coefficient of the throttling device, and the diameter of the throttling device, respectively.

The transition piece installed between the tested fan and the air duct evens the fan outflow, so the flow entering the air duct can be assumed to be basically uniform. Therefore, the fan total pressure rise $P_{T,F}$, fan static pressure rise $P_{S,F}$, and total pressure efficiency $\eta$ of the test fan is calculated based on the experimental data by the
Figure 8. Aerodynamic performance experimental setup: (a) Schematic of the experimental system: 1, tested fan; 2, transition piece; 3, cross straightener; 4, diffusion duct; 5, cell straightener; 6, pressure taps; 7, throttling device. (b) Photograph of the experimental facility.

The following equation:

\[
\begin{align*}
&\frac{P_t}{F} = \Delta p + k \cdot \frac{1}{2} \left( \frac{Q_v}{A} \right)^2 \\
&P_s = P_t - \frac{1}{2} \rho \left( \frac{Q_v}{A} \right)^2 \\
&\eta = \frac{Q_v \cdot P_t}{W_0} \times 100\% \tag{8}
\end{align*}
\]

where \( A \) represents the area of the fan outlet, \( k \) is a constant determined by the area ratio between sections A and B, and \( W_0 \) is the overall fan input power.

Since the fan motor is AC, the rotation speed differs when the fan operating condition shifts. The fan performance in the following analysis was dimensionless to facilitate the description of the operating conditions at different flow rates. The flow coefficient \( \Phi \) and pressure coefficient \( \Psi_t, \Psi_s \) were calculated as:

\[
\begin{align*}
&\Phi = \frac{Q_v}{0.25\pi u_2 D_2^2} \\
&\Psi_t = \frac{P_t}{0.5\rho u_2^2} \\
&\Psi_s = \frac{P_s}{0.5\rho u_2^2} \tag{9}
\end{align*}
\]

where \( Q_v \) is the fan volume flow rate, \( P_t/F \) is the fan total pressure rise or static pressure rise, \( D_2 \) and \( u_2 \) are the impeller outer diameter and the tip speed at the impeller outlet, respectively.

The aerodynamic performance test was carried out in the test chamber with the ambient temperature of \((20 \pm 5)\, ^\circ C\), a relative humidity of no more than 85% and no external airflow and thermal radiation. The testing was not started until the fan was preheated for over 1 h, and then the fan’s operating condition was adjusted by switching the throttling devices. The measurement at each condition lasted for at least 5 min, to ensure that the fan flow field was stable. The performance data were averaged every 10 s, and the recorded data were further averaged by 2 min. The uncertainty of the measured total pressure at most of the conditions was less than 1%, while the efficiency uncertainty was less than 0.5%.

Figure 9(a) shows the pressure curves of the entire machine. The dashed line shows the predicted pressure curves obtained by CFD, which are also dimensionless. In the working condition area where the flow rate is greater than \( \phi_{BEP} \), the calculated value is in good agreement with the test curve, and the relative deviation of the static pressure at the same flow rate is within 3%. However, when the flow rate is lower than \( \phi_{BEP} \), the deviation increases and exceeds 5% under the 0.6\( \phi_{BEP} \) condition.

As stated in Equation (8), the test efficiency \( \eta \) is the total pressure efficiency of the entire machine, which contains the motor efficiency \( \eta_{motor} \). However, the motor efficiency is not counted in the CFD simulation, so the predicted efficiency is actually the fan efficiency \( \eta_1 \). In order to compare the two, the experimental fan efficiency could be calculated by removing the motor efficiency from the machine efficiency:

\[
\eta_1 = \eta / \eta_{motor} \tag{10}
\]

Figure 9(b) compares the predicted and experimental fan efficiency. It can be observed that the predicted efficiency is lower than the test at a flow rate larger than the BEP, while higher than the test at a flow rate lower than the BEP. The changing trends of the numerical and experimental results are close, and the maximum efficiency deviation between them is about 3 percentage.

The deviation between predicted and experimental results was likely induced by the geometric differences between the calculation model and the actual fan, as well as the inherent error of performance testing and CFD simulations. Despite these deviations, the changing trend of calculated performance curves was still consistent with the experimental data. Therefore, the grid model and numerical method in this study can be considered reliable, and the results can be considered credible.
4. Results, analysis, and discussion

4.1. Reverse flow at low-flow conditions

Figure 10 shows the radial velocity distribution at the impeller inlet under different operating points. The velocities are also nondimensionalized by the tip speed of the blade’s trailing edge $u_2$. It is illustrated that the radial velocity generally decreases as the fan flow rate decreases, especially near the volute tongue. There is an area where the radial velocity is negative at the tongue, which means the reverse flow occurs there. At large-flow $1.4\phi_{BEP}$ condition, a slight reverse flow develops only between 40–60°, and is mainly located near the rotor shroud. As the fan flow rate continues to decrease, the reverse region expands along the axial and circumferential direction. When the fan flow rate decreases to $0.6\phi_{BEP}$, almost the entire impeller inlet between 0–60° enters into a back flow state.

Figure 11 shows the flow state of the reverse flow after returning to the impeller intake zone at corresponding operating conditions. This part of airflow enters the impeller the second time someplace downstream, forming a cross-impeller flow (CIF) structure. In large-flow conditions, CIF mainly occurs in the shroud-side of the rotor and occupies about 1/6 of the impeller in the circumferential direction. The span of the CIF increases gradually as the fan flow rate decreases. As shown in Figure 16(c), the span of the CIF is significantly enhanced at the flow rate of $0.6\phi_{BEP}$, and more than half of the impeller is blocked. Even part of the CIF passes in reverse through the inlet nozzle’s exit-section and flows back to the intake chamber.

This CIF further affects the airflow leak through the clearance between the rotor shroud and the inlet nozzle. Figure 12 shows the streamlines of leakage airflow at both BEP and low-flow $0.6\phi_{BEP}$ conditions. At the BEP, a slight leakage occurs at most of the impeller’s circumferential position and returns to the impeller quickly after reflecting around the rotor shroud (see streamline A1). The closer to the volute tongue, the more severe the back flow through the clearance. At this point, a local jet is generated and will flow across multiple blades along the direction of rotation, approaching the hub-side of the impeller (see streamline A2). The leakage flow closest to the volute tongue then flows back to the impeller quickly (see streamline A3). This flow state demonstrates significant changes at the low-flow $0.6\phi_{BEP}$ point. As more than half of the impeller entrance is occupied and blocked by the CIF, the leakage airflow can’t return to the rotor in time and will pass more passages before finally returning into the impeller (see streamline B2, B3). Part of the jet leakage-flow through the clearance near the tongue will have no choice but to blow into the inlet chamber in reverse along the surface of the inlet nozzle (see streamline B4).

4.2. Design of D-type nozzle

Figure 13 shows the distribution of the velocity component $u_z$ on the end-section of the front nozzle along the
Figure 11. Flow pattern of the cross-impeller flow with a flow rate of: (a) 1.4\(\phi_{BEP}\), (b) \(\phi_{BEP}\), and (c) 0.6\(\phi_{BEP}\).

Figure 12. Flow pattern of the clearance leakage airflow with a flow rate of: (a) \(\phi_{BEP}\), and (b) 0.6\(\phi_{BEP}\).

circumference of \(R = 0.9R_{nozzle}\) under different working conditions. For the front air intake path, a negative \(u_z\) represents an inward flow direction, and a positive \(u_z\) represents an outward flow direction. The velocity distribution of each working condition is similar: the velocity between 180–360\(^\circ\) is relatively high and inward as the bottom of the nozzle is closer to the entrance of the range hood, while the top-side has a lower velocity, especially between the tongue location 60\(^\circ\) and 180\(^\circ\). When the fan flow rate declines to lower than \(\phi_{BEP}\), positive \(u_z\) appears between 120–180\(^\circ\), which means reverse blowing develops here. The curves of both \(\phi_{BEP}\), 0.8\(\phi_{BEP}\), and 0.6\(\phi_{BEP}\) illustrate a peak around 150\(^\circ\), representing the predominant position of the reverse flow. Therefore, the installation direction of the D-type inlet nozzle \(\theta_D\) is selected as 150\(^\circ\), so that its shielding area can prevent reverse air flow from passing through this area.

The distance between the cut line and the center \(l_D\) is the other critical parameter that needs to be selected for determining the final shape of the D-type nozzle. Nozzle schemes with \(l_D/R_{nozzle}\) varying from 0.9 to 0.3 were modeled, and the flow state of each scheme at low-flow 0.5\(\phi_{BEP}\) condition was calculated. Each scheme’s back flow ratio was counted by dividing the reverse mass flow by the mass flow positive pass through the nozzle’s exit-section. Figure 14 illustrates how the back flow ratio varies with the \(l_D/R_{nozzle}\) and that the back flow region of scheme \(l_D/R_{nozzle} = 0.9, 0.7, \) and 0.5.

Figure 13. Velocity distribution at the nozzle-section at a relative radius \(R = 0.9R_{nozzle}\).

As the \(l_D/R_{nozzle}\) decreases, the area of the new nozzle’s blockage area expands, while the flowable throat area shrinks correspondingly. It can be observed that the back flow ratio decreases gradually with the reduction of \(l_D/R_{nozzle}\). When \(l_D/R_{nozzle} = 0.9\), the nozzle’s exit-section shape changes little, and the back flow is still severe; when the \(l_D/R_{nozzle}\) reduces to 0.7, the back flow area decreases significantly, looks like being ‘cut’ by the new nozzle. Finally, when the \(l_D/R_{nozzle}\) reaches 0.5, the back flow area is almost entirely eliminated, and the back flow ratio reaches the lowest at almost zero. A slight
reverse blowing phenomenon re-appears if the $l_D/R_{nozzle}$ continues decreasing, and the optimal $l_D/R_{nozzle}$ seems to be 0.5 from the perspective of controlling this back flow.

However, a small throat-section area may restrict the fan performance under large-flow conditions. Each scheme’s fan pressure and efficiency under three working conditions: low-flow $0.4\phi_{BEP}$, best efficiency point, and large-flow $1.6\phi_{BEP}$ were calculated by CFD. The results are shown in Figure 15. On the one hand, the fan pressure under $\phi_{BEP}$ and $1.6\phi_{BEP}$ conditions decreases directly, but the drop is nearly imperceptible when $l_D/R_{nozzle} > 0.7$. At the same time, the fan pressure under the low-flow $0.4\phi_{BEP}$ does not drop, but gradually rises to the maximum when $l_D/R_{nozzle} = 0.6$. However, the fan pressure of $0.4\phi_{BEP}$ will also inevitably decrease if the area of the nozzle’s exit-section continues to shrink. On the other hand, the D-type nozzle negatively affects the fan efficiency in all flow rates. The efficiency of each working condition gradually decreases with the reduction of $l_D/R_{nozzle}$. However, similar to the change trends of the fan pressure, the efficiency drop is not apparent when the blockage area is small, and it only declines rapidly when $l_D/R_{nozzle} < 0.7$. In summary, the D-type inlet nozzle with $l_D/R_{nozzle} = 0.7$ was selected as the comprehensive best design, which achieved a significant improvement in fan pressure under low-flow conditions with minimal reduction to the performance of other operating points.

## 4.3. Effects of D-type inlet nozzle

The final D-type nozzle was $l_D/R_{nozzle} = 0.7$, which was installed in the direction of $\theta_D = 150^\circ$. Figure 16 shows the back flow ratio curve at both the fan front and rear entrance, before and after installing the D-type inlet nozzle. It can be observed that the back flow ratio is reduced under all working conditions, especially when the reverse blowing phenomenon is severe. In the prototype fan, the back flow is aggravated suddenly at $0.6\phi_{BEP}$ condition, and the back flow ratio increases significantly from less than 4% to 13.4% at $0.4\phi_{BEP}$. After installing the new inlet nozzle, the back flow ratio at $0.6\phi_{BEP}$ is only 0.5%, and that of $0.4\phi_{BEP}$ is 5.4%, lower than half of the prototypes. The reverse blowing in the fan rear inlet chamber appears to develop earlier than that in the front chamber. Severe back flow begins to develop once the fan flow rate decreases to less than $\phi_{BEP}$; the back flow ratio increases correspondingly and finally reaches 122% at $0.4\phi_{BEP}$. After installing the new nozzle, the operating point where the severe back flow starts to develop is delayed to $0.8\phi_{BEP}$, and the upward trend of the back flow ratio is not as sharp as before. The back flow ratio under $0.4\phi_{BEP}$ is reduced to 95%.

Figure 17 shows the streamline of the front CIF at the $0.4\phi_{BEP}$ working condition, before and after the installation of the new nozzle. The D-type nozzle shields the region between $120^\circ$ and $180^\circ$ where the CIF is most likely to blow into the inlet chamber, and all the CIF are contained in the fan. Figure 18 shows the corresponding streamlines of the clearance leakage flow. Although the CIF still exists and blocks a large span of the impeller entrance, making the leakage airflow difficult to return

![Figure 14](image1.png)  
**Figure 14.** The nozzle back flow ratio $\delta_{nozzle}$ and back flow area changes with decreasing $l_D/R_{nozzle}$.

![Figure 15](image2.png)  
**Figure 15.** Effect of $l_D/R_{nozzle}$ on fan (a) pressure, and (b) efficiency under different operating conditions.
the impeller, there is nearly no CIF reverse flow and leakage airflow outward. Therefore, only a small part of the leakage airflow blows in reverse into the intake chamber. As shown in Figures 17(a) and 18(a), the traditional axisymmetric nozzle fails to restrain this reverse blowing, and large-scale reverse airflow (both CIF and leakage flow) develops and forms a vortex in the inlet chamber. After installing the new D-type inlet nozzle, the back flow at the inlet nozzle is significantly inhibited and no vortex can be observed in the inlet chamber.

Q criterion (Hunt et al., 1988) is used to illustrate the three-dimensional vortex structure in the inlet chamber:

\[ Q = -\frac{1}{2} (S_{ij}S_{ij} - \Omega_{ij}\Omega_{ij}) \]  

(11)

where:

\[ S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right), \quad \Omega_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right). \]

The distribution of lumps of the Q iso-surface in the inlet chamber of the prototype and modified fan \((Q = 2 \times 10^4)\) are indicated in Figure 19. The lumps are colored by the dimensionless velocity. In the prototype fan, jet flow enters the inlet chamber reversely near the position of 150°, carrying a large number of small-scale vortex lumps. This reverse blowing pushes the flow field in the intake chamber to rotate and form a large vortex near the entrance center (as shown in Figure 19(a)). As the reverse airflow is blocked in the fan by the D-type inlet nozzle, the center vortex disappears and the small lumps around it also decrease.

The modified nozzle was manufactured (as shown in Figure 20), and the hood performance with the new nozzle installed was measured under the same test environment. The performance comparison between the fan with the prototype nozzle and the new nozzle is shown in Figure 21. The total pressure slightly decreased by about 1% at the BEP and decreased by less than 3% at the maximum flow rate. As compensation, the fan pressure improvement at the low-flow conditions was far more significant. When the flow rate was less than 0.8φ_{BEP}, the pressure of the fan installed with the new nozzle was higher than that of the prototype. The total pressure increment at the zero flow rate point exceeded 6%. Additionally, the fan efficiency at each working condition decreased as expected; the drop did not exceed 1%, which was acceptable.

The test fan of this study operated in an inlet chamber, and the air came from the bottom-side direction.
Therefore, the airflow passing the top-side of the nozzle’s exit-section was inherently smaller, which was why the upward installed D-type nozzle had little effect on the fan performance under BEP and large-flow conditions.

5. Conclusion

In this study, the internal flow field in a squirrel-cage fan was numerically studied using the SST $k$-$\omega$ turbulence model based on the RANS method. It was found that airflow passed the blade channels near the volute tongue in reverse and then re-entered the impeller downstream, forming a unique cross-impeller flow structure in the rotor center. With the decrease of the fan flow rate, the span and intensity of this cross-impeller flow gradually intensified, carried the clearance leakage airflow, and finally blew into the inlet chamber in reverse near the circumferential position of $\theta = 150^\circ$.

A non-axisymmetric D-type inlet nozzle was proposed to manage the reverse flow under low-flow conditions. The D-type nozzle’s installation direction was determined by referring to the position where reverse blowing developed, and several schemes with different extents of blockage were numerically compared. By installing the D-type nozzle with $l_D/R_{\text{nozzle}} = 0.7$ along the direction of $\theta_D = 150^\circ$, the reverse blowing mixed by the cross-impeller flow and the clearance leakage airflow was significantly inhibited, and the vortex in the inlet chamber disappeared correspondingly. Experimental data illustrated that this newly designed inlet nozzle improved the fan pressure under low-flow conditions by about 6% and barely reduced fan performance at other working points.

The present study reveals the back flow evolution mechanism in a squirrel-cage fan, which provides a reference for the performance improvement of fans operating at a relatively low flow rate. The successful application of a D-type inlet nozzle illustrates that non-axisymmetric components provide potential performance advantages under off-design conditions. However, the reverse blowing that develops under low-flow conditions is essentially a transient flow structure. Transient flow field simulations...
or experiments should be carried out in the future to study the unsteady flow around the fan’s inlet nozzle and the effects generated by the new D-type inlet nozzle. In addition, whether this D-type nozzle can still achieve the pressure improvement demonstrated in this work on a fan operates in an open-air intake environment is also worthy of further investigation.

Acknowledgments
The authors acknowledge the support of National Key Research and Development Program of China No. 2018YFB0606101, and all the other scholars for their advices in the process of improving this article.

Disclosure statement
No potential conflict of interest was reported by the author(s).

Funding
This research was supported by National Key Research and Development Program of China (Grant number 2018YFB0606101).

ORCID
Boyan Jiang http://orcid.org/0000-0001-8802-7993

References
Adachi T., Sugita N., & Yamada Y. (2004). Study on the performance of a sirocco fan (flow around the runner blade). International Journal of Rotating Machinery, 10(5), 415–424. https://doi.org/10.1155/S1023621X04000417

Akbari G., Montazerin N., & Akbarizadeh M. (2012). Stereoscopic particle image velocimetry of the flow field in the rotor exit region of a forward-blade centrifugal turbomachine. Proceedings of the Institution of Mechanical Engineers Part A: Journal of Power and Energy, 226(2), 163–181. https://doi.org/10.1177/1350650111423459

Akimoto R., Nakagawa N., Kawahata T., Kondo F., & Noyama H. (1996). Development of compact air conditioning unit for vehicles. Mitsubishi Heavy Industries Technical Review, 33(3), 125–129.

Ansys I. (2011). ANSYS FLUENT user’s guide.

Bardina J. E., Huang P. G., & Coakley T. J. (1997). Turbulence modeling validation, testing, and development. NASA Technical Memorandum.

Darvish M., & Frank S. (2012). Toward the CFD simulation of sirocco fans: From selecting a turbulence model to the role of cell shapes. In International conference on fan noise, technology and numerical methods (FAN 2012), Senlis, France, April (pp. 18–20).

Eck B. (1973). Fans: design and operation of centrifugal, axial-flow, and cross-flow fans. Pergamon.

Elfarra M. A. (2019). Optimization of helicopter rotor blade performance by spline-based taper distribution using neural networks based on CFD solutions. Engineering Applications of Computational Fluid Mechanics, 13(1), 833–848. https://doi.org/10.1080/19942060.2019.1648322

Fukutomi J. E. A., Kuwachi T., Itabashi A., Nakase Y., & Kuwada T. (1999). A study of sirocco fan with eccentric inlet nozzle. Transactions of the Japan Society of Mechanical Engineers Series B, 65(634), 2023–2029. https://doi.org/10.1299/kikaib.65.2023

Gholamian M., Rao G. K. M., & Bhamara P. (2013). Numerical investigation on effect of inlet nozzle size on efficiency and flow pattern in squirrel cage fans. Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy, 227(8), 896–907. https://doi.org/10.1177/0957650913504566

Gholamian M., Rao G. K. M., & Panitapu B. (2013). Effect of axial gap between inlet nozzle and impeller on efficiency and flow pattern in centrifugal fans, numerical and experimental analysis. Case Studies in Thermal Engineering, 1(1), 26–37. https://doi.org/10.1016/j.csite.2013.08.003

Gyeong R., Kawahashi M., Hirahara H., & Kitadume M. (2015). Application of stereoscopic particle image velocimetry to experimental analysis of flow through multiblade fan. JSME International Journal Series B, 48(1), 25–33. https://doi.org/10.1299/jsmeb.48.25

He L., & Sato K. (2001). Numerical solution of incompressible unsteady flows in turbomachinery. Journal of Fluids Engineering, 123(3), 680–685. https://doi.org/10.1115/1.1383595

Hunt J. C., Wray A. A., & Moin P. (1988). Eddies, streams, and convergence zones in turbulent flows. In Proceedings of the 1988 summer program, center for turbulence research report, CTR-S88, Stanford.

Jiang B., Liu H., Li B., & Wang J. (2018). Effects of cut volute profile on squirrel cage fan performance and flow field. Advances in Mechanical Engineering, 10(3), 1–14. https://doi.org/10.1177/1687814018766915

Kadota S., Kawaguchi K., Suzuki M., Matsui K., & Kikuyama K. (1994). Experimental study on low-noise multiblade fan. 1st report, visualization of three-dimensional flow between blades. Transactions of the Japan Society of Mechanical Engineers Series B, 60(1), 102–107.

Kim J. S., Jeong U. C., Kim D. W., Han S. Y., & Oh J. E. (2015, March). Optimization of sirocco fan blade to reduce noise of air purifier using a metamodel and evolutionary algorithm. Applied Acoustics, 89, 254–266. https://doi.org/10.1016/j.apacoust.2014.10.005

Kind R. J., & Tobin M. G. (1990). Flow in a centrifugal fan of the squirrel-cage type. Journal of Turbomachinery, 112(1), 84–90. https://doi.org/10.1115/1.2927426

Kitadume M., Kawahashi M., Hirahara H., Uchida T., & Yanagawa H. (2007). Experimental analysis of 3D flow in scroll casing of multi-blade fan for air-conditioner. Journal of Fluid Science and Technology, 2(2), 302–310. https://doi.org/10.1299/jfst.2.302

Menter F. R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. AIAA Journal, 32(8), 1598–1605. https://doi.org/10.2514/3.12149

Montazerin N., Damangir A., & Mirian S. (1998). A new concept for squirrel-cage fan inlet. Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy, 212(5), 343–349. https://doi.org/10.1243/0954405981515950
Montazerin N., Damangir A., & Mirzaie H. (2000). Inlet induced flow in squirrel-cage fans. *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy, 214*(3), 243–253. https://doi.org/10.1243/1350650001543142

Oro J. M. F., García B. P., González J., Díaz K. M. A., & Velarde-Suárez S. (2013). Numerical methodology for the assessment of relative and absolute deterministic flow structures in the analysis of impeller-tongue interactions for centrifugal fans. *Computers & Fluids, 86*, 310–325. https://doi.org/10.1016/j.compfluid.2013.07.014

Öztürk İ., Çetin C., & Yavuz M. M. (2019). Effect of fan and shroud configurations on underhood flow characteristics of an agricultural tractor. *Engineering Applications of Computational Fluid Mechanics, 13*(1), 506–518. https://doi.org/10.1080/19942060.2019.1617192

Qi D. T., Pomfret M. J., & Lam K. (1996). A new approach to the design of fan volute profiles. *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science, 210*(3), 287–294. https://doi.org/10.1243/PIME_PROC_1996_210_198_02

Raj D., & Swim W. B. (1981). Measurements of the mean flow velocity and velocity fluctuations at the exit of an FC centrifugal fan rotor. *Journal of Engineering for Power, 103*(2), 393–399. https://doi.org/10.1115/1.3230733

Ramezanizadeh M., Alhuyi Nazari M., Ahmadi M. H., & Chau K. W. (2019). Experimental and numerical analysis of a nanofluidic thermosyphon heat exchanger. *Engineering Applications of Computational Fluid Mechanics, 13*(1), 40–47. https://doi.org/10.1080/19942060.2018.1518272

Wang K., Ju Y., & Zhang C. (2018). Numerical investigation on flow mechanisms of squirrel cage fan. *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy, 233*(1), 3–16. https://doi.org/10.1177/0957650918773932

Wang J. B., & Ou Y. D. (2004). Effect of inlet on the performance of multi-blade centrifugal fan. *Fluid Machinery, 32*(10), 22–25. https://doi.org/10.3969/j.issn.1005-0329.2004.10.006

Wang J., Ou Y., & Wu K. (2005). Analysis on the vortices flow in the blade passages of the fan for air-conditioner and the influence of the eccentric fan inlet. *Journal of Engineering Thermophysics, 26*(6), 951–953. https://doi.org/10.13321/j.issn:2053-231X.2005.06.016

Wen X., Qi D., Mao Y., & Yang X. (2013). Experimental and numerical study on the inlet nozzle of a small squirrel-cage fan. *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy, 227*(4), 450–463. https://doi.org/10.1177/0957650913487530

Yamazaki S., & Satoh R. (1986). An experimental study on the aerodynamic performance of multi-blade blowers: 1st report, measurement of flow patterns within the blowers. *Transactions of the Japan Society of Mechanical Engineers Series B, 52*(484), 3987–3992. https://doi.org/10.1299/kikaib.52.3987

Yamazaki S., & Satoh R. (1987). An experimental study on the aerodynamic performance of multi-blade blowers: 2nd report, prediction of the shaft power. *Transactions of the Japan Society of Mechanical Engineers Series B, 53*(485), 108–113. https://doi.org/10.1299/kikaib.53.108

Zhu D., Xiao R., Yao Z., Yang W., & Liu W. (2020). Optimization design for reducing the axial force of a vaned mixed-flow pump. *Engineering Applications of Computational Fluid Mechanics, 14*(1), 882–896. https://doi.org/10.1080/19942060.2020.1749933