Numerical study of the effect of inner guide vanes on performance of multi-blade centrifugal fan of range hood

Dongfang Zhao, Ruoning Chen and Xue-yi You*
Tianjin Key Lab of Indoor Air Environmental Quality Control Department, School of Environmental Science and Engineering, Tianjin University, Tianjin 300350, China
*Corresponding author’s e-mail: xyyou@tju.edu.cn

Abstract. This paper focus on the effect of inner guide vanes on the aerodynamic performance of multi-blade centrifugal fan (MCF) used to range hood. The Reynolds-averaged Navier-Stokes equations with the RNG k-ε turbulence model are discretized and solved by Fluent. A detailed comparison of dimensionless characteristic coefficients of flow, pressure and power, between the origin MCF and the MCF with inner guide vanes is conducted for a full exhaust condition. The results suggest that the MCF with inner guide vanes perform better in terms of both flow and pressure coefficient. The exhaust airflow rate increases by 6.4% at working condition of No. 6, and there is a best efficiency point for the MCF with inner guide vanes at flow and pressure coefficient of 0.25 and 1.05, respectively. By comparing the flow field between the origin MCF and the MCF with inner guide vanes, it is found that the inner guide vanes effectively reduce flow loss near volute tongue areas.

1. Introduction
Multi-blades centrifugal fan (MCF) is widely used, especially in the field of range hood, for its excellent exhaust airflow rate compared to axial fan. However, some MCFs are inadequate to cater for the required enhanced performance as a result of increased system resistance. Therefore, structure parameters optimization of MCF to improve aerodynamic performance has become a focus of research. Meanwhile, China promulgated relevant standards, such as GB 18483-2001 (Emission standard of cooking fume), for catering industry and GB/T 17713-2011 (Range hood), for domestic. In this context, the investigation of MCF performance improvement has already become extremely important.

Volute is one of the key components for MCF, and its function is to collect fluid from impeller and transfer to the delivery pipe. Hariharan and Govardhan [1] proposed parallel wall volutes for industrial centrifugal blower, and predicted the aerodynamic performance by simulation method. It was found that the performance with parallel wall volutes is improved up to 6%, compared with the traditional rectangular volutes. Patil et al. [2] investigated the influence of volute tongue clearance on performance of centrifugal blower. It was shown that when volute tongue clearance decreases from 12.5% to 6% of impeller diameter, the total pressure and efficiency increases by 19.52% and 21.90%, respectively. In addition, Baloni et al. [3] adopted Taguchi method and ANOVA approach to optimize the structure parameters of centrifugal blower volute. It was found that 1.5 time width of impeller, 24° tongue angle and 10% reduction in volute outer radial locations show better performance compared with the original centrifugal volute.

Moreover, Li et al. [4] studied the effect of impeller enlargement by blades tip extension on centrifugal fan performance through the means of numerical simulation and experiment. The results showed that the enlarged impeller is able to lead low uniformity of flow field and increase the volute
loss. Abanto and Reggio [5] used commercial CFD software to predict flow field of domestic kitchen hood. It was found that the flow and load coefficients are shown to be in good agreement with the experimental data. Son et al. [6] investigated the effect of ratio between inlet radius and bell mouth of centrifugal blower on the flow rate by numerical simulation. It was found that the optimal ratio is about 0.86 to improve centrifugal blower flow rate, and it is best to have the ratio at about 0.09.

Most of the current studies are about structure parameters of volute, impeller, annular plate height, bell mouth. However, few research works have been reported the effect of inner guide vanes on MCF performance. In addition, numerical method has been widely used to study flow field and structure optimization for low cost and high efficiency [7-8]. Thus, the present study explores numerically the effect of guide vanes at impeller outlet on aerodynamic performance of MCF for range hood. The overall performance and local behaviours of the flow field inside MCF are studied. In addition, the better guide vane has been suggested by assessing the aerodynamic performance both origin MCF and MCF with guide vanes.

2. Numerical simulation

In this study, several appropriate assumptions are made: (1) the air is treated as incompressible and Newtonian fluid; (2) The effects of temperature on the flow field are neglected for small temperature gradient inside MCF for range hood. Since air flow is high turbulence, the commercial CFD software Fluent is used to solve the three-dimensional incompressible Navier-Stokes equations. The mesh generation is carried out by Gambit.

2.1. Mathematical model

The RNG k-ε model is used to simulate the three-dimensional airflow field as its high accuracy for swirling flows [9-10]. The governing equations for incompressible fluid, including the equations of continuity, momentum and turbulence model are expressed as the general form [11-13]:

$$\frac{\partial}{\partial t}(\varphi) + \nabla \cdot (u \varphi) = \nabla \cdot (\Gamma_\varphi \nabla \varphi) + S_\varphi$$

(1)

where \( \varphi \) represents three velocity components, \( u, v, w \), the kinetic energy of turbulence \( k \), the dissipation rate of kinetic energy \( \varepsilon \). When \( \varphi=1 \), the equation becomes the continuity equation. \( u \) is velocity vector, \( \Gamma_\varphi \) is effective diffusion coefficient for each dependent variable and \( S_\varphi \) is source term. The coefficients of turbulence model are \( C_\mu=1.424, C_\nu=1.38, C_\rho=0.0845, \alpha_\varepsilon=0.07 \).

The moving reference frame approach is used for coupling between the impeller and the adjacent fluid zone with interface boundary condition used to wall information exchange. The inlet and outlet are set to pressure conditions. No slip conditions are used at the walls and the standard wall functions are employed at the near-wall cells. The Semi-Implicit Method for Pressure Linked Equations algorithm is adopted to solve the discrete equations.

2.2. Mesh independence analysis and model validation

The MCF for domestic range hood consists of impeller and volute generated by logarithmic law, and its structural details are presented in Table 1. Electrical motor, oil filter and outside shape are not considered in this investigation. The length of the inlet duct is set to 4 times the diameter of the inlet in the process of simulation. The cell numbers of the grid independence test are 1.17, 1.66, and 2.22 million. In figure 1, several sampled points are extracted in blade outlet zone, and their velocity magnitudes are compared. Figure 2 shows the effect of the grid size on velocity distribution at blade outlet. It can be noted that the variations of velocity magnitudes are less than 4% compared to that of 2.22 million cells, when the cell number is 1.66 million. Therefore, the grid with 1.66 million cells is selected as the optimal grid system for numerical study.
Table 1. Structure parameters of the MCF.

| Parameter                | Value  |
|--------------------------|--------|
| Impeller inlet diameter  | $D_1$ 210 mm |
| Impeller outlet diameter | $D_2$ 250 mm |
| Inlet blade angle        | $\beta_1$ 79°   |
| Outlet blade angle       | $\beta_2$ 163° |
| Number of blades         | $z$ 60 |
| Impeller width           | $b$ 146 mm |
| Volute width             | $H$ 180 mm |

The experiment is completed in accordance with Chinese standard GB/T 17713-2011 for air performance. The comparisons of exhaust airflow rate ($q_V$) under different working conditions are shown in figure 3. Each of the working conditions is characterized by an orifice plate with specific diameter. The simulation data is basically consistent with the experimental result [14] and this means the numerical model is feasible to predict the flow characteristics of the MCF. However, there is relatively large deviation between simulation and experiment at working condition of No. 7 to 10. This is because the resistance of shape and inlet filter is neglected.

3. Result and discussion

Numerical simulations are performed at original MCF (OMCF) and MCF with inner guide vanes (MCFG). The results are discussed based on performance curves. Flow field inside the blower is visualized using contours and vectors. The dimensionless characteristic coefficients used in this study define as followings:

- Flow coefficient

$$\phi = 240q_v \left( \pi^2 n D_2^2 \right)^{-1}$$

- Pressure coefficient

$$\psi = 3600 \Delta p \left( \rho \pi^3 n^3 D_2^3 \right)^{-1}$$

- Power coefficient

$$\lambda = 2.88 \times 10^4 M \left( \rho \pi^3 n^3 D_2^3 \right)^{-1}$$

where $q_v$ is the exhaust airflow rate, m$^3$/s; $\Delta p$ is the total pressure of centrifugal fan, Pa; $M$ is the torque of impeller, Nm; $\rho$ represents the fluid density, kg/m$^3$; $n$ represents the rotational speed of impeller, rpm; $D_2$ is the outlet diameter of impeller, m.
3.1. Structure of guide vanes
The guide vanes are installed at blade outlet and arranged in a ring with even space. The space span of guide vanes is 225° as in figure 4. Generally, the inlet angle of guide vane is equal to $\beta_2$, considering the effect of vortex separation along the blade tip, there has 3° angle of attack as in equation (5).

$$\beta_1 = 180° - (\beta_2 - 3°) = 20°$$

Figure 4. The structure of inner guide vanes.

$$\beta_1 = 180° - (\beta_2 - 3°) = 20°$$

3.2. Performance of multi-blades centrifugal fans
The performance curves of the OMFC and MCFG are plotted in figure 5 and 6. The simulation result indicates that the inner guide vanes have significant effect on the flow coefficient of MCF. The growth rate is 6.4% at working condition of No. 6 (the number of 6 orifice plate with pore diameter of 140 mm). This phenomenon can be explained from the structural feature of the angle of attack. In a certain range, the lift coefficient increases with the increasing of angle of attack. Therefore, proper measurement is thus proposed to improve the exhaust airflow rate of MCF, especially for low system resistance condition. From figure 6, it can be found that the MCFG achieves higher level of $\Psi$ than that of the OMCF. Moreover, the results also indicate that there is one optimal operating point in the range of working flow rate. The best efficiency point at working condition of No.5 corresponds to $\Phi=0.25$ and $\Psi=1.05$. With the increasing of $\Phi$, the $\lambda$ of MCFG and OMCF increases simultaneously. This is because impeller shift power of centrifugal fan depends on the quantity of airflow rate.

To obtain an overall understanding the influence of inner guide vanes, flow visualization is carried out numerically at meridian and near volute tongue, as shown in figure 7 and 8. It can be seen that the inner guide vanes lead to some flow separation at A, B, C zone. This means improper application for guide vanes will reduce MCF efficiency to a certain extent. In zone D for OMCF, the velocity magnitude is larger than neighbouring areas, and the swirling flow leads the fluid could not pass through the impeller. Although this flow pattern is modified by inner guide vanes, it appears reverse flow in zone D for MCFG, and results into loss of energy and poor performance. However, it is found that the velocity is almost parallel to the wall vector of volute tongue in zone E of OMCF. This could be the reason that the OMCF has relatively low efficiency. The method of inner guide vanes
successfully changes the flow direction near volute tongue. It makes the fluid discharged from impeller possesses relatively large tangential velocity and reduces the flow loss. Therefore, there are some inappropriate zones for inner guide vanes, but the performance improvement of flow field near volute tongue is good for MCF. Form figure 8, it is found that the poor uniformity of velocity distribution occurs in zone F for the MCFG. Moreover, it can be seen that the velocity magnitude becomes very large in zone G for the OMCF. This is strong possibility of formation the new noise source. However, this situation is improved by inner guide method as shown in figure 8(b).

Figure 7. Vector of velocity in flow field at meridian and near volute tongue.

Figure 8. Velocity magnitude distribution at meridian (m/s).

4. Conclusion
The present paper numerically studies the effect of inner guide vanes on performance of the multi-blades centrifugal fan for domestic range hood. The results show that the MCFG exhibits higher dimensionless characteristic coefficients of flow, pressure and power compared to the OMCF at a full exhaust condition. The exhaust airflow rate increases by 6.4% at working condition of No. 6. There is a best efficiency point for the MCFG at $\phi=0.25$ and $\Psi=1.05$. In addition, the inner guide vanes effectively reduce flow loss near volute tongue areas. The comparison of the results of velocity field indicates that the inner guide vanes method is a good way to improve the MCF performance.

Acknowledgments
The research was financially supported by the National Key R&D Program of China through grant No. 2017YFC0211500.

References
[1] Hariharan C, Govardhan M. (2016) Improving performance of an industrial centrifugal blower with parallel wall volutes. Applied Thermal Engineering, 109: 53–64.
[2] Patil S.R., Chavan S.T., Jadhav N.S., Vadgeri S.S. (2018) Effect of Volute Tongue Clearance Variation on Performance of Centrifugal Blower by Numerical and Experimental Analysis. Materials Today: Proceedings, 5: 3883-3894.
[3] Baloni B, Pathak Y, Channiwala S. (2015) Centrifugal blower volute optimization based on Taguchi method. Computers and Fluids, 112: 72-78.

[4] Li C, Wang S, Jia Y. (2011) The performance of a centrifugal fan with enlarged impeller. Energy Conversion and Management, 52: 2902-2910.

[5] Abanto J, Reggio M. (2006) Numerical investigation of the flow in a kitchen hood system. Building and Environment, 41: 288-296.

[6] Son P.N., Kim J.W., Byun S.M., Ahn E.Y. (2012) Effects of inlet radius and bell mouth radius on flow rate and sound quality of centrifugal blower. Journal of Mechanical Science and Technology, 26: 1531-1538.

[7] Wong Y.W., Chan W.Kong., Hu W. (2007) Effects of tongue position and base circle diameter on the performance of a centrifugal blood pump. International Center for Artificial Organs and Transplantation, 31: 639-645.

[8] Huang C.K., Hsieh M.E. (2009) Performance analysis and optimized design of backward-curved airfoil centrifugal blowers. HVAC&R, 2009: 461-488.

[9] Perry R., Gree D., Maloney J. (1984) Perry’s Chemical Engineers’ Handbook. McGraw-Hill. New York.

[10] Fluent 14.0: Theory Guide, ANSYS. Inc., Canonsburg, PA. 2011. 51-54.

[11] Zhang T.H., Liu F.G., You X.Y. (2014) Optimization of gas mixing system of premixed burner based on CFD analysis. Energy Conversion and Management, 85: 131-139.

[12] Shih Y.C., Hou H.C., Chiang H. (2008) On similitude of the cross flow fan in a split-type air-conditioner. Applied Thermal Engineering, 28: 1853-1864.

[13] Alvin C.K.L., Chen F. (2006) Modeling particle deposition and distribution in a chamber with a two-equation Reynolds-averaged Navier-Stokes model. Aerosol Science, 37: 1770-1780.

[14] HE L.B., Tao J., LI Z.F., Wang J. (2016) Orthogonal design and experiment analysis of multi-blades centrifugal fan impeller. Fluid Machinery, 44: 7-11.