Proceedings of the 3rd International Conference on Fluid Flow and Thermal Science (ICFFTS'22)
~~Seoul, South Korea~~ Virtual Conference - October 27 - 29, 2022
Paper No. 108
DOI: 10.11159/icffts22.108

Internal Flow and Aerodynamic Optimization of a Multi-Blade Centrifugal Fan

Ronghui Shao[1], Huanxin Lai[1]
[1] School of Mechanical and Power Engineering,
East China University of Science and Technology,
130 Meilong Road, Xuhui District, Shanghai, P. R. China
1012844327@qq.com; hlai@ecust.edu.cn

Abstract **-** In this paper, internal flows in a multi-blade centrifugal fan are numerically studied for two purposes. The first purpose is to clarify the influence of supports and brackets on the flow field. Two models with and without bracket and supports respectively, are established and considered. The calculations are carried out by RANS. The influences of the bracket and supports at the entrance are analyzed. The second purpose of this paper is the aerodynamic optimization of the fan. It is carried out by using double-circular-arc blades to replace the original single-circular-arc blades, so as to reduce the inflow incidence. The results show that, when the bearing bracket and impeller supports at the entrance are included in the calculation, the performance is only slightly lowered. And, properly reducing the blade inlet angle can improve the inflow incidence, so the total outlet pressure and efficiency are improved. The reason is further explained and analyzed based on the visualization of the flow fields.

Keywords**:** Multi-blade centrifugal fan; CFD; Aerodynamic optimization; Blade inlet angle

1. Introduction

Multi-blade centrifugal fans are widely used in many engineering practices, such as the smoke extraction fans for kitchen, and the ventilation and air-conditioning fans. The optimization of its efficiency has always been the focus of research in recent years. Due to the characteristics of short blade length and the large turning angle for flow at the entrance, the overall flow efficiency of the fans is relatively low. Therefore, study of the flow and optimize the flow path so as to improve the efficiency is quite necessary for these fans.

The multi-blade centrifugal fans refer that the inlet and outlet diameter ratio is greater than 0.8, the width-diameter ratio is greater than 0.25, the blades shape are generally single arc or double arc, and the number of impeller blades is large. Due to its low noise and small size, it is widely used in household appliances such as air-condition fans and smoke-evacuator fans [1-3]. However, because of the short blade length and large turning angle in flow entrance, the overall flow efficiency is relatively low [4-6]. Considering the huge number of household appliances which consume a lot of energy. It is quite necessary to improve the efficiency for multi-blade centrifugal fans.

For the optimization of multi-blade centrifugal fans, there are usually two methods: experiment and simulation. Through experiment, many prototypes with different parameters need to be manufactured, and each prototype has to be tested separately, which is costly and troublesome. Oppositely, simulating calculation is more convenient. By reprocessing the calculation results, the details of flow field which experiment can't find is showed out, making easier to find the cause of the reduction in efficiency, then carry out targeted optimization. Dou [7] et al. studied the influence of the volute tongue at the outlet of the multi-blade fans on the flow, and proposed a bionic volute tongue similar to the wing edge of an owl. Fu [8] optimized a multi-blade fan for smoke-evacuator through simulation and improved its efficiency. Zhou [9] et al. obtained the optimal blade shape under a design condition of the fan through a genetic algorithm, which increased the efficiency of the fan by about 4%.

However, in the research of most scholars, the supports for fixing the impeller shaft and the brackets for bearings at the inlet is always ignored when establishing the calculation model, at the same time the influence of the supports and brackets on the flow field is still lacking, which is undoubtedly unconducive to the development of fans simulation calculation. Secondly, most optimizations of the blade shade are aimed at maximizing the efficiency, so the calculated blade shape is always some complex function curve, which is not convenient to produce in fact. In this regard, this paper respectively establishes and considers two models with and without supports and brackets, calculating by ANSYS Fluent, comparing and analyzing the influence of supports and brackets on the simulation results. Meanwhile, aerodynamic optimization of the fan is carried out by using double-circular-arc blades to replace the original single-circular-arc blades, so as to reduce the inflow incidence.

2. Geometric Model and Blade Optimization

The simulation object is a certain type of central air-conditioning fan. The fan is a double-suction fan with symmetrical structure on both sides, as shown in Figure 1. It adopts a multi-blade centrifugal impeller, and the blades' shade are single arc with equal thickness, the diameter of the fan impeller D=18inch (457mm), limit speed n_{max}=1100rpm, inlet diameter D_{in}=365mm.

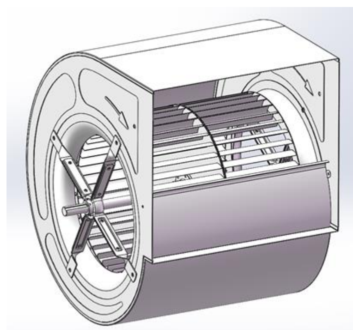
Fig. 1: Geometric model.

The impeller is a key component of the fan, and the shape of the blade determines the physical characteristics of the impeller. As shown in Figure 2(a), the original blade of the fan is a circular arc blade of equal thickness, the pressure surface radius is 30mm, the suction surface radius is 31mm, the blade thickness is 1mm (only the pressure surface radius is marked in the figure), and the blade inlet angle is 76.3°, the blade outlet angle is 161.6°. According to the Euler equation of the impeller machinery, when the impeller rotation speed is constant, the energy obtained by the fluid flowing through the blade is related to the relative velocity when it flows into and out of the blade, and irrelevant with the flow process in the blade. When the attack angle referred to difference of fluid inlet angle and the blade inlet angle is zero, it is called non-impact inflow. In this fan, the inlet angle of the blade is large, and the air flows in with impingement, which will lose some energy and reduce the total pressure and efficiency. As shown in Figure 2(b), in order to ensure the same blade outlet angle and the convenience of engineering producing, while reducing the blade inlet angle to reducing the impact loss, a shape with double tangent circle arcs is proposed. By changing the arc radius Rx of the inlet section to change the blade inlet angle, a total of six blade models with different blade inlet angles of 40°, 50°, 60°, 70°, 73° and 80° were established. The total pressure P_t and shaft power H was obtained by simulation, then calculate the efficiency η to draw the performance curves under different blades, and comparing them to get optimal blade inlet angle of this model.

$$P_t = P_{out} - P_{in} \tag{1}$$

$$\eta = 9.549 \frac{P_t Q}{nH} \tag{2}$$

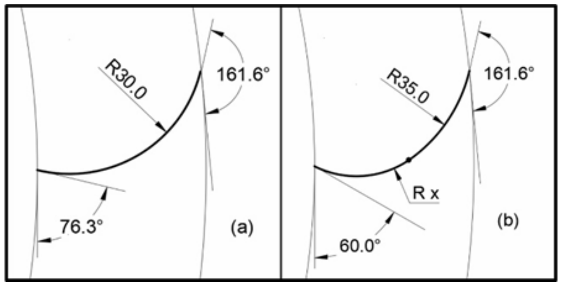

Fig. 2: Blade shade: (a) prototype (b) optimization.

3. Mesh and Numerical Model and Boundary Conditions

Because the left and right structure of the fan is symmetrical, and referring to the simulation results of similar fans [10], the internal flow field of this type of fan is symmetrically distributed. Therefore, during the calculation, only half of the internal watershed is divided along the symmetry plane, and the boundary conditions are set as symmetry in the software. As shown in Figure 3(a), the flow area of inlet part takes a hemispherical area with a radius of $2D_{in}$, and the inlet boundary is the hemispherical surface of this area to respect the physical fact that the gas can flow into the fan from all directions [11], and the flow area of outlet part simulates the actual outlet pipeline, which is a cuboid area with the same cross-sectional area as the outlet section and a length of $2D$. The grid is divided into blocks according to the inlet, volute, impeller, and outlet. Each block adopts a structured grid, and the overall grid quality is greater than 0.45, which respects the calculation requirements, and the Y plus of the whole wall surface is 60.

As shown in Figure 3, the mesh of the supports and brackets at inlet part is same structure grid as others. The diameter of the rotating shaft is d_1=25mm, the total diameter of the bearing and its bracket is d_2=70mm, and the thickness of the bracket support is t=7.5mm.

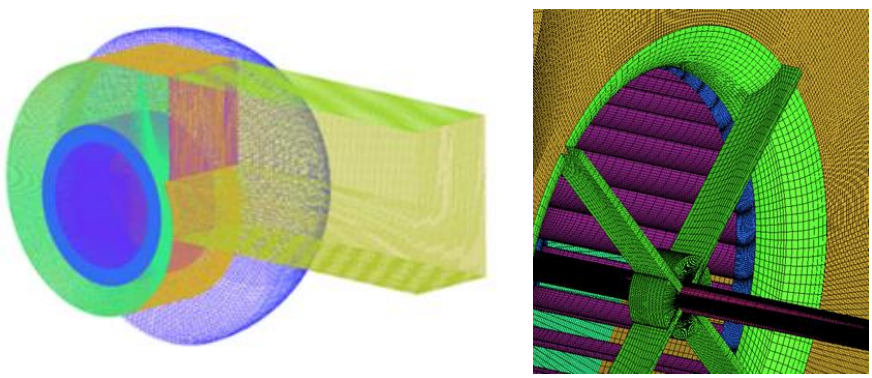
Fig. 3: Overall meshing and the part of the supports and brackets.

The multi-blade centrifugal fan model was numerically simulated using the commercial CFD software ANSYS Fluent. The working environment pressure of the fan is P_0=101325Pa, the inner fluid medium is standard air, the density ρ=1.293kg/m^3, and the dynamic viscosity μ=1.72×10^{-5}Pa·s. The outlet of the impeller has the fastest linear velocity. The calculation shows that the Mach number $Ma=2D\pi n/60c$=0.0985 is much less than 0.3, where c is the local sound speed, so it is treated as an incompressible flow without considering the energy equation. The Navier Stokes equation is solved by the finite volume method, and the minimum Reynolds number $Re > 10^5$ can be obtained through the flow calculation under different working conditions. According to the suggestion of Bouhelal [12], the RNG k-ε model using the second-order upwind scheme with the Scalable wall function is selected, which can get relatively better results. So this paper also uses the same setting.

For the calculated boundary conditions, the symmetry plane is directly set as the symmetry boundary condition. The inlet is set to the mass flow inlet, with the back pressure is 0Pa, and the direction is perpendicular to the hemispherical surface. The outlet is set to the outflow boundary condition. The rotating shaft and blade's wall and the internal flow area of the impeller is set to MRF (Motion Reference Frame) method to simulate the rotation. And the rest of the walls are set to static no-slip walls.

The grid independence verification was carried out under the condition of Q=4m^3/s and n=700rpm. The grids 1 million, 2 million, 3 million, 4 million, 5 million and 6 million. The verification results are shown in Figure 4. When the elements number is more than 5 million, with the element increase, the total pressure is only changed by 0.7%. The calculation precision could not be improved, so 5 million elements' grid were finally selected for calculation.

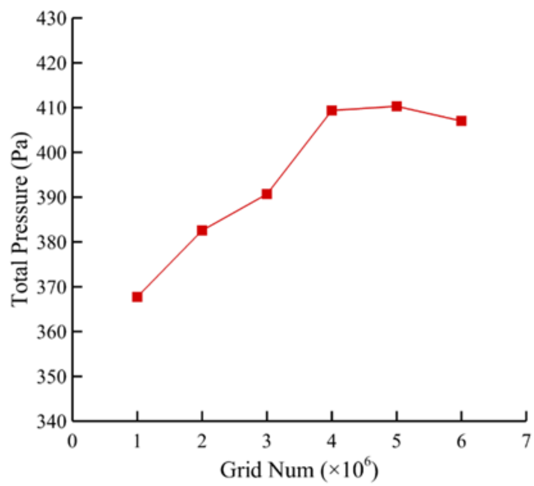

Fig. 4: Independence verification of grid.

4. Calculation Model Verification

To ensure the accuracy of the numerical simulation results, the total pressure and efficiency obtained from the numerical simulation were compared with the experimental results. The experimental results are from the technical manual provided by the fan manufacturer, carried out on a laboratory bench conforming to AMCA. Figure 5 shows the total pressure and efficiency curves of simulation and experiment at two rotational speeds of 400 rpm and 700 rpm, respectively, where Q_n is the design flow rate at 400 rpm and 700 rpm, respectively. Under the condition of large flow rate, the curve is in good agreement with the experimental result; under the condition of small flow rate, the simulation result is larger than the experimental value. A possible reason is that, at low flow conditions, the airflow does not fill the entire path, resulting increase in pressure and the calculated efficiency. In addition, the pressure of the experimental data is the result of the average of multiple measuring points on the outlet surface, and is greatly affected by factors such as the location of the measuring points. Therefore, except that the flow rate is too small, the numerical simulation results can provide a reference for the design and optimization of the fan.

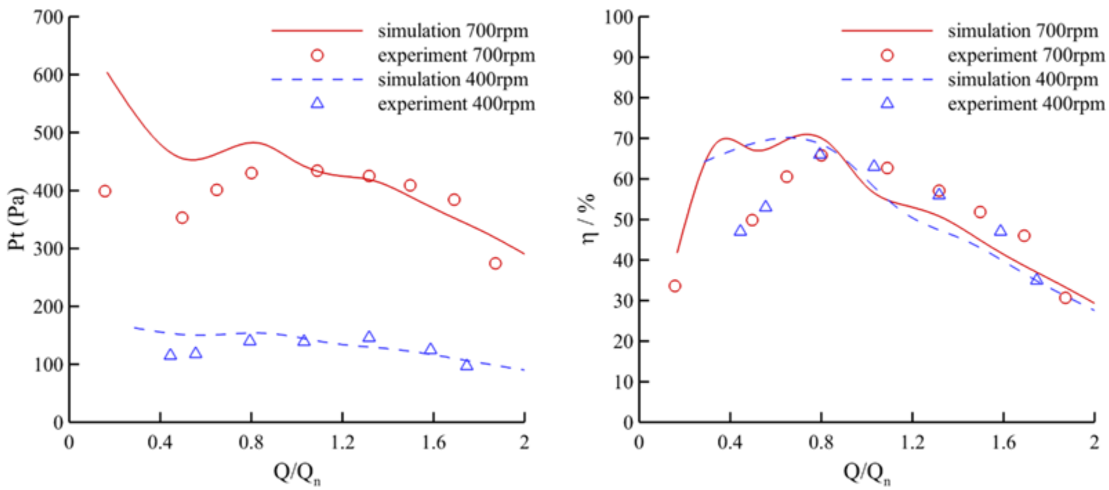

Fig. 5: Comparison of total pressure and efficiency.

5. Influence of Supports and Brackets

As shown in Figure 6, the experimental and simulated performance curves under the 700-rpm condition are shown respectively when the supports and brackets are and are not considered. It shows that the simulation results of the two models are similar to the experimental results, and whether the supports and brackets are considered has little effect on the total pressure and efficiency, the difference between the two is only about 3%. The curve with supports and brackets is lower than the curve without. Near the design flow rate, the two models are in good agreement with the experimental results, which are acceptable in engineering.

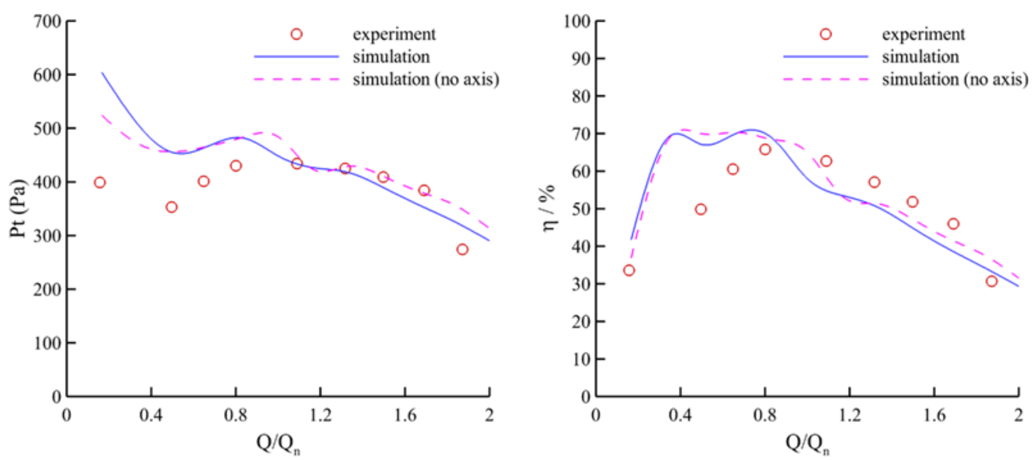

Fig. 6: Total pressure and efficiency curve with and without supports and brackets.

Taking the flow rate Q=4m^3/s as an example, as shown in Figure 7, it is the total pressure contour of different Z sections from the front of the support to the impeller inlet. It shows that the influence of the supports and brackets on the total pressure is limited to the area near them, the influence has completely disappeared at the inlet of the impeller, and it has almost no effect on the inner area of the impeller, but because it has some resistance to the air, it reduces the average total pressure of the entering gas. Therefore, in the previous total pressure and efficiency curve, results with supports and brackets, is lower than results without them.

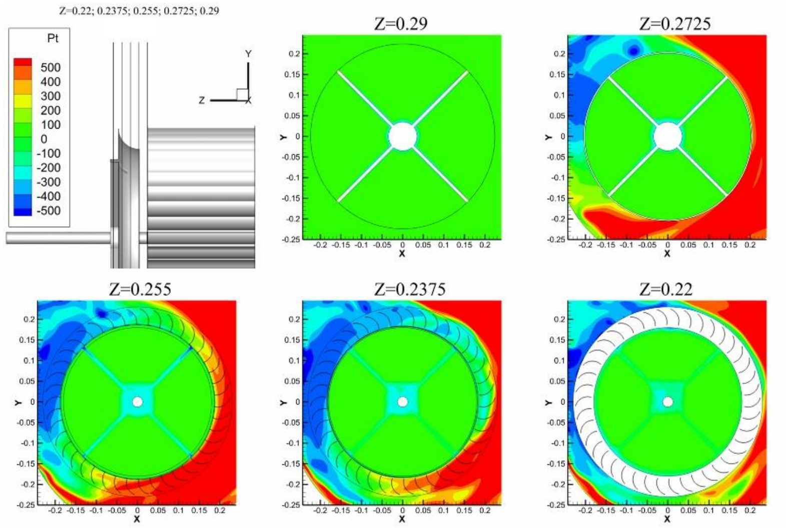

Fig. 7: Total pressure contour near the supports and brackets.

6. Results of Blade Optimization

Reducing the blade inlet angle can reduce the impact loss of entering air flow, thereby improving total pressure and efficiency. For the convenience of analysis, where the simulation results are in good agreement with the experimental results, that is, $Q \geq 2.5\mathrm{m}^3/\mathrm{s}$, is took out. As shown in Figure 8, the total pressure and efficiency curves at each blade inlet angle are respectively shown. It can be seen from the figure that reducing the blade inlet angle can improve the total pressure and efficiency of the fan. The average optimized total pressure percentage and average efficiency optimization is defined, Table 1 lists the average value of the new blade inlet angle compared with the original,

$$\overline{P_t} = \frac{\sum_{q=2.5}^{6}\left[\left(P_{tq} - P_{t0q}\right)/P_{t0q}\right]}{8} \times 100\% \tag{3}$$

$$\bar{\eta} = \frac{\sum_{q=2.5}^{6}\left(\eta_q - \eta_{0q}\right)}{8} \tag{4}$$

where P_{tq} is total pressure of new blade inlet angle in flow rate $Q=q$, P_{t0q} is total pressure of original blade inlet angle in flow rate $Q=q$, η_q is efficiency of new blade inlet angle in flow rate $Q=q$, η_{0q} is efficiency of original blade inlet angle in flow rate $Q=q$.

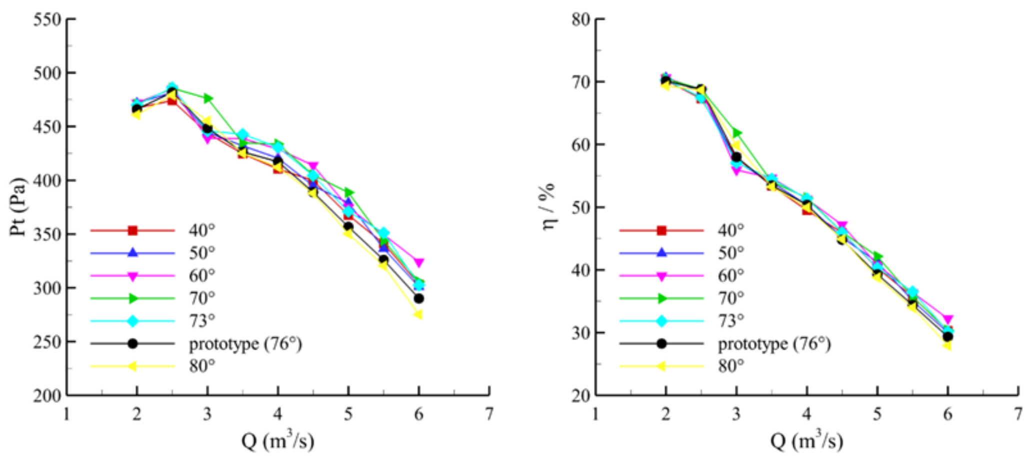

Fig. 8: Total pressure and efficiency curve of each blade inlet angle.

Table 1: Comparison of different blade inlet angle.

Blade inlet angle	$\bar{P}_t$	$\bar{\eta}$
40°	1.16%	0.21%
50°	1.91%	0.37%
60°	3.99%	1.01%
70°	4.20%	1.36%
73°	3.13%	0.56%
80°	-1.18%	-0.21%

It shows that the total pressure and efficiency of the fan can be improved by appropriately reducing the blade inlet angle. This is because through reducing the blade inlet angle, the impact loss at the inlet is reduced and the energy loss is reduced. In this fan model, when the blade inlet angle is reduced 6 degrees to 70 degrees, the total pressure and efficiency increase the most, which are increased by 4.2% and 1.36% respectively. When the blade inlet angle is further reduced, although the total pressure and efficiency are higher compared with the original blade, but lower than 70 degrees. Although according to Euler equation, the energy obtained by the air after flowing through the blade is only related to the inflow angle and the outflow angle, but the conditions of Euler equation assume an infinite multi-blade model, while the actual number of fan blades is limited. When the blade inlet angle is too small, the circle arc radius *Rx* of the inlet section is too small, resulting in the increase of its curvature *k=1/Rx*, which will reduce the working ability of the blade. Therefore, further reducing the blade inlet angle will resulting in reduced total pressure and efficiency.

According to the velocity triangle, the absolute velocity of the air in the impeller obtained by the simulation, and the circular motion velocity of the impeller is subtracted by the post-processing software, the relative velocity of the air flowing in the impeller can be obtained, as shown in Figure 9, which is the near-wall surface (*Z*= 0.01) when *Q*=4.0m3/s, the relative velocity streamline of original blade inlet angle. It shows that due to the large blade inlet angle, a large impact loss is caused, and even a rotating stall phenomenon occurs at the inlet of some blades, which will reduce the total pressure and efficiency. As shown in Figure 10, through reducing blade inlet angle, the attack angle also decreases simultaneously, and the impact loss decreases. Meanwhile, the rotating stall phenomenon at the inlet of each blade is significantly weakened, which can improve the total pressure and efficiency of the fan.

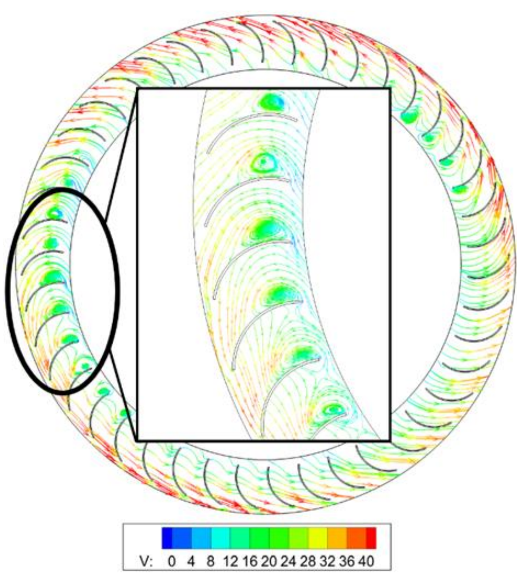

Fig. 9: Velocity streamlines near wall.

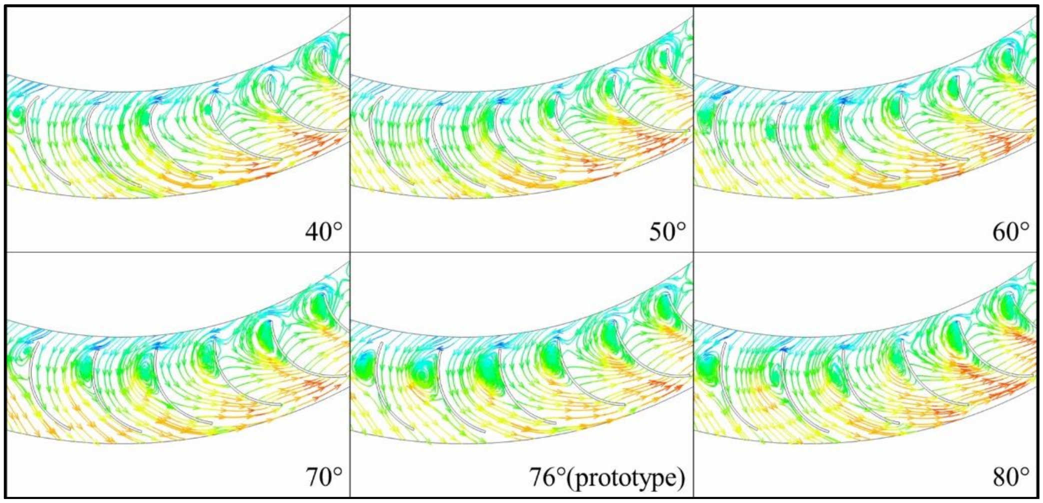

Fig. 10: Velocity streamlines of different blade inlet angle.

7. Conclusion

Through the comparison of simulation and experiment under the working conditions of 400rpm and 700rpm, it shows that the calculation model predicts better. At different rotating speeds, the prediction is better near the design flow, and the prediction has large deviation at small flow rate.

Total pressure and efficiency curves show that the supports and brackets have little influence on the simulation results. The total pressure contour near the supports and brackets shows that their influence on the flow field is only near them, and after the air entering the impeller, the influence of them has almost disappeared. However, because of the resistance of the supports and brackets, the simulation results with them are relatively lower than the results without them.

The impact loss can decrease by properly reducing the blade inlet angle, which improves the total outlet pressure and efficiency. However, excessive reduction of the blade inlet angle will reduce the working capacity of the blade and reduce the total outlet pressure and efficiency. For this model, the efficiency is improved best when the blade inlet angle is 70°, and the double-circular-arc blade has a simple process, which can improve the efficiency while taking into account the manufacturing, having engineering value.

Acknowledgment

This study is funded by the NSFC under Grant 51976061.

References

[1] Liu Z.Y., Zhang Y., Study on the Trailing-edge Serrated Impeller of the Multi-blade Centrifugal Fan in the Range Hood, China Household Appliances Technical Conference, 2021, pp. 1174-1180.
[2] Lan Y., Shi M., Optimization Design of Volute Profile of Multi-Blade Centrifugal Fan for Ducted Type Air Conditioner, China Household Appliances Technical Conference, 2021, pp. 869-874.
[3] Xie C.D., Shu Y., He M., Liu Z.L. and Yang L.N., Study on the Cross-section Structured Impeller of the Multi-blade Centrifugal Fan in the Range Hood, Fluid Machinery, Vol. 47, 2019, pp. 23-27.
[4] Kawasaki M, Hirahara H, A Study on the Process of Low-Frequency Noise Generation in a Multi-Blade Centrifugal Fan, International Journal of Fluid Machinery and Systems, Vol. 13, 2020, pp. 292-301.
[5] Yu S.q., Wu D.H., and Yang S., Numerical Study on the Influence of Blade Outlet Angle on the Performance of Multi-blade Centrifugal Fan, Fluid Machinery, Vol. 47, 2019, pp. 1-7.
[6] Zhao D, Chen R, You X, Numerical study of the effect of inner guide vanes on performance of multi-blade centrifugal fan of range hood, IOP Conference Series: Earth and Environmental Science. IOP Publishing, Vol. 332, 2019.

[7] Dou H.S., Dong X. Effects of bionic volute tongue bioinspired by leading edge of owl wing and its installation angle on performance of multi-blade centrifugal fan, Journal of Applied Fluid Mechanics, Vol. 14, 2021, pp. 1031-1043.
[8] Fu Z., Flow field numerical simulation and performance optimization of kitchen ventilator, Guangdong University of Technology, 2019.
[9] Zhou S., Zhou H., Yang K., Research on blade design method of multi-blade centrifugal fan for building efficient ventilation based on Hicks-Henne function, Sustainable Energy Technologies and Assessments, Vol. 43, 2021.
[10] Li Z., Ye X., Wei Y., Investigation on Vortex Characteristics of a Multi-Blade Centrifugal Fan near Volute Outlet Region, Processes, Vol. 8, 2020, pp. 1240.
[11] Wen X., Qi D., Mao Y., Experimental and numerical study on the inlet nozzle of a small squirrel-cage fan, Proceedings of the Institution of Mechanical Engineers, Vol. 227, 2013, pp. 450-463.
[12] Bouhelal A, Smaili A, Guerri O, Numerical investigation of turbulent flow around a recent horizontal axis wind Turbine using low and high Reynolds models, Journal of Applied Fluid Mechanics, Vol. 11, 2018, pp. 151-164.