Comparative Study of Free Vortex and Random Vortex in the Design of Wind Tunnel Axial Fan

Zihai Geng 1,2, *, Jinsheng Cai 1, Zhilong Huang 2

1 School of Aeronautics, Northwestern Polytechnical University, Xi’an 710072, China;
2 China Aerodynamics Research and Development Center, Mianyang 621000, Sichuan, China

*Corresponding author e-mail: gzhlxqtt_11@163.com

Abstract. The design methods of axial flow fan mainly include free vortex design method and random vortex design method. The random vortex design method and free vortex design method were used to design aerodynamic parameters of two fan systems, and then the performance of these two fan systems was compared by numerical simulation. In the numerical simulation, the random vortex method and free vortex method are applied, respectively, based on the fan design parameters given by the acoustically guided wind tunnel fan section, and the physical model was established, the numerical simulation comparison study was conducted for the two sets of fan systems. The comparative analysis of the numerical simulation results shows that compared with the free vortex method, the fan designed by the random vortex method can reduce the local separation of tip and root of the blade, and the axial velocity distribution of fan blade and fan section outlet can be more uniform, and there is a certain increase in fan blade efficiency. These results verify that the design method of random vortex fan is feasible in large wind tunnels, especially in wind tunnels with aerodynamic noise reduction requirements, and can provide powerful conditions for optimizing aerodynamic and acoustic performance of the fan.

Keywords: fan, random vortex, free vortex, aerodynamic design, numerical simulation.

1. Introduction

At present, the design methods of axial flow fan mainly include free vortex design method and random vortex design method [1]. The free vortex design method is relatively mature and widely used.

The design method of free vortex fan is based on the blade-element theory and ignores flow three-dimensional effect in blade design. It assumes that the flow is uniform, the axial flow velocity is equal at different radial positions; and there is no radial flow of corresponding cylindrical sections at different radial positions. It deduces the flow characteristics of each fan section by classical aerodynamic mass equation, momentum equation, energy equation and Bernoulli’s equation, and then calculates the resistance, torque and power of fan. Specifically speaking, the free vortex method can be divided into Paterson, Collar, and Wallis design method [2].

Paterson’s method considers the fan and rectifier system as a whole, rather than simply involves the fan itself [3]. The fairing of the built-in motor, the support chip of the fixed fairing and the fan rotor
were used as parts of the whole fan section. In order to optimize the airflow quality of the fan section, less airflow energy is dissipated, and downstream of the fan section, it must be guaranteed that it has only a small rotational speed. If considering that pre-rotation blade was arranged in upstream of fans and the straightening vanes were arranged in downstream of fan to enhance the working function of the fan, then such a fixed rectifier system is used to eliminate the air rotation speed under all different working conditions as much as possible, and the selection of blade parameters are becomes very important. In the parameter’s selection of the blade, the axial thrust generated by the stator blades that eliminate the airflow rotation must also be incorporated into the pressurization category of the fan blade, and corresponding consideration is needed in the fan design. Each section of the fan is operated at equal efficiency, although it is of little value in improving fan performance, there is no need to conduct usual graphical integration for the thrust and torque load curves in design, therefore, the design of fan is simplified [4].

The design idea of the random vortex design method was introduced by Wallis in the early days [5]. In the existing wind tunnel application, the fan of German-Netherlands DNW-LLF wind tunnel conducted fan design with random vortex design method. However, in general, there are not many random vortex design methods in engineering applications, and there is still lack of sufficient design experiences and results as references [6].

The main work carried out in this paper has two aspects: the first is the aerodynamic design of the axial flow fan of the low-speed wind tunnel, the free vortex design method and random vortex design method are applied, respectively, and the design parameters of the fan are given, the second is the numerical simulation of the axial fan flow of the low-speed wind tunnel, according to the fan parameters given by the random vortex design method and free vortex design method, the corresponding model is established to carry out numerical simulation of the fan flow field. The results of these two methods are compared and analyzed, and the fan performances given by the two methods are compared.

2. Comparison of Design Methods for Free Vortex and Random Vortex

The free vortex design method is based on the fan design method of two-dimensional blade-element theory, it is assumed that the axial velocity of the fan flow is equal everywhere, and the pressure rise of the air flow at different radial heights after passing through the fan is equal. Free vortex design method is characterized by simple principle, handy calculation, and it has been proven to have reliable design accuracy through many tests and engineering practices.

The random vortex design method no longer assumes that the flow axial velocity of the incoming flow and the radial pressure rise of the blade are equal. Its main feature is that assume the axial velocity distribution of the incoming flow is not uniform, and the radial distribution of the rotational velocity of the fan airflow during the fan design process is no longer inversely proportional to the radius, the airflow pressure rise of corresponding blade upstream and downstream is no longer uniform along the radial direction. During the design process, the axial velocity distribution of the incoming flow may be determined by scale model test or based on test results of existing similar devices. The distribution of the airflow rotation speed is calculated and optimized by the designer based on experience. Under the condition that axial velocity distributions of incoming flow are known, the different rotational speed distributions correspond to axial velocity distributions of different fan outlet airflow. Then, in the fan design, if airflow axial velocity distribution of relatively uniform fan outlet as one of the design goals, a corresponding rotational speed distribution can always be selected to meet the demand, and thereby determine other related fan system aerodynamic parameters. That is to say, in the design of the random vortex method, the airflow rotation velocity distribution can be changed according to actual needs.

The actual wind tunnel flow field requires axial velocity at the fan outlet to be uniformly distributed; its purpose is to improve the possible airflow separation in the downstream diffusion section of the fan. In the wind tunnel loop, the characteristic of the axial velocity distribution of the incoming flow in the fan section is that the outer velocity near cavity wall is lower, while the velocity near the center body is higher. If according to the conventional fan design method, no measures are taken to improve the uniformity of the axial velocity distribution of the outlet, it is possible that the central part of the fan
outlet has a higher velocity, while the outer part has a lower velocity. This situation is not conducive to the aerodynamic performance of the downstream diffusion section of the fan section, which may cause air separation. On the contrary, under the condition of reaching the same fan flow and pressure rise indicators, the free vortex method can form a more uniform axial velocity distribution at the fan outlet, which is conducive to restrain the possible flow separation in the downstream diffusion section.

Actual wind tunnel requires high aerodynamic efficiency and low noise of axial fan. Therefore, stall separation of blades should be avoided, which is in conflict with improving aerodynamic efficiency. The random vortex method can be used to adjust blade load conveniently according to trial calculation results and aerodynamic performance of the blade airfoil, thus ensuring that the blade does not stall and has better aerodynamic efficiency. Especially when the wind tunnel has many operating conditions, the fan pressure rise, speed and flow change range of incoming flow are wider, the random vortex method can more accurately predict the local blade load (lift coefficient) distribution at different radial positions of the fan blade, so as to optimize and adjust it. In addition, the assumption that the axial velocity distribution of incoming flow is not necessarily uniform in random vortex method is closer to the real situation, and the local load distribution of blades can be calculated more accurately, thus ensuring the proper stall margin, which is more conducive to guarantee the performance of fans.

Compared with the free vortex method, one of the defects of the free vortex method is more complicated calculation process. In the design process, it is necessary to calculate the downstream axial velocity distribution of the fan blade and the straightening vanes in allusion to the axial velocity distribution of the incoming flow, and then determine the aerodynamic parameters of the blade; these processes greatly increase the computational load. Therefore, in many wind tunnel practices, if the uniformity of fan outlet velocity or the local stall of fan blade need not be paid special attention to, the free vortex design method is often used. Only in some cases where special attention needs to be paid to these contents, the random vortex design method is applied.

3. Numerical Calculation
The fan 1 has 10 moving blades and 7 straightening vanes. Since the support blade only plays supporting role, it does not contribute much to the aerodynamic performance of the fan, so the support blade is not designed in order to simplify the physical model and reduce the overall size of the grid.

In numerical simulation, the position of the inlet and outlet has an important influence on the calculation result, in order to reduce this effect as far as possible, and considering the actual situation and computing resources, the inlet of the fan section is selected at the front end 0.83m to the rectifying center bow cap, the outlet of fan section is selected 1m from the last end of the rectifying center tail cap.

**Figure.1** fan model

**Figure.2** 3D grid of fan section
The grid of fan moving blade and straightening vane is the content that needs to be focused on in numerical simulation. When building the grid around moving blades and straightening vane, it is necessary to ensure the grid quality of this part. In order to ensure good orthogonality of the grid, a C-shaped grid is used around the blade in this paper. Through calculation, the height of the first layer of grid of the blade surface is about 0.3mm.

During numerical simulation of flow field of the free vortex fan, the inlet adopts the mass flow inlet condition, the outlet adopts the outflow condition, and the MRF model is used to realize the rotation in the moving blade; the rectifying center body, the hole wall, the moving blade, the straightening vane all adopt a solid wall surface, the wall surface of the moving blade is set as a rotating wall surface, and the rotation speed is consistent with the fan.

4. Comparison of Results

Figure.7 to Figure.12 show the shear force and static pressure distribution of the moving blade surface at different heights corresponding to the above working conditions. When the fan of the closed test section is 100m/s, the shear force distribution and static pressure distribution near the root and the middle of the blade are similar, the corresponding blade performance of the two methods is similar. In the shear force distribution near the tip of the blade, the moving blade of random vortex fan and the free vortex fan appear close to 0 at L/C = 0.9 and 0.85, respectively, from the surface static pressure distribution, there are also some changes in these two position, so we analyse the two positions corresponding to their respective separation positions. Obviously, this also shows that the separation area of the random vortex fan under this working condition is significantly smaller than the separation area in the case of free vortex. When the wind speed of the closed test section is 130 m/s, the moving blade of random vortex fan and the free vortex fan show a similar phenomenon. These distribution characteristics are also consistent with the above-described oil flow line distribution.

Figure.13 and Figure.14 show the velocity distribution of the free vortex fan and free vortex fan at the fan section outlet. It can be seen from the figure that the wake effect of central body tail is weaker
than that of free vortex fan in the case of random vortex fan. Therefore, the use of random vortex fan is helpful to improve the quality of the downstream airflow.

**Figure 7** The wind speed of the test section is 100m/s, shear force and static pressure distribution on the surface of the moving blade with 0.28m radial height.

**Figure 8** Wind speed of the test section is 100m/s, shear force and static pressure distribution on the surface of the moving blade with 0.36m radial height.

**Figure 9** Wind speed of the test section is 100m/s, shear force and static pressure distribution on the surface of the moving blade with 0.445m radial height.

**Figure 10** Wind speed of the test section is 130m/s, shear force and static pressure distribution on the surface of the moving blade with 0.28m radial height.
Figure 11 wind speed of the test section is 130m/s, shear force and static pressure distribution on the surface of the moving blade with 0.36m radial height

Figure 12 wind speed of the test section is 130m/s, shear force and static pressure distribution on the surface of the moving blade with 0.445 radial height

Figure 13 wind speed of test section is 100m/s, comparison of outlet axial velocity of random vortex fan and free vortex fan

Figure 14 wind speed of test section is 130m/s, comparison of outlet axial velocity of random vortex fan and free vortex fan

5. Conclusion

In the design process, the random vortex fan can adjust the load distribution of the moving blades, so a reasonable design result can always be obtained. Through the analysis and comparison of the results, it can be found that the moving blade separation area of the random vortex fan is smaller than the moving blade separation area of the free vortex fan, the axial velocity distribution of the straightening vane outlet is more uniform, the wake of the fan center tail has a smaller influence range, therefore, the aerodynamic performance of the fan designed by the random vortex method is better than that of the fan designed by the free vortex method.

References

[1] Kang S, Hirsch C, Numerical Simulation of Three-Dimensional Viscous Flow in a Linear
Compressor Cascade with Tip Clearance, ASME Journal of Turbomachinery, Vol. 118, No. 3, PP. 492–505, 1996.

[2] Dawes W.N., Simulating Unsteady Turbomachinery Flows on Unstructured Meshes Which Adapt Both in Time and Space, ASME Paper, No. 93-GT 104, 1993.

[3] Wu Xianhong, Chen Maozhang, New procedure for simulating rotor/stator interaction in turbomachinery, PROGRESS IN NATURAL SCIENCE, Vol. 9 No. 7, 1999.

[4] Johan B. H. M. Schulten, Unsteady Leading-Edge Suction Effects on Rotor-Stator Interaction Noise, AIAA JOURNAL, Vol. 38 No. 9, September 2000.

[5] Florian Kromer, Stefan Becker, Experimental investigation of the sound reduction by leading edge serrations on a flat-plate axial fan, AIAA AVIATION Forum, 2018, 10.2514/6 2955.

[6] Thomas Hainaut, Thomas Le Garrec, Cyril Polacek, et al., Aerodynamic and aeroacoustic numerical investigation of an axial fan using lattice boltmann methods, AIAA AVIATION Forum, 2018, 10.2514/6 3922.