Numerical simulation of the whole flow field of an axial-flow fan used in an air conditioner

Q G Chen*, F Li, Q Hu, and Y N Gao

College of Mechanical & Electronic Engineering
Shandong University of Science and Technology, Qingdao 266590, P. R. China

E-mail: chenqingguang03@tsinghua.org.cn

Abstract: Numerical simulation of a three-dimensional turbulent flow in the whole flow field of an axial-flow fan is performed. Unstructured grids are used to discretize the computational domain. Pressure boundary conditions are specified to the inlet and the outlet. The SIMPLE algorithm, in conjunction with the RNG k-ε turbulent model, is used to solve the three-dimensional Navier-Stokes equations. The moving reference frame is adopted to transfer data between the interfaces of the rotating field and the stationary field. Based on the calculation of the inner-flow in the fan, the velocity field and pressure field were further analyzed. The simulation results are of important significance to the optimal design and noise control of the fan.

1. Introduction
With the popularity of household air conditioner among the domestic users, the energy-saving and comfort requirements for air conditioner have been higher and higher. And more and more attentions have been paid to the aerodynamic performance and the noise problem in axial flow fan used for outdoor machine. In the 1990’s, a lot of experimental analysis and improvement have been done in terms of air conditioning axial flow fan by the air conditioner industry abroad. In order to reduce the noise in fan, the flow separation and vortex phenomenon are analyzed. The status of the air flow is improved by adapting the internal flow analysis methods, such as the flow visualization[1]. Currently, CFD methods have been gradually applied to the research and development of air conditioner products with the development of computational fluid dynamics. The improvement plan is put forward to enhance the outdoor machine’s air volume and reduce noise by comparing and researching the data of numerical simulation results and experiment.

It is established that the flow channel analysis model of the whole flow field for outdoor air conditioner. The numerical simulation is conducted for axial flow fan in the voltage of 220V, as well as, a detailed analysis has been conducted for velocity field and pressure field. Then some empirical conclusions are drawn by combining with previous experiments and experience.

2. Geometric model and mesh generation
Because the structure of outdoor air conditioner is very complex, the block grids technology is adopted in meshing the physical model. Currently, grids used in the numerical calculation can be divided into two categories, including structured and unstructured grids. In view of the complex geometry of the outdoor air conditioner, unstructured grids is selected as adaptability grid type to the geometry
structure which can adapt with the irregularly changing fan blade shape fitly\cite{2}. So the model can be divided into air inlet domain, the impeller domain and export domain. It generates more than 2.3 million grids using unstructured grids method, the result of the meshing is shown in figure1. It can be seen that the impeller part has much more grids than the flow field, i.e. The dense grids mainly spread over the domain where the geometric structure of the fan blade’s surface is complex and flow field’s gradient changes fast. Reversely, the relatively sparse grids spread over the domain where the geometry structure of the fan blade’s surface is relatively simple and the flow field’s gradient changes relatively slowly. So the calculated amount can be allocated reasonably.

Figure 1. Grid distribution

3. Boundary conditions
In the numerical simulation of axial-flow fan, the impeller domain is usually defined as the rotating domain, and rotating coordinate frame is adopted. When the rotating speed of fluid is given, the inlet and outlet domains are defined as static domain, and the static coordinate frame is used. The inlet boundary conditions are set in rotating field and stationary field in order to transfer data between the rotating field interfaces with the stationary field. All impeller blade surfaces are defined as the rotating wall, when rotation speed is equal to the speed of the adjacent fluid domains. The surfaces of the outdoor machine and the inlet are stationary wall, and the no-slip conditions are applicable for rotating wall and stationary wall\cite{3}.

Pressure-inlet boundary conditions include the fan’s inlet total pressure, the fluid flow direction, turbulent kinetic energy and import & export rate of turbulent kinetic energy dissipation. Also, pressure-outlet boundary condition includes fan’s outlet static pressure. When backflow appeared, other scalars in backflow should be set as pressure-outlet boundary condition instead of mass-outlet boundary condition, so that a faster convergence rate could be gained\cite{4}.

4. The numerical algorithm
In the process of numerical calculation, the RNG $k$-$\varepsilon$ turbulent model is adopted to simulate turbulent flow; the SIMPLE algorithm is used to achieve the coupling of velocity and pressure; the wall-function method is applied to the near wall domain. The moving reference frame is adopted to transfer data between the rotating field interfaces with the stationary field, so as to complete the numerical calculation on the condition of steady flow\cite{5}. When the calculation accuracy is $10^{-5}$, the monitoring curve of outlet flow in the process of calculation is shown in figure 2. It can be seen that flow changes inconspicuously after 2300 steps, which can be thought that calculation is convergent.
5. CFD results analyses

5.1. Analyses of velocity field

Figure 3 shows the path line of the whole flow field in the outdoor air conditioner. Due to the axial-flow fan rotation doing work and low pressure forming at the fan domain, the fluid flows in from the rear and both sides of the outdoor air conditioner, and then shroud into the atmosphere through the fan under the action of the impeller. It also can be seen that the fluid-outlet flows along the spiral into the atmosphere, and there is less airflow in the middle of the outlet domain while that distributes mainly in the peripheral domain.

Figure 4 shows the velocity vector in the axial section of outdoor air conditioner. It can be seen that, the outward airflow flows into outside machine through the retral evaporator and the side board, then around the motor and into the atmosphere under the action of impeller doing work. The airflow velocity outside impeller is the highest and it is a rather large vortex reflux in the middle domain where the airflow flows into the atmosphere, so the speed is lower. The existence of the gap between the outside of impeller and the fan shroud makes the existence of vortex in the medial and lateral of fan shroud, resulting in the intense vortex detachment around the fan shroud, which leads to the stronger aerodynamic noise.

Figure 5 shows the velocity vector on the surface of the axial-flow fan in the outdoor air conditioner. It can be seen that, the airflow is preferably attached to the surface of vane. Airflow detaches from the blade surface and forms shedding vortex after flowing around the blade. The discrete noise formed by vortex shedding is the main source of aerodynamic noise of axial-flow fan, and the main way to improve the fan’s air volume and reduce aerodynamic noise is designing the blade profile reasonably and determining the installation angle of the import and export of the blade appropriately.
5.2. Analyses of pressure field

Figure 6 shows contours of the static pressure and dynamic pressure in a typical radial section of the axial-flow fan domain. It can be seen that, the static pressure of the blade’s pressure surface is obviously higher than the suction surface. The pressure surface promotes airflow to flow along the axial by doing work on airflow. It can be seen that the pressure distribution exists the obvious gradients in the high-pressure side, which gradually decreases from the side close to the blade surface to outward. While there exists obvious low-pressure domain in the trailing edge of the blade suction surface side, which is due to the vortex shedding phenomenon in the side of trailing edge and also is the main cause of generating aerodynamic noise. The contours distribution trend of dynamic pressure is opposite; this is due to the lower airflow velocity and pressure in the higher static pressure domain.

Figure 7 shows contours of static pressure of eight different radial sections in the axial fan domain, from which it can be seen that the static pressure distribution along the circumferential direction of the fan is symmetric, and the radial pressure distribution of whole fan is affected by the periodic rotation of the fan. The static pressure distribution on the pressure side of the fan is higher than the suction side on the whole, and the domain affected by pressure has a certain extension forward.
Figure 7. Static Pressure field of the different radial sections
6. Conclusions
CFD method is used to complete the numerical calculation of the inner-flow velocity field and pressure field in the axial-flow fan of outdoor air conditioner; the results of numerical analysis reveal the basic characteristics of the inner-flow field in the outdoor axial-flow fan, which are of important significance to the optimal design and noise control of the fan.

References
[1] Velarde-Sandra 2002 Numerical prediction of the aerodynamic tonal noise in a centrifugal fan[J] vol.26(5) (Experimental Thermal and Fluid Science) pp 421-434
[2] Karin H 1981 A study on centrifugal impeller and diffuser flow[J] vol.103 (ASME Journal of Engineering for Power) pp 688-697
[3] Miyamoto Hiroyuki, Nakashima Yukitoshi 1992 Effects of splitter blades on the flows and characteristics in centrifugal impeller[J] vol.35(2) (JSME International Series) pp 238-246
[4] Wan-Ho Jeon 2003 A numerical study on the effects of design parameters on the performance and noise of a centrifugal fan[J] vol.265 (Journal of Sound and Vibration) pp 221-230
[5] Roland Ewert 2008 Broadband slat noise prediction based on CAA and stochastic sound source from a fast random particle-mesh (RPM) method[J] vol.37 (Computers & Fluids) pp 369-387

Acknowledgments
The research reported here is financially supported by the Special Funds of "Taishan Scholars" Project approved by the Government of Shandong Province, and the Instructive Project on Scientific and Technological Research approved by China National Coal Association under No. MTKJ 2011-366.