Air-structure coupling features analysis of mining contra-rotating axial flow fan cascade

Q G Chen∗, W Sun, F Li, and Y J Zhang
College of Mechanical & Electronic Engineering
Shandong University of Science and Technology, Qingdao 266590, P. R. China
E-mail: chenqingguang03@tsinghua.org.cn

Abstract: The interaction between contra-rotating axial flow fan blade and working gas has been studied by means of establishing air-structure coupling control equation and combining Computational Fluid Dynamics (CFD) and Computational solid mechanics (CSM). Based on the single flow channel model, the Finite Volume Method was used to make the field discrete. Additionally, the SIMPLE algorithm, the Standard k-ε model and the Arbitrary Lagrangian-Eulerian dynamic grids technology were utilized to get the airflow motion by solving the discrete governing equations. At the same time, the Finite Element Method was used to make the field discrete to solve dynamic response characteristics of blade. Based on weak coupling method, data exchange from the fluid solver and the solid solver was processed on the coupling interface. Then interpolation was used to obtain the coupling characteristics. The results showed that the blade's maximum amplitude was on the tip of the last-stage blade and aerodynamic force signal could reflect the blade working conditions to some extent. By analyzing the flow regime in contra-rotating axial flow fan, it could be found that the vortex core region was mainly in the blade surface, the hub and the blade clearance. In those regions, the turbulence intensity was very high. The last-stage blade’s operating life is shorter than that of the pre-stage blade due to the fatigue fracture occurs much more easily on the last-stage blade which bears more stress.

1. Introduction
Contra-rotating axial flow fan is a kind of the ventilation equipment, where an impeller is installed on the back of another impeller, and two impellers rotate in opposite directions. This type of fan is widely used in energy security and mine ventilation and tunnel engineering with its compact structure, high efficiency, good performance against wind and wide adjustable range, etc. The ventilator is an important part of ventilation system; and its operation situation directly relates to production safety. Wu[1] used coupling solving method to study cascade flutter problems of all kinds of angle of attack, and studied stall transmission frequency through numerical results. Based on the weak coupling method, the aerodynamic force acting on the blade and blade vibration and deformation characteristics were analyzed which provided the theoretical data to the cause of the failure that the ventilator might occur in the operation process.

2. The object of study
The contra-rotating axial flow fan with large air volume was studied as the research object in this paper. Its rated air volume is 65m³/s, the outer diameter of impeller is 1900 mm, the hub ratio is 0.54,
the number of the pre-stage blade is 13, the number of the last-stage blade is 9 and the speed of two stage blades is 980r/min. According to the above-mentioned data, molded line parameters of the blade were calculated. The geometry model of the blade and the single flow channel model which was used in the numerical calculation were drawn by 3D software.

The mesh model of single flow channel in contra-rotating axial flow fan is shown in figure 1. The inlet boundary and outlet boundary of the calculation area were respectively extended. The unstructured grids was applied to the mesh generation, and area of the impeller with violent motion was refined locally; there were 40,638 nodes and 195,808 units after meshing.

Contra-rotating axial flow fan is the impeller machine that rotates periodically, blade channel is symmetry periodically. Based on this character, the model could be simplified. \(1/z\) of the whole fan blade was calculated, which could decrease calculation area and scale; \(z\) is the number of blades of the contra-rotating axial flow fan. Based on meshing principle, the Finite Element Mesh was adopted \([2]\); and calculation domain of the solid was generated, as shown in figure 2.

![Figure 1. The single channel model](image1)

![Figure 2. Grid chart of blades](image2)

3. The condition of the fluid-structure coupling simulation

3.1. The constraint condition of the fluid
Fan’s flow regime is turbulent, so the standard \(k-\varepsilon\) model was adopted \([3]\). Based on the SIMPLE algorithm of the unstructured grids, the governing equations were solved, and the wall function method was used to solve the near-wall region. On the whole fluid domain, the periodic boundary conditions were given. The speed condition of air in inlet was 32.36 m/s, and the outflow was set as pressure boundary condition. The joint face was defined as mixing plane to ensure that the data transfer completely between the two fluid domains shown in figure 1. With ALE dynamic grids introduced, the blade and hub were set as motion areas, and the blade as the fluid-structure interaction surface that was used to deliver data. During the air-structure coupling calculation, the stationary solution gained in steady calculation was regarded as the initial value in transient flow field, which could ensure the stability of the unsteady flow’s dynamic solution.

3.2. The constraint condition of the solid
The surface of the blade that contacted with fluid was set as the air-structure coupling surface \([4]\), while constraints of roots fixing and blade rotating were added to blade structure. The Finite Element Method was used to solve solid control equations, and then the displacement change of solid structure was achieved.

3.3. Properties of fluid-structure coupling
Based on weak coupling method, the coupling order was set according to the fluid firstly and then solid. In the process of air-structure coupling calculation, time step length and time step number must be consistent with that in the flow domain and solid domain. In order to get dynamic response curves accurately, coupling time step length was set as 0.001 s; the maximum number of iterations was set as 50 each step, there were 200 time steps totally. Data in the fluid solver and the solid solver exchanged five times in the calculation of each step, which ensure the reliability of the coupling calculation.

4. The numerical calculation and the result analysis

4.1. Analysis of flow field characteristics

When air flows through cascade of the contra-rotating axial flow fan, friction and vortex flow are emerge in boundary layer of the blade surface and in the wake flow\(^5\). Because of difference pressure between the concave and convex surface, second-vortex was formed. The phenomenon, such as eddy current and vortex, emerged in the surface of the annular channel formed between fan casing and moving blade. In multistage axial flow fan, when air flow was flowing through the clearance from concave to convex, second-vortex was formed because of the complex flow in the effect of thickening of the wall boundary layer. By the simulation of gas flowing condition in the contra-rotating axial flow fan, the vortex core is shown in figure 3.

**Figure 3. Vortex core region**

It can be seen from figure 3 that the vortex core region is mainly concentrated on the surfaces of blade, hub and in the tip clearance. Several sections of the single flow channel model along the radial direction are selected to observe the internal turbulent flow field, as shown in figure 4.

In figure 4, \( R = 0.513 \) m means closing to the wheel hub; \( R = 0.95 \) means closing to the blade tip. The turbulence intensity of vortex around the blade is higher, which is consistent with the region where large vorticity occurs. In the 1st stage, turbulence intensity around the blade surface gradually increases from the root to the tip. In the 2nd stage, the range of turbulence decreases from root to tip, but the highest turbulence intensity appears at \( R = 0.55 \) m. According to figure 3 and figure 4, turbulent fluctuations are in the vortex core region.
4.2. Analysis of aerodynamic characteristics

Three monitoring points are selected in the flow field. As shown in figure 5, their pressure fluctuation signals were obtained from the coupling simulation, and response curves in frequency domain could be obtained by Fast Fourier Transform technique, as shown from figure 6 to figure 8.

From the curves of power spectral density versus frequency at point 5, point 6 and point 7, the pressure fluctuation frequency of flow field is 95.9596 Hz.

4.3. Displacement response analysis of blade
Under the effects of dynamic load, the response is different from dynamic characteristics of the points on two levels of the moving blade, such as displacement characteristics and stress. The displacement of the blade in two stages is shown in figure 9 and the two stages of the blade have maximum amplitude in the area around point 1 and point 2.

![Figure 9. Contours of total displacement](image)

The maximum displacement amplitudes for the two stages all appear on the top of the blades. As for the 1\textsuperscript{st} stage, the maximum amplitude occurs at the trailing edge of the blade tip, and decreases gradually toward the blade root. Its gradient is relatively large. As for the 2\textsuperscript{nd} stage, the maximum amplitude occurs at the leading edge of the blade tip, and decreased gradually toward the blade root, but the gradient is relatively small. The displacement power spectral densities at point 1 and point 2 are shown in figure 10 and figure 11.

![Figure 10. Frequency response curve of total displacement at point 1](image)  
![Figure 11. Frequency response curve of total displacement at point 2](image)

The vibration frequencies of two stages of moving blades are 95.9596Hz, which is got from the displacement response curves in frequency domain at point1 and point 2. The value of frequency 95.9596Hz is consistent with the pressure fluctuation frequency of flow field. It shows that the signals of the aerodynamic forces could reflect the working condition of the blade to some extent, and it can be used to uncover the coupling characters between dynamic respond of blades and unsteady flow field.

4.4. Stress response analysis of blade
The life of a solid material mainly depends on its alternating stress amplitude and fatigue strength. The distributions of Von Mises stress on the blades of the two stages are shown in figure 12, the maximum equivalent stresses of the blades occur on point 3 and point 4 respectively.
The root area of fan blade is the most dangerous region that can be seen from the equivalent stress distribution. The maximum stresses acting on the blades of two stages are located in the root of blades and closed to the leading edges, and the stress acting on the blade of the 2nd stage relatively large. The blade of the 2nd stage is significantly affected by the rate of air flow and the flow regime, and the blades in the 2nd stage are more prone to fatigue damage.

5. Conclusions
(1) The turbulence intensity around the blade is higher, which is consistent with the region where large vorticity emerges.
(2) The vibration frequency of the fan’s moving blades is consistent with the pressure fluctuation frequency of the flow field. It shows that the signals of the aerodynamic forces could reflect the working condition of the blade to some extent, and it can be used to uncover the coupling characters between dynamic respond of blades and unsteady flow field.
(3) The maximum deformations of the blades in the two stages are located on the tip of the blades, the maximum stresses acting on the blades are located in the root area of blades and close to the leading edges. The blade of the 2nd stage is significantly affected by the rate of air flow and the flow regime, and the blades in the 2nd stage are more prone to fatigue damage. So the lives of the blades in the 2nd stage will be shorter than those in the 1st stage.

References
[1] Wu W Q 1994 The numerical simulation of cascade aeroelastic discrete eddy: II numerical experiments [J] vol.15(4) (Journal of Engineering Thermophysics) pp 372-377
[2] T Belytschko and D P Flanagan 1982 Finite element method with user-controlled meshes for fluid-structure interaction[J] vol.33 (Computer Method in Applied Mechanics and Engineering) pp 669-688
[3] Zhong J W, Li Y M and Wu Q 2011 Gas-structure coupling vibration analysis of contra-rotating axial flow fans[J] vol.33(1) (Journal of Mining Machinery) pp 28-31
[4] Li L L and Huang Q B 2006 Research on air-structure coupling characteristic of fan blade[J] vol.34(4) (Journal of Fluid Machinery) pp 23-27
[5] Wang S L, Sun Z and Wu Z R 2011 Strength investigation of the centrifugal fan impeller based on fluid-structure coupling[J] vol.38(4) (Journal of North China Electric Power University) pp 81-85

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.