Static Structural Analysis of 2MW Wind Turbine Blade Based on ANSYS Workbench

Z. K. HE¹, J. CAI¹,*, S. S. CUI¹, W. C. LIU², B. W. LIU¹ and Y. D. WANG³

¹School of Nuclear Science and Engineering, North China Electric Power University, Beijing 102206, P. R. China
²Blade Department of R&D Institute of Guo Dian Wind Power Technology, Guadian United Power Technology Co. Ltd, Beijing 100039 China
³State Key Laboratory of Metal Materials, Beijing University Science and Technology, Beijing 100083 China
*Corresponding author

Keywords: Static structural analysis, Wind turbine blade, ANSYS Workbench.

Abstract. In this paper, finite element software ANSYS Workbench is used in the static structural analysis of a kind of 2MW wind turbine blade. The results show that under the maximum possible load, the finite element model of the blade is within the safe range. But relatively speaking, the beam and the trailing edge are relatively bearing larger stress and more dangerous.

Introduction

Wind energy is a clean renewable energy, compared to traditional energy, with fuel-free costs, no carbon emissions and other advantages [1]. Compared with other renewable energy, it also has some advantages, such as its technology is relatively mature, the wide range of resources, low cost and so on. It will certainly become one of the very important forms of energy in the future.

China is rich in wind resources, in the west, north, especially at sea. In recent years, due to the tilt of the policy, the wind power industry of China has a rapid development [2]. From 2012 onwards, China's wind power installed capacity has been ranked first in the world. But compared to Western countries, China's wind power technology is still a big gap.

The blade is one of the most critical parts of the wind turbine. The blades and wheels are composed of energy harvesting mechanism, which transfer the wind energy into mechanical energy. At the same time, the blade is the main bearing part of the wind, and it is the key role to guarantee the wind turbine running safely [3]. To ensure that the wind turbine can keep safe and normal operating in its design life, the static structural analysis of the blade is necessary to be done. Static analysis can also provide a reliable basis for the improvement and optimization of wind turbine blade structure.

Mainstream large-scale wind turbine blades are made of glass fiber reinforced plastic composite material, it has higher strength, higher stiffness, lighter weight, can be designed easily, strong bearing performance and other characteristics [4,5]. However, it also makes the composite structure and ply structure of large-scale composite wind turbine blade is very complex, its shape is built from different airfoils, is a long three-dimensional curved shell structure, and there are a lot of transition layer and sandwich structure. In addition, the load distribution of the wind turbine blade is not regular, and plenty of calculation to get the analytical solution of the blade structure of the composite wind turbine is needed, so the finite element software ANSYS Workbench is widely used in the blade structure analysis of the wind turbine.

Finite Element Model of Blade Structure of Wind Turbine

Wind turbine blade is a slender flexible body, subject to wind pressure load will be a greater deformation. To make the deformation of the leaves still have good aerodynamic performance, the
wind turbine blades often adopt pre-curved leaves \[^6\]. When pre-curved wind turbine blades under the action of wind pressure load, the deformation just offset by the pre-curved blade deformation, the blade under the action of the pneumatic load can also have good aerodynamic performance.

The wind turbine blade used in the calculation is a kind of 2MW pre-curved fiberglass composite blade which has been applied to the wind field and is with double shear webs. The blade consists of the pressure surface, the suction surface, the tip, the root, and the inner two shear webs. In the middle of the pressure surface and suction surface, there is a beam, connecting the leaves and the tip, perpendicular to the shear web. The total length of blade is 59m, and the basic model of the blade is shown in Figure 1.

The ACP (ANSYS Composite PrepPost) software in ANSYS Workbench is used to lay the basic model to obtain a complete and accurate composite blade model.

At First, set the engineering date, enter the material properties. Then fabrics and stackups are built by the different combinations of the basic materials, which are the basis of the composition of the blade unit. Then we lay fabric and stackups to the correct areas in turn as the designed laying order. Eventually forming a complete leaf model.

Then mesh the model. As the model is too large, only part of the mesh is displayed, as shown in Figure 2. There are 125,486 elements of the blade totally.

![Figure 1. Model of wind turbine blade.](image1)

![Figure 2. Part of the mesh of the model.](image2)
At last, apply the maximum load data that the blade may bear to the model, and apply a full constraint to the root boundary. Unified unit, and calculate it.

**Results of the Static Structural Analysis**

From Figure 3, it can be seen that the maximum deformation of the blade is 14.236m under the load. The deformation is relatively large, although the blade pre-bending structure offset most of the deformation, but still have a certain risk, because if the deformation is too large, the blade may hit the tower of the wind turbine.

![Figure 3. deformation of the blade.](image)

As shown in Figure 4, the maximum tensile stress and compressive stress are 146.13 MPa and 147.5 MPa, respectively. The tensile instability strength and the compressive instability strength of the wind turbine blade composite material measured by the experiment are 750MPa and 550MPa respectively, so the possibility of blade failure is very small.

![Figure 4. Stress of tensile and compress.](image)

As shown in Figure 5, the maximum equivalent stress distribution occurs near the contact position of the suction surface, pressure surface and the shear webs. And concentrated in the larger cross-sectional area of the surface. The risk factors of these parts are relatively larger during operation.
It can be seen from Figure 6 that the maximum shear stress is 15.236 MPa, and the safe shear force should be below 45MPa, so the possibility of interlayer stripping is very small. The relatively large areas of shear are mainly distributed in the trailing edge of the leaves near the root, the local risk coefficient of these areas is relatively large.

Conclusion

From the numerical results, it be concluded that the total deformation is relatively large, it still has a certain risk although pre-bending structure of the blade offset most of the deformation. The maximum tensile stress and compressive stress are calculated to be much smaller than the experimental strength of the composite material, the maximum value of interlaminar shear stress is smaller than allowable stress. While, the beam and the trailing edge of the blade have relatively larger stress concentration. Even so, the designed blade is safe in the operation.

References


