Numerical simulation and parameter analysis of the strength of a wind turbine main shaft

Yu Xing Wu*, Zhun Bei Zheng, Zhan Jun Yang

Metal Technology Research Lab, Northwest Branch of China Datang Corporation Science and Technology Research Institute Co., Ltd., Xi’an ShaanXi 7100231, China

* xibeisuo@cdt-kxjs.com

Yu Xing Wu: 646165562@qq.com

Abstract: With the vigorous development of wind power generation worldwide, the problem of stress concentration at the position of the main shaft shoulder of the wind turbine has become increasingly prominent. Therefore, a finite element model was established for the main shaft of a wind turbine, and the stress field of the main shaft of the wind turbine under the limit load condition was simulated and calculated, and the influence of the position of the shoulder on the flange side on the stress concentration under different radii of curvature was analyzed. It provides important guidance for the manufacture and inspection of the main shaft of the wind turbine.

1. Introduction

Wind power has attracted worldwide attention because of its advantages such as no need for fuel, no pollution, low operating cost and no radiation. The production and manufacturing level of wind turbines is an important factor to measure the development level of a country’s wind power. The main shaft of the wind turbine is an important part of the wind turbine [1]. Once the main shaft has structural damage during work, the entire wind turbine will not be able to operate, so in the process of wind turbine strength analysis and nondestructive, the strength analysis of the wind turbine main shaft is particularly important.

At the same time, since the size of the main shaft of the wind turbine is affected by many aspects such as machining accuracy and working environment [2], and the size of the shaft shoulder of the main shaft of the wind turbine has a certain influence on the stress concentration, the quantitative analysis and calculation of the size on the stress concentration It is necessary to get a reasonable shoulder size.

Because the structure of the main shaft of the wind turbine is relatively complicated, it is difficult to obtain an accurate solution using the method of theoretical derivation. The finite element method is currently the most extensive numerical calculation. Some scholars used finite element software to analyze the structural strength of the hub and nacelle base, and discussed the finite element modeling methods of components [3-6]. In this paper, the professional finite element software is used to solve the stress field of the wind turbine main shaft structure under extreme load conditions. Using the finite element method to analyze the main shaft of the wind turbine is of great significance for its structural design optimization [7]. Through the finite element calculation results, the weak position of the main shaft of the wind turbine was found, which made an important foundation for the structural failure analysis and detection of the main shaft of the wind turbine. At the same time, the stress field of the shaft shoulder on flange side at different radius of curvature is also analyzed and calculated, which provides an important guidance for the manufacture of the wind turbine main shaft.
2. Establishment of finite element model

2.1. Finite element meshing dividing
This model uses the C3D8R element (8 node hexahedral linear reduction integral element). In order to calculate the stress value of the shoulder position more realistically, the local mesh encryption is performed on the shoulder position. The quality of the finite element mesh was checked and the mesh quality was excellent. The finite element mesh diagram is shown in Figure 1 and Figure 2.

![Figure 1. Mesh of wind turbine main shaft](image)

![Figure 2. Mesh of bearing](image)

2.2. Geometric parameters and material properties
The geometric parameters of the main shaft are derived from the manufacturer's design drawings. The material density of the main shaft is 7850 kg/m³, the elastic modulus is 210 Gpa, and the Poisson's ratio is 0.3, the material is 34CrNiMo6 and the yield limit is 800 Mpa.

2.3. Boundary conditions
The contact relationship between the bearing close to the flange side and the main shaft is established. The normal direction in the contact properties uses hard contact, so as to avoid the penetration of the contact surface of the main shaft and bearing as much as possible. Penalty friction formula is used for tangential direction, and the friction coefficient is 0.1. Limit the translational and axial rotation of the outer surface of the bearing. Limit the translation and axial rotation of the contact surface between the main shaft and the gear box.

2.4. Constraints and loads
In the strength analysis, it is not necessary to load all the working conditions for calculation, because the occurrence probability of some working conditions is very small, or compared with the load value in other working conditions is much smaller, so in order to reduce the calculation cost, increase Work efficiency, usually select some working conditions with obvious characteristics for calculation. The extreme load required by the wind turbine main shaft refers to the paper data, the load data are as follows:

\[
M_x = 5523.7\, KNm, \quad M_y = 2138.3\, KNm, \quad M_z = -4585.0\, KNm \\
F_x = -100.2\, KNm, \quad F_y = 143.4\, KNm, \quad F_z = -515.7\, KNm
\]

In order to improve the accuracy of load transfer, a reference point is created at the center of the hub, and a distributed coupling constraint is established between the point and the connecting surface of the hub and the main shaft, and the ultimate load required by the main shaft of the wind turbine is applied to this reference point.

3. Verification of Finite Element Convergence and Calculation results
The finite element method is a numerical analysis method, so the problem of convergence must be considered in the calculation. The convergence of the finite element method means that when the meshing becomes more and more dense, the finite element calculation results are closer to the exact solution, and the stress change range is getting smaller and smaller, which is close to convergence. But
within the allowable range of the result error, reducing the number of units can save computing resources. The main shaft and bearing of the wind turbine in this model are both flexible structures, and mesh generation is required, and verify the convergence of finite element. After mesh refinement, the maximum equivalent stress value of the main shaft is changed from 381Mpa to 393Mpa, and the change rate of the stress calculation result is only 3.05%, which is within the allowable error range, which verifies the convergence of the finite element in this simulation. The total number of elements after mesh refinement is 52917. The stress cloud diagram is shown in Figure 3, 4, 5.

From the Stress nephogram, the stress distribution on the surface and inside of the main shaft of the wind turbine can be clearly seen. The maximum Mises equivalent stress on the main shaft is 393Mpa, which appears on the front shoulder of the flange side, which meets the expected assumption.

4. Parameter analysis
Because the size of the structure at the position of the shoulder on the flange side varies greatly, there is a problem of stress concentration. Therefore, this chapter calculates the stress distribution at different radius of curvature of the shoulder near the flange, and analyzes the effect of the shoulder at different radius of curvature on the stress concentration. The radii of curvature were taken as 0.01m, 0.02m, 0.05m, and 0.1m, the schematic sketches with radius of curvature 0.02m and 0.1m are shown in Figure 6 and Figure 7.
Figure 6. The radius of curvature is 0.02m

Figure 7. The radius of curvature is 0.02m

Calculate and draw the stress cloud diagram when the radius of curvature is 0.01, 0.02, 0.05, 0.1, as shown in Figure 8, 9, 10, 11.

Figure 8. The radius of curvature is 0.1m

Figure 9. The radius of curvature is 0.05m
Figure 10. The radius of curvature is 0.02m

Figure 11. The radius of curvature is 0.01m

It can be seen from the stress nephogram that the peak stress is 393Mpa when the radius of curvature is 0.1m. When the radius of curvature is 0.05m, the peak stress is 414Mpa. When the radius of curvature is 0.02m, the peak stress is 456Mpa. When the radius of curvature is 0.01m, the peak stress is 623Mpa.

The change law of peak stress with radius of curvature is as follows:
Figure 12. Variation of peak stress

The smaller the radius of curvature at the flange side shoulder, the larger the peak stress at the flange side shoulder. Moreover, in the range of radius of curvature within 0.02 meters, as the radius of curvature decreases, the peak stress increases significantly, that is, the stress concentration is very obvious. It can be concluded that the radius of curvature at the shoulder of the flange side has a great influence on the stress concentration, which provides important technical support for the manufacture and maintenance of the wind turbine main shaft.

5. Conclusion
Aiming at the problem of stress concentration at the position of the main shaft shoulder of a wind turbine, a finite element model was established for the main shaft of a wind turbine. The stress field of the wind turbine main shaft under the ultimate load condition is simulated, the weakest position of the wind turbine main shaft is found, and the influence of the flange shoulder position on the stress concentration under different radius of curvature is quantitatively analyzed and calculated. It provides important guidance for the manufacture and inspection of the main shaft of the wind turbine.

References
[1] Du J, Niu X, He Y, et al. Fatigue analysis of mainshaft of MW level wind turbine[J]. Acta Energiae Solaris Sinica, 2013, 34(4):591-597.
[2] Xinjian Z, Guangxing L I, Zhiqiang L I. STRUCTURAL STRENGTH AND RELIABILITY ANALYSIS OF MW SCALE WIND TURBINE'S MAIN SHAFT[J]. Journal of Mechanical Strength, 2019, 41(02):349-355.
[3] Chuan-Zong S, Xing-Jia Y, Guang-Kun S, et al. Strength analysis of MW wind turbine hub[J]. Journal of Shenyang University of Technology, 2008, 30(1):46-49.
[4] Xing-Jiaa Y, Li-Dongb Y, Guang-Kunb S. Finite element analysis of wind turbine hub based on MSC.Patran[J]. Journal of Shenyang University of Technology, 2010, 032(006):620-624.
[5] SHAN Guang-kun, YAO Xing-jia. Mode analysis on MW grade wind turbine[J]. Journal of Shenyang University of Technology, 2008, 30(003):276-279.
[6] Xing-Jiaa Y, Li-Dongb Y, Guang-Kunb S. Application of super-element method in finite element analysis of hub[J]. Journal of Shenyang University of Technology, 2011, 33(1):31-35.
[7] Du J, Niu X, He Y, et al. Fatigue analysis of mainshaft of MW level wind turbine[J]. Acta Energiae Solaris Sinica, 2013, 34(4):591-597.