Advanced Materials Research
ISSN: 1662-8985, Vols. 455-456, pp 187-193
doi:10.4028/www.scientific.net/AMR.455-456.187
© 2012 Trans Tech Publications, Switzerland

Online: 2012-01-24

Contact Simulation and Analysis on Planetary gear of Megawatt Wind Turbines Yawing Reducer

E Jia-qiang, DONG Jiang-dong, LIU Guan-lin

College of Mechanical and Vehicle Engineering, Hunan University, Changsha, Hunan, China

Email: ejiaqiang@126.com, Email: dongjiangdong1@126.com

Keywords:wind turbines yawing reducer; planetary gear; finite element; contact analysis; engagement pairs

Abstract.According to the operation condition of megawatt wind turbines yawing reducer, an assembly model of megawatt wind turbines yawing reducer is established, and aiming at different load conditions, the tooth surface of three teeth planetary gear model is analysised with using finite element analysis software-ANSYS ,including three-dimensional nonlinear contact analysis and structural finite element analysis which simulates contact stress of engagement pairs during the operation process of planetary gear. The results indicated that when the torque is M_1=100N·m, the maximum stress on the engagement pairs is 177.584MPa; when the torque is and M_2=300N·m, the maximum stress on the engagement pairs is 329.607MPa, which were less than the yield limit of planetary gear materials, but it can meet the design and operation requirements of planetary gear of megawatt wind turbines yawing reducer.

Introduction

In recent years, with the continuing shortage of energy and deteriorating of ecological environment, wind energy, as the most promising development and utilization of a renewable and clean energy, which is paid more and more attention to. China's wind energy resource is very rich and wind power technology is becoming mature daily, the development of wind power is also very fast. At the end of " Twelfth Five-Year Plan ", according to the present speed more than 10 million kilowatts per year of new installation capacity growth rate, China's wind power installation capacity is expected to reach 90 million kilowatts to 100 million kilowatts, by 2020,wind power capacity will reach 150 million kilowatt.

The new turbines yawing reducer is transmissed by using of planetary and push rod pin wheel live gear. Planetary transmission is a typical low-speed ,heavy-duty and variable-torque transmission. The failure of planetary gear is about 40% of the failure of the whole gear box[1] ,which is one of the main reasons in the gearbox failure. During the gear transmission process, the teeth surfaces receive the alternative contact stress. After more than certain limit load cycles, the teeth surfaces are easy to form metal surface pitting or spalling, resulting in fatigue failure.

With the rapid development of computer technology, some scholars analyze the factors which impact nonlinear analysis in the finite element analysis software [2-3] . The analysis of gear tooth profile is carried out in the reference [5-6] by the finite element analysis software. Lee[7] analyzed dynamic contact question between the engagement gear teeth by using the finite element and multi-body dynamic technology. In order to improve the capacity of bearing, ensure the reliability of gearbox's operation and prolong service life, it is necessary to consider,comprehensively,the influence of gear strength factors, and to design the gears considering this influence. By using finite element analysis software-ANSYS, the contact strength of megawatt wind turbines yawing reducer is analyzed in this paper. Using the contact analysis of Ansys,the touch can be identified automatically ,and the gap closed can be realized. According to tooth contact, the loads are allocated more close to the actual situation.

The contact stress calculation of gear based on traditional theoretical analysis

The tooth surfaces of the involute gearing is sophisticated shape. However, the contact dimensionless width is far less than curvature radius at the point of contact, mating flanks can be simplified. Week et al[8] concluded that the mesh of two gears could be convertible into contact of two cylinder along their bus as shown in Figure 1, and the radius of two cylinders were equal to curvature radius of tooth surface.

Under the action of normal pressure(F_n), the rectangle interface whose width is 2b and length is L is formed as shown in Figure.1.

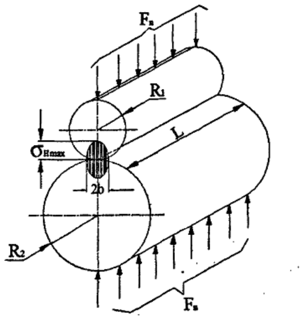


Figure 1. Schematic diagram of standard spur cylindrical gears in contact

The half Hertzian contact width can be compressed Eq.(1):

$$b=\sqrt{\frac{4F_n}{\pi L}\times(\frac{1-v_1^2}{E_1}+\frac{1-v_2^2}{E_2})/(\frac{1}{R_1}+\frac{1}{R_2})} \tag{1}$$

Where, E_1,E_2 are elastic modulus, v_1,v_2 are Poisson radio, and R_1,R_2 are radius of two cylinders.

The force applied to contact surface is different everywhere and the distribution of pressure vector is a semi-ellipsoid. Maximal pressure happened in the points at initial contact wire is twice as large as the mean pressure. If contact stress is σ_{Hmax}, the combined strength is $\sigma_{\mathrm{Hmax}}bL/2$. The stress on interface should be balance with external force(F_n), so:

$$F_n=\frac{\pi\sigma_{\mathrm{Hmax}}bL}{2} \tag{2}$$

$$\sigma_{\mathrm{Hmax}}=\frac{2F_n}{\pi bL} \tag{3}$$

The fundamental formula of contact stress is obtained after Eq.(1) substituted into Eq.(3):

$$\sigma_{\mathrm{Hmax}}=\sqrt{\frac{F_n}{\pi L}\cdot[(\frac{1}{R_1}+\frac{1}{R_2})/(\frac{1-v_1^2}{E_1}+\frac{1-v_2^2}{E_2})]} \tag{4}$$

$$\frac{1}{Z_E^2}=\pi\left(\frac{1-v_1^2}{E_1}+\frac{1-v_2^2}{E_2}\right),\qquad R=\frac{R_1R_2}{R_1+R_2}$$

So, Eq.(4)can be simplified Eq.(5):

$$\sigma_{H\max}=Z_E\sqrt{\frac{F_n}{L}\cdot\frac{1}{R}} \tag{5}$$

Where, L is width of mating gear, F_n is normal force of its contact surface, Z_E is material coefficient of mating gear, R is comprehensive curvature radius.

The structure of gear and its material properties are shown in Tab.1.

Tab.1 Related parameters of planetary gears with three grade

Teeth number of central gear	23	Teeth number of planet wheel	35
Modulus of central gear	2	Modulus of planet wheel	2
Pitch diameter of central gear	46	Pitch diameter of planet wheel	70
Engaging angle of central gear	20	Engaging angle of planet wheel	20
Gear material	20CrNi3	Poisson ratio	0.295
Elasticity modulus	2.08×10^5MPa	Material density	7800kg/m^3

Building the finite element model of planetary gear of wind turbines yawing reducer

In this paper, finite element analysis software ANSYS is used for contact analysis. Geometry model drawn by Pro/e is based on actual structure and size, and the first-class planetary reduction gear mechanism is analysed.

Building the assemble model of planetary gear reduction mechanism. This planetary gear reduction mechanism is the first-class reduction mechanism situated in the gear box of 1.5 megawatt wind turbines yawing reducer. The mechanism consists mainly of one planet carrier, one sun wheel, one ring gear,four planet wheels,four gear shafts and so on,as shown in the Figure .2. Parametric modeling makes structural adjustment much convenient.

The gears of the planetary mechanism are all involute gears, so the edge shape of these gears are all Involute curve. The model of these gears are building by parametric technique. First, a parametric involute curve is drawn, then 2D model is generated by sweeping and trimming, and 3D model is created by extruding along the direction of tooth thickness. In the end , local features of the 3D model will be adjusted.

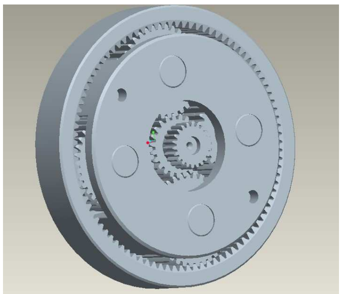


Figure 2 Assembly Drawing of The Planetary Gear Reduction Mechanism

Building the finite element model. Assembly drawing of sun wheel and planet wheel is exported as IEGS file, then the file is imported to ANASYS, to be splited and modified.

Contact analysis is a nonlinear computation and need lot's of resources, so appropriate simplification of the gear model is prerequisite in the modelling phase. The model should be described precisely, and computational efficiency should be considered at the same time. The following processing methods are adopted.

Three teeth-meshing model is utilized. Two gear teeth mesh with each other at a time, so other teeth aren't weighted. In order to reducing the amount of calculation, three teeth-meshing model is utilized to replace the original model. Celik compared the two stress and strain results which were calculated by the three teeth-meshing model and original model, and found that the calculation result of original model was more close to the real situation. Because the error between the two results is less than 2%, it's acceptable that three teeth-meshing model is utilized to replace the original model[9].

Non-tooth profile edge is cut to 35mm, the longitudinal length is 7mm to the root of tooth. Chabert found out it's reasonable or not based on saint venant principle[4], he pointed that the displacement of the edge after Reducing isn't more than 3.5% of the loading point's. Based on relative references abroad[10-11], the displacement on the border is less than 3% of the least displacement when the lateral length of non-tooth profile edge is nine times of modulus and the longitudinal length to the root of tooth is 1.5 times of modulus. At this rate, regarding this displacement as zero doesn't influence the stress and strain results.

Mesh division of finite element model . Using 8-node and PLANE42 unit of SOLID45 to create the mesh. In contact analysis of gear, because the tooth surface deformation is very small, SOLID45 unit can get very accurate results of mesh division. It is unnecessary to choose more advanced SOLID95 unit to create the mesh.

Mesh will have a direct bearing on the accuracy of FEA results, and accurate results depend on finite element conditions of the mesh mapping generation and generate the plane quadrilateral mesh, gear face is divided into several pieces first, and all the opposite edges are joined by applying Concatenate, then the quadrilateral mesh is stretched by manual control. In addition, the mesh of tooth surface contact area and root area should be divided relatively densely. It is proved that mapping method can divide the gear into high-quality hexahedral mesh. Relying on the ways offered above, the sun gear and the planet gear can be divided into 32261 units, the finite element meshing model is shown as figure 3.

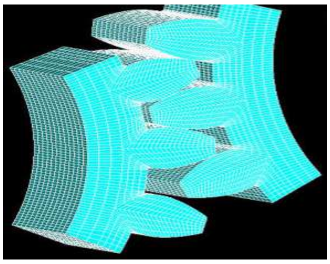


Figure.3 Finite element meshing model of three pairs of teeth

The establishment of contact pair. When define the contact, first determining the nature of contact that contains friction or no friction. Through relative literature reading, it is found that the influence that friction between the teeth bring to gear root stress can not be ignored. So, the friction coefficient is defined as 0.3 according to the gear material, then finish the definition of contact pair by face to face contact. When define the face to face contact, two surfaces in contact can be divided into active and passive one, and the selection of active or passive surface direct influence the result show. Therefore, using face to face contact in this model, take the sun gear surface as the target unit and take the planet gear surface that is contacted to the sun gear as the contact unit, then define contact. Contact174 is used in contact unit, and Target170 is used in target unit. Contact pair model can be established as shown in figure 4.

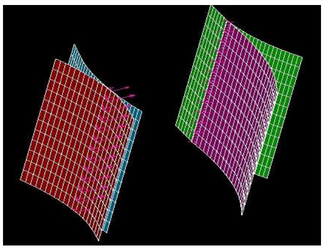


Figure.4 Contact pair model

Loading of boundary conditions and loads

In this paper, static analysis is used in contact analysis of gear. At any moment, viewing gear meshing as quasi-static process, the part away from the tooth of driven gear is stationary and its displacement can be regarded as zero when it doesn't feel the driving effects of the driving gear. So, imposing the full constraints on the nodes on the inner hole surface of the planet gear as well as the spoke boundary in order to limit the rotation of the planet gear. In addition, taking the sun gear as driving gear, and impose the axial and radial constraint on the nodes on the inner hole surface and the spoke boundary with reserving the circumferential orientation constraint.

Because of selecting the unit type of SOILD45 that can not accept the role of torque, the torque should be changed into tangential force that act on the all nodes on the inner hole surface of the sun gear to simulate the rotation situation of gears. Specific method is:

$$F = \frac{T}{L \times N} \tag{6}$$

In this formula, T is torque, L is the distance from the rotation center of the sun gear to the inner hole surface, N is the number of all nodes on the inner hole surface.

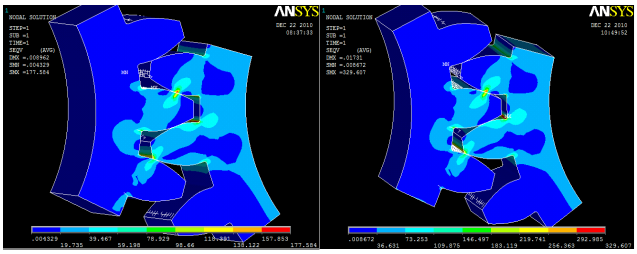
Ansys of the finite element analysis result. Torque in two conditions (M_1=100N·m and M_2=300N·m)is applied in the model, The engagement pairs are calculated and analyzed.

1) Torque M_1=100N·m

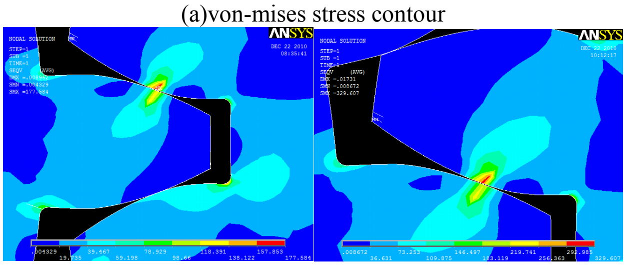
According to the formula (6), resulting the force F_1 = 7.28N which is applied on each node of the sun gear inner surface. The von-mises stress contour ,contact stress contour and contact stress along the tooth depth direction contour in the condition are illustrated in Figure 6left, the result of the simulation shows that the maximum stress is 177.584 MPa,the calculated value is far less then the yield limit of planetary gear materials which is 785MPa,and this proves that planetary gear in this condition meets the design and operation requirements.Figure 6left (a) shows that the stress on the driving wheel (Sun gear) is generally higher than that on the driven wheel (Planetary Gear) ;great stress occurs in the tooth flanks contact zone and the tooth root .

2) Torque M_2=100N·m

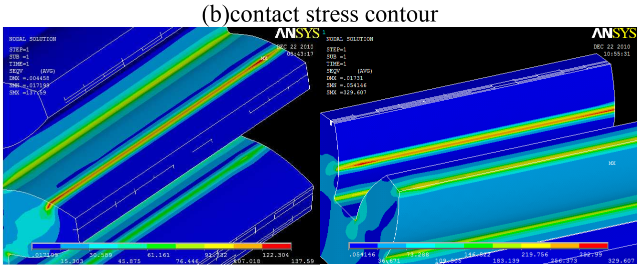
According to the formula (6), resulting the force F_2 = 21.85N. The von-mises stress contour ,contact stress contour and contact stress along the tooth depth direction contour in the condition are illustrated in Figure 6right, the result of the simulation shows that the maximum stress is 329.607 MPa,the calculated value is less then the materials'yield limit which is 785MPa,and this proves that planetary gear in the second condition also meets the design and operation requirements. And which is different from case 1 , the maximum stress occurs in the tooth root which is at the back surface of second teeth, the stress-focus phenomenon of tooth root occurs.



(a)von-mises stress contour



(b)contact stress contour



(c)contact stress along the tooth depth direction contour

Figure.5 Finite element analysis result of gear pairs in two condition

Figure 5left (a), Figure 6left (b) and Figure 6right (a), Figure 6right (b) agreed that due to the second tooth come to close engagement status, the first tooth plays a major role in engagement, the torque passed through the first tooth is greater than the second one. Therefore, as the facts, The maximum contact stress of the matrix occurs in the tooth flanks contact zone of the first sun gear tooth ;since the first tooth has already entered engagement status, the contact stress of the first tooth is bigger then that of second one ; stress in the tooth flanks contact zone of the first tooth is the biggest one ,the stress in the tooth root is also big ,Distinguished from the tooth flanks contact zone, the stress of the lower tooth is greater than that of the upper;on the longitudinal direction of tooth, contact stress is unequaled distributed along the contact line and reduced sharply around the contact line .Comparison of the two conditions, the analysis shows the maximum stress of the second condition occurs in the tooth root of the second tooth, it's due to the stress-focus phenomenon of tooth root.

Conclusion

1) Considering the two conditions, the contact analysis of gear pairs shows that the stress of gear teeth is mainly on the engaging teeth. When the gear tooth just entered engagement, the von-mises stress is relatively big; when maximum stress both occurs at the top of the engaging gear teeth and at the root of the engaging teeth, the stress is also big.

*2)*By Comparing of the two conditions, the analysis illustrates that the maximum stress of the second condition occurs in the tooth root of the second tooth, it's due to the stress-focus phenomenon of tooth root.Because the root stress distribution is an important factor for weighting gear transmission performance. Therefore,under a large and impaction load,Flexural Strength at tooth root of sun wheel is relatively the weakest. So, the tooth root strength of this gear pairs should be enhanced in order to improve the engagement life.

3) Contact stress is unequaled distributed along the contact line on the longitudinal direction of tooth,If the error of angle between the gear pairs installation and the error of stress and displacement of the planetary shelf are taken into account,this phenomenon will become more obvious,The model in this paper can be analysised under the condition which is closer to the actual gear working condition and exists various errors.

References

[1] LIU Zhong-ming,DUAN Shou-min, WANGChang-lu,"Development and Prospect of Design and Manufacture Technology of Gearboxes for Wind Tuibines,"Joural of Mechanical Transmission, vol. 30(6),pp.1-6,2006.

[2] Finite Element Contact Analysis of Meshing Gears in Gearbox of Wind Turbine Generator[D]. Dalian: Dalian University of Technology,2007.

[3] YANG Sheng-hua Applications of Substructuring and Submodeling in Calculations of the Gear Stress and Deformation [A].Engineering and Computational Mechanics in Science[C]. BEIJING:Beijing University Press,2001:701-707.

[4] LI Guanghui FEM Analysis of Engine Gear's Strength[D].Taiyuan:Taiyuan University of Technology,2007.

[5] Litvin L. F., Fuentes A., Perez G. I.,"Modified involute helical gears:computerized design, simulation of meshing and stress analysis,"Comput. Methods in Appl.Mech.Engrg,vol.194(36-38),pp,3870-3901,2005.

[6] Wang J., Howard I.,"The Torsional Stiffness of Involute Spur Gears,"Meehanical Engineering Seienee, vol.218(1),pp.131-142,2004.

[7] Lee K., "Analysis of the dynamic contact between rotating spur gears by finite element and multi-body dynamics techniques," Proceeding of the Institution of Mechanical Engineers,vol.215(4), pp.423-435,2001.

[8] Wech M., Zruse A.Gohritz A., "Determination of surface fatigue of gears material by roller tests," J. of Mechanical Design, vol.(100),pp. 433-439.1978.

[9] Celik M., "Comparison of three teeth and whole body models in gear analysis,"Mechanism and Machine Theory,vol.(34),pp.1227-1235,1999.

[10] Chabert G., Dang T. T., Mathis R., "An evaluation of stresses and deflection of spur gear teeth under strain,"ASME Journal of Engineering for Industry, vol.96(1), pp.85-93,1974.

[11] LI Yong-xiang,BI Xiao-qin,ZHANG Jun-sun., "Loaded Contact Analysis of a StraightTeeth Face Gear by ANSYS,"Mechanical Science and Technology for Aerospace Engineering,vol.28(7),pp.931-937,2009.