Dynamic Contact Analysis for RV Reducer

Mu Rui[1, a *], Chang-zhong Hao[2,b] ,Fu Kai[3,c]

[1]Shenyang Ligong University, China

[2]Shenyang Ligong University, China

[3]Shenyang Ligong University, China

[a]781574096@qq.com, [b]hcz5613@163.com, [c]1253121317@qq.com

Keywords: the RV reducer; gear; dynamic contact analysis; the finite element

Abstract: We make a dynamic contact analysis for a pair of gears in the first stage device of the RV reducer by the 3 d modeling software Pro/E and the finite element analysis software Abaqus. The simulation results are the dynamic changes of the stress distribution on the tooth and the results are compared with the allowable values and the theoretical position. Both intuitively show the of tooth stress distribution of tooth in the RV reducer, and verify the reliability of the gear strength.

Introduction

In recent years, the demand for industrial robots has increased dramatically in our country. Last year, China has become the largest consumer of the robot. But the key parts and components, such as the RV reducer, still rely on imports. The domestic RV reducer is not only lack of manufacturers, but also far from the international level. Therefore, the analysis of its structure is an indispensable part of the study on the RV reducer.

The RV reducer is a closed differential gear train, composed of two part, they are the planetary gear transmission and the cycloid gear transmission. The load surffed by gear in operation is big and the stress caused by it is complex. Traditional methods for analysis of gear strength are analytical calculation method and experimental analysis. Experimental analysis is of high cost and long cycle; analytical calculation method is based on the empirical formula,which is brought into the gear's parameters and coefficients, however, these parameters are uncertain in the actual design. So the limitation and uncertainty are increasingly prominent. While, the finite element simulation method have the characteristics of low computation and high precision.

In this paper, a pair of meshing gears of the first reduction device is the research object, and the dynamic contact analysis is made for them. Considering the emphasis is tooth, and computer memory is limited, the two gears are simplified as two tooth circle. In this way, computational efficiency is improved.

Modeling

Parameters of RV reducer: input power P=4.28kw, output speed n=5r/min, transmission ratio i=81.222, geometry parameters of gears in the first level are shown in table 1.

Table1 The Gear Geometry Parameters

Number of tooth z_1/z_2	Module m	Pressure angle α	Tooth breadth b_1/b_2	Addendum coefficient ha^*	Tip clearance coefficient c^*	Modification coefficient X
18/38	3mm	20°	18/14mm	1.0	0.25	0

Due to the structure of gear is relatively complex, the complicacy of modeling in Abaqus must be avoided. So, the parameterized modeling method in Pro/E is selected.

(1) The establishment of the gear model

First of all, after the geometry parameters was inputted, root circle, base circle, reference circle and addendum circle can be drawn; secondly, the involute formula must be inputted into [the cartesian

coordinate system] so that we get the involute (as shown in figure 1); then, what still need to be done is roughing out the graphics of the tooth profile; finally, the gear model is established by [stretching] and [axial arraying] the tooth profile in Pro/E. Parameters of involute equation is as follows:

theta=45*t

r=db/2

x=r*cos(theta)+r*sin(theta)*pi*theta/180

z=r*sin(theta)-r*cos(theta)*pi*theta/180

y=0

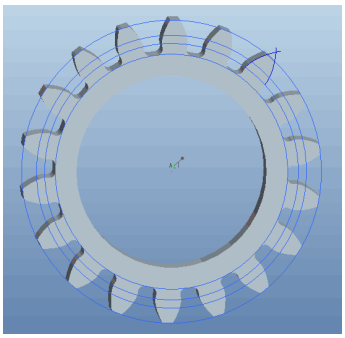

Figure1 Tooth Profile Graph

Figure2 The Non-backlash Mesh

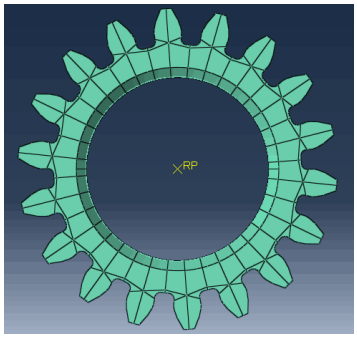

Figure3 Gear's Subdivision

(2) the assembly of the gears

The first thing is to build two datum axis whose horizontal distance is 84mm in the assembly environment; following that, gears are added into the assembly part with the method of pin connection to make sure that the gears' center axises and the two axises above are coincident. Besides, the gears' middle surfaces plumbing to the topland must be center align with each other; then, the non-backlash mesh will be come true (as shown in figure 2).At last, interference detection for the final assembly mode show that there are no interference of parts, assembly is completed.

The Preprocessing for the Analysis Based on Abaqus

The fabricated 3 d model is saved as the '.stp' format and imported into the Abaqus.

(1) Material

Material of gears is 20Cr, elastic modulus is 2.07×10^{3}MPa, poisson's ratio is 0.254,the density is 7.83g/cm^3. Two entities uniform cross section are defined and assigned to the gears.

(2) Mesh

The rationality of mesh affect the calculation results and the computational efficiency, so, each tooth is divided into four parts in this article (as shown in figure 3). That is convenient for mesh refinement. According to the 'scanning' method, the explicit C3D8R hexahedron unit is selected. Total number of units is 89240.

(3) Interaction

The 'surface-to-surface explicit contact interaction' is built by the finite sliding kinematic method. The pinion's surface is active surface, and the big gear's is driven surface; and the two gears don't invade to each other. Contact interaction property is: 'hard' contact, friction coefficient is 0.1.

(4) Load

Gears couple with the central points (as shown in figure 4), thus, they form two linkage bodies. So, both the constraint and load can be added to the points. Gears rotate around the center axis (Y), the remaining five degrees of freedom should be limited. Besides, the speed (UR2=43.773) and the torque (CM2=200.23) are applied to the pinion and driven wheel. Total analysis time is 0.04 S, and, the load achieves the maximum and hold constant in 0.007 seconds.

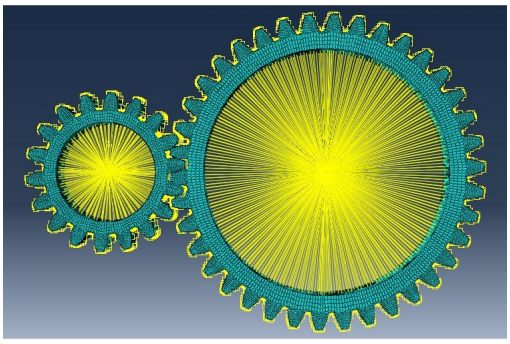

Figure 4 Kinematic Coupling

The Gears' Dynamic Contact Analysis

(1) The stress distribution of tooth

Stress on the RV reducer is complex. In order to guarantee the operational reliability of the gear, the stress distribution of gear tooth need to be understood, in addition to the maximum stress value. So, figure 8 extracted from one mesh cycle of teeth 1 and 1'clearly shows changes of the stress distribution. What we get is that with the change of the contact positions, stress's action site, working area, maximum value and its position are also mutative.

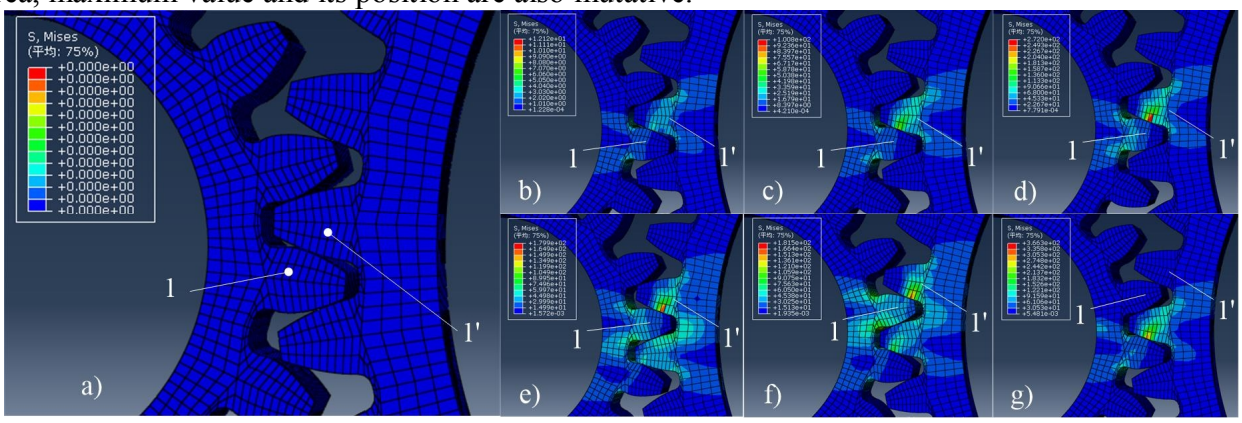

Figure 5 The stress distribution of teeth 1 and 1'

(2) Contact stress

Coincidence of the two gears is 1.572, and the maximum contact stress usually occurs in the process of single tooth meshing according to the theoretical knowledge. And the contact stress nephogram of dynamic analysis shows that calculation results accord with the above. In the process of single tooth meshing, the maximum contact stress occurs at the pitch circle of the tooth surface along the thickness direction (as shown in figure 6). The maximum contact stress is 1022.37MPa, and it is smaller than the allowable value (σ_{HP}=1281.123MPa). Besides, the contact stress of any node increases to maximum instantly and then quickly reduces to zero in the short contact (as shown in figure 7).

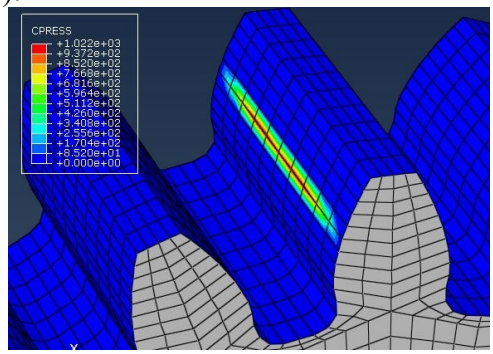

Figure 6 Nephogram of Maximum Contact Stress

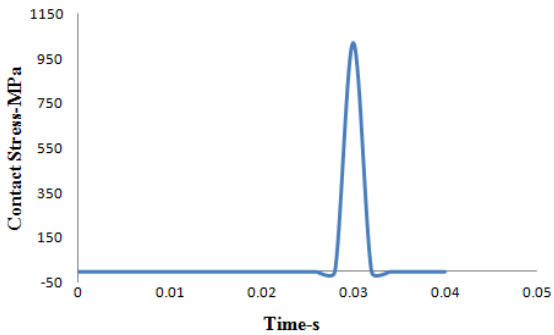

Figure 7 Contact Stress Curve

(3) Bending stress

When tooth in the double tooth meshing area, the load is borne by two pairs of teeth. Although arm of force is big, the root of bending moment is not the biggest. So, only in the single tooth meshing area, root of tooth get the maximum bending moment. Figure 8 is the nephogram of tooth 1 when it bear the maximum bending stress, and bending stress of a node located in the center of dedendum varies with the contact time(as shown in figure 9). Based on those, it is find that the maximum bending stress is 151.981 MPa, and it is less than the allowable bending stress (σ_{FP}=524.515MPa).

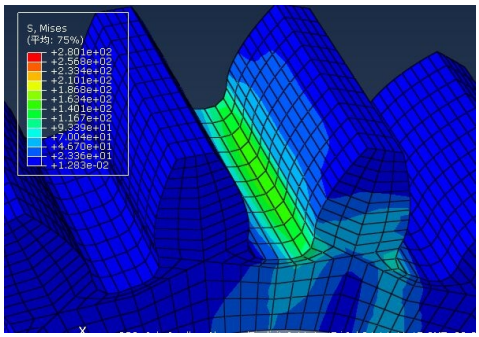

Figure 8 Nephogram of Maximum Bending Stress

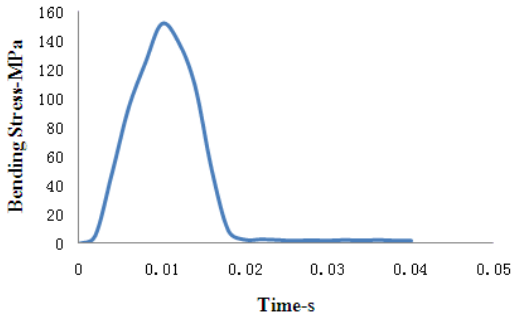

Figure 9Bending Stress Curve

Conclusion

First, the three-dimensional models of gears which belong to the first stage device of RV reducer are finished with the parametric modeling method in Pro/E. Second, two gears are assembled without interference by cam connection. Third, using dynamic explicit method, a dynamic contact analysis whose duration is 0.04s for the two gears is made. Finally, the stress nephogram of meshing is got, and, at the single tooth meshing area, the maximum contact stress and the bending stress which are occurred at the pinion's pitch circle are smaller than the allowable value. So, Gear strength is proved to be reliable, and this article provides important basis for RV reducer's further study.

Reference

[1] Zhi-li Sun etc. Mechanical design. Shenyang: Northeastern University Press (2011).

[2] Xiu-shan Wang, Shu-min Zhang. Precise modeling and assembly of virtual gear based on Pro/E, J. Journal of China Agricultural University, 2004(9).

[3] Sheng-hua Yang. Finite element analysis of gears' contact, J. Journal of Computational Mechanics, 2003,20(2).

[4] Chen jing, Wen-ku Shi. Simulation analysis of dynamic process for reverse gear transmission system of gearbox, J. Journal of Mechanical Design, 2009,26.