Research on the Efficiency of Ball Mill Based on Finite Element Analysis

Zhiqiang Xu

Department of Physics and Technology, Kunming University, Kunming, 650214, China

2793436659@qq.com

Abstract: As large-scale mills play an important role in the beneficiation process, people pay more attention to improving the efficiency of maintenance and repair of them. It is necessary to minimize downtime to improve the operating rate of large mills, obtaining greater economic benefits. The liner is a consumable part of a large mill, which consumes a lot of money every year. Yet replacing the liner consumes most of the downtime. The semi-automatic lining process can be realized by the lining manipulator, in which the worn liner is removed and a new liner is placed skillfully on the mill liner system. This paper mainly analyzes the relining manipulator of Φ6500×7500 ball mill, and carries out CATIA finite element analysis on the gear strength of a certain transmission end.

1. Introduction

The work of the ball mill is to pulverize and mix materials by the impact of the falling abrasive body (such as a steel ball, etc.) and the grinding action between the abrasive body and the inner wall of the ball mill. When the ball mill rotates, the abrasive body is brought up in the direction of rotation and then drops due to the friction between the grinding body and the inner wall of the ball mill. The materials are continuously crushed in this way. With the construction of large mines, especially large open pit mines, the application of efficient, simple and economical grinding technology makes the utilization and development of various large and efficient mills the main trend of today's plant selection [1, 2]. However, replacing the mill liner consumes most of the downtime. Especially for large mills, the liners are numerous and heavy, the manual methods can not meet the needs of the relining. It takes too much time and a lot of manpower to finish the relining. I addition it is extremely unsafe. Therefore, the relining manipulator becomes an auxiliary equipment that must be equipped for large mills. The practical application of the relining manipulator brings great economic and social benefits to the concentrator [3, 4].

2. Finite Element Analysis

In the engineering field, for many mechanical problems, there are corresponding basic equations and boundary conditions, which can be solved by analytical methods, but only for a few problems where the properties of the equation are relatively simple and the geometric boundaries are fairly regular. However, the engineering problems encountered in real life are quite complicated: the geometry of the object is complex, and some features of the problem are nonlinear. So there are few analytical solutions. A new numerical method- the finite element method, has emerged in the last three decades. Its basic idea is to discretize the continuous solution domain into a combination of a finite number of elements, which can resolve the simulated solution domain or the approximation solution domain. The discrete units are connected to each other by the nodes of the unit. The setting, nature and number of the unit nodes are determined according to the specific situation. Generally speaking, the finer the unit division is, the closer to the actual situation will be, and the more accurate the calculation result will be [5, 6].

CATIA is a 3D parametric design software launched by Dassault, France. It can be used for 3D mechanical design, mechanical manufacturing and engineering analysis. It has a unified user interface, data management and application programming interface, which absorbs and integrates the characteristics of other excellent 3D softwares. Engineering analysis in CATIA refers to finite

element analysis, including static analysis and dynamic analysis. Dynamic analysis is divided into a limited state natural frequency analysis (Frequency Analysis) and a free state natural frequency analysis (Free Frequency Analysis). The former imposes a certain constraint on the object, the latter object has no constraints, that is, completely free [7, 8].

The finite element analysis functions in CATIA V5 include the finite element analysis GPS (Generative Structural Analysis) for individual parts, the Generative Assembly Structural Analysis (GAS) for assembly parts, and their CATIA Elfini Structural Analysis.

Usually the process of finite element analysis is roughly divided into three steps: pre-processing, solution and post-processing. The steps of CATIA finite element structural analysis are roughly as follows.

(1) Establish a solid model of finite element analysis (including 3D solid model and 2D surface model).

(2) Add information about the mechanical structure system, such as the material properties of the model, the unit properties, the necessary virtual parts, the additional mass, the connection relationship of the defined components, and the connection characteristics to the model.

(3) Apply a "displacement" boundary condition - constraint. The boundary condition of this displacement is the most essential description of the environment in which the mechanical structural system is located.

(4) Define the load, which is the "force" boundary condition of the mechanical structural system.

(5) Solve the selected analysis conditions, calculating (including automatic mesh division), solve the equation and generate stress strain results.

(6) View the analysis calculation results, including cell grid, stress or deformation display.

(7) Refine the analysis results. If the analysis results do not meet the accuracy, you can refine the grid or change the unit type to re-analyze, or use the adaptive solution method until the analysis results that meet the engineering accuracy requirements are obtained.
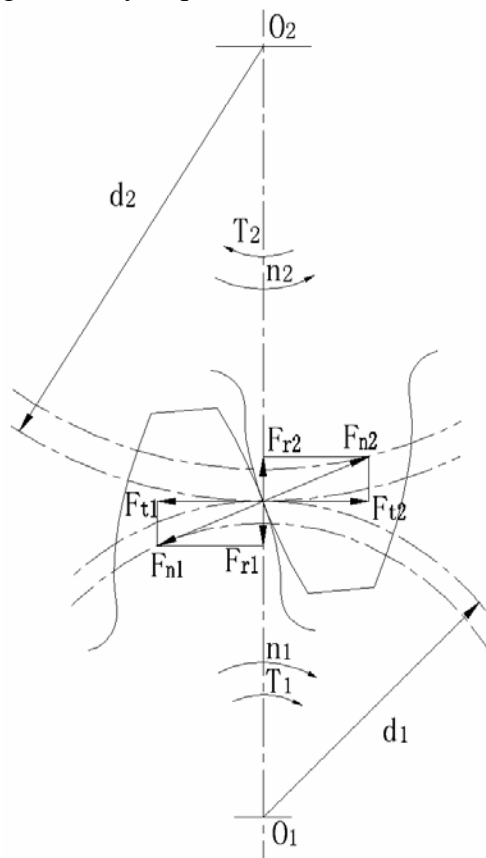

Figure 1 Force analysis of spur gear transmission

3. Static Strength Analysis of Gears

The CATIA V5 R17 GPS module is used to perform finite element strength analysis on the gears at the end of the grip.

First, the gear force analysis is performed. According to the design parameters: the arm rotation speed 2r/min; the hydraulic system maximum working pressure: 20MPa; the maximum working flow of hydraulic system: 62 L/min.

The CMG 2080 external gear motor is used. Its parameters are as followed: pressure 16/20MPa, displacement 80.4ml/r, theoretical torque 201.0N·m.

The normal force F_n acting on the surface acts on the surface in the direction of the mesh line at the node. On the index circle, the normal force F_n is decomposed into two mutually perpendicular component forces: the circumferential force F_t and the radial force F_r.

$$F_t = \frac{2T_1}{d_1}$$

$$F_r = F_t \tan\alpha$$

$$F_n = \frac{F_t}{\cos\alpha} = \frac{2T_1}{d_1 \cos\alpha}$$

Where: T_1 - the nominal torque transmitted by the pinion, N·mm;

n_1 —Pinion speed, r/min;

d_1 —Pinion index circle diameter, mm.

The arm rotation speed is 2r/min, that is, the slewing ring speed is 2r/min. It can be seen that the rotation speed of the slewing external gear is also 2 r/min ($n_2 = 2$ r/min). The model of the slewing bearing: 011.020.450.401. The modulus of the external gear is found to be 5 mm, the diameter is $\Phi 540$ mm, the pinion gear modulus is 5 mm, and the number of teeth is 23, and the diameter is found to be $\Phi 115$ mm.

Since the pinion gear is connected to the hydraulic motor through an accelerator. The torque transmission efficiency is 95%, the torque transmitted to the pinion gear is:

$$T_1 = 201.0 \times 95\% = 190.95 \,\text{N·m} = 190950 \,\text{N·mm}$$

Therefore, the surface force of the gear is:

$$F_{t1} = \frac{2T_1}{d_1} = \frac{2 \times 190950}{115} = 3320.87 \,\text{N}$$

$$F_{r1} = F_{t1} \tan\alpha = 5217.57 \times \tan 20° = 1208.70 \,\text{N}$$

$$F_{n1} = \frac{F_{t1}}{\cos\alpha} = \frac{2T_1}{d_1 \cos\alpha} = \frac{2 \times 190950}{115 \times \cos 20°} = 3533.99 \,\text{N}$$

The CATIA static strength analysis of the gears is then performed.

(1) Establish a solid model of finite element analysis: establish a three-dimensional model of the gear in the 3D solid modeling module.

(2) Add the material properties of the model: select Metal→Steel, set the display mode to Shad.

(3) Apply constraints: select the menu [Start]→[Analysis and Simulation]→[Generative Structural Analysis]. A new analysis example dialog is popped up, select Static Analysis in the dialog box, and click the OK button. A new analysis example can be conducted. Because the root strength analysis is only performed on the gear, the inner side of the inner spline of the gear is directly restrained. When the restraint is applied, directly constrain the inside of the internal spline

of the gear.

(4) Defining the load: it is assumed that only one tooth is under stress at the same time. According to the coordinate system of the model, it is limited to the lack of the model itself, and the force cannot be loaded onto the index circle. Therefore, only the force can be applied to the addendum circle, and the corresponding load is applied in the X-axis and Y-axis directions.

(5) Calculate the solution: click the button to modify the size of the mesh of the tooth root, and then click the button to start the calculation.

(6) View the analysis results: click the button to display the grid distribution map; click the button to view the stress cloud map.

4. Conclusion

The maximum bending stress at this time is obtained from the stress analysis result: $\sigma_F = 148$ MPa. According to the allowable bending stress calculation formula of the spur gear:

$$[\sigma]_F = \frac{\sigma_{F\lim}}{S_{F\lim}} Y_N$$

where：Y_N —The life coefficient of bending fatigue strength is obtained by reference [9], Figure 6-21 on page P131, and the gear material is $40Cr$. Suppose $N = 10^6$, and we have $Y_N = 1.05$；

$\sigma_{F\lim}$ — The bending fatigue limit of the test gear is checked by reference [9], Figure 6-22 on page P132, and $\sigma_{F\lim} = 500$ MPa when the alloy steel is quenched and tempered;

$S_{F\lim}$ — The minimum safety factor for the calculation of bending fatigue strength is taken for general gears and most industrial gears according to general reliability requirements. Taking $S_{F\lim} = 1.25$.

It can be obtained:

$$[\sigma]_F = \frac{\sigma_{F\lim}}{S_{F\lim}} Y_N = \frac{500}{1.25} \times 1.05 = 420 \text{ MPa}$$

Because σ_F is much smaller than $[\sigma]_F$, it can be stated that the bending fatigue strength of this gear is sufficient.

References

[1] Jiawei Liu, Min Feng, Huaidong Yang. Development of XJJ-5 Ball Mill Replacement Relining manipulator [J]. Mineral Processing machinery. 1994, 01: 11-14, 16.

[2] Huaidong Yang. Application and Structural Features of Milling Machine for Milling Machine [J]. Design and Research of Nonferrous Metallurgy, 1995, 03: 43-48.

[3] Jun Zhu, Jun Yang. Research and Application of Large Ball Mill Liner Material [J]. Foundry Technology, 2005, 12:1119-1021.

[4] Zhenming Yu, Peng Wang. Problems in the Application Research of Ball Mill Liners in China [J]. Mining Express, 2001, 10: 302-303.

[5] Wenliang Li, Tao Yang, Xiangjun Yu, Qixi Du, Zhibo Li, Chunran Li. Development Status of Large Ball Mills Abroad [J]. Mining Machinery, 2007, 01: 14-16

[6] Jianfeng Chen, Feifeng Xiao. Overview of the Development Direction of Ball mill [J]. China Mining, 2006, 08:97-100.

[7] John Russell. Advanced Grinding Mill Relining for Process Metallurgists and Management. RME, 2006.

[8] Mill Relining Machine Boom Slew Drive Gearbox-Failure due to Incorrect Machine Operation.RME.2004.2.24

[9] Qingchang Tan, Hongzhi Zhao. Mechanical Design [M]. Beijing: Higher Education Press, 2004.7