New Materials, Machinery and Vehicle Engineering
J. Xu and J.P. Davim (Eds.)

doi:10.3233/ATDE230154

Finite Element Analysis of the Main Components of a Gear Rotor Pump

Bin Zheng[a,1], Hongji Jian[a] and Haoyu Qiu[a]
[a] *School of Intelligent Manufacturing, Panzhihua University, Panzhihua, Sichuan*

Abstract. With the rapid development of automobiles, people's demand for high-speed cars is constantly increasing. For a certain rotor pump, this article uses 3D modeling software to establish a solid model of the rotor pump and conduct finite element analysis. According to the original data provided by the manufacturer, the torque of the internal gear and external gear ring of the gear rotor pump during calibration working conditions is calculated to be 12250 N· mm. Then, the finite element simulation was conducted using ANSYS software, including analyzing its structural strength and stiffness, and finite element modal analysis. The simulated contact stress value was 127.63MPa. The natural frequency and modal vibration of the gear rotor pump were obtained. The research results provide a reference basis for the optimization design of subsequent gear rotor pump.

Keywords. Gear rotor pump, 3D modeling, finite element analysis, modal analysis

1. Introduction

The gear pump plays an important role in driving a car. If the gear pump is damaged, the gear pressure will be lost, and the alarm light will light up. In this situation, continuing to drive can pose a fatal hazard to the engine. Therefore, the internal meshing of the rotor in the core components requires high reliability [1]. The working environment of the rotor pump is complex, and the gear is prone to impurities such as iron chips and carbon deposits that can cause significant damage to the gear contact of the rotor pump [2]. However, the oil itself has a high fluidity, and the temperature has little impact on its rotor. Hence, the main focus is on analyzing the contact stress of the rotor pump gear and the stress field of the components. The variation of the stress field during the gear pumping process of a rotor pump has a significant impact on the oil pumping performance [3].

The gear pump is the main working device of the automotive lubrication system, and it is widely used in automotive lubrication systems due to its own advantages [4]. The main characteristics of the rotor pumps are simple structure, compact size, low noise, and stable operating speed. The traditional design method has a long cycle and high cost [5]. But, the application of CAD/CAE technology in the rotor pump design can greatly reduce development costs, shorten development cycles, and improve design quality. The finite element method is one of the effective methods for simulating the contact analysis of gear rotor pump. It can predict the working process of the rotor pump based on the simulation model. At the same time, it can provide the theoretical

[1] Bin Zheng, School of Intelligent Manufacturing, Panzhihua University, Panzhihua, Sichuan, China; E-mail: 22198334@qq.com.

basis for the contact analysis problems through numerical simulation. It also lays the foundation for the contact stress analysis and subsequent optimization design of the rotor pump. It has high practical value and broad application prospects. At the same time, it is also a long-term direction for future research on the gear rotor pump.

2. Three dimensional modeling of the gear rotor pump

The gear pump mainly consists of an internal rotor and an external rotor, as shown in Figure 1. The tooth shape is strictly meshed according to the involute [6]. The tooth number is designed according to the parameters of the rotor pump, and the number of teeth in the internal rotor is Z_1. The number of teeth on the external rotor is $Z_2 = Z_1+1$. When the internal rotor rotates clockwise, the external rotor also rotates simultaneously. During this motion, its contact points are divided into several internal cavities by meshing the internal cavity [7]. When a certain working chamber passes through the oil inlet nozzle, the capacity of the oil chamber will gradually increase to form a vacuum. At the same time, the oil is also pressed into the working chamber through the oil inlet nozzle. While a certain working chamber rotates past the oil outlet, the capacity of the oil chamber gradually decreases, resulting in an increase in oil pressure. In the meantime, the oil is also pressed open through the oil outlet. When the external rotor of the rotor pump rotates for one cycle, each sealing space sequentially sucks and discharges the liquid.

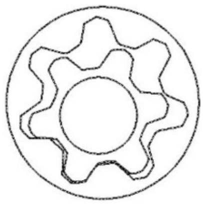


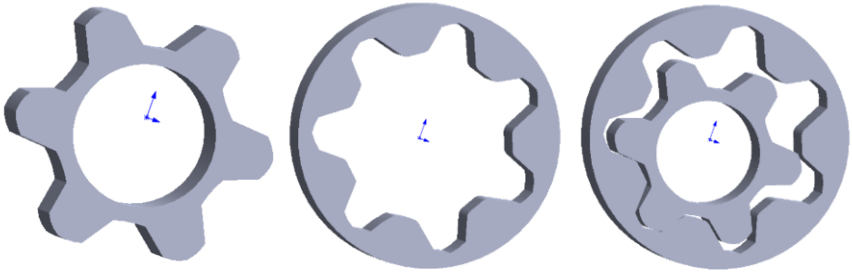
Figure 1. Structural diagram of the gear rotor pump.

The paper takes a certain gear rotor pump provided by an automobile enterprise as the research object. The CATIA software is used to establish the three-dimensional model based on the dimensional parameters of the gear rotor pump. The physical diagram of the main components of the gear pump is shown in Figure 2.

(a) Internal gear. (b) External gear ring. (c) Assembly drawing.

Figure 2. Assembly drawing of the gear rotor pump.

The main components of the gear rotor pump are composed of the internal gear and the external gear ring. Therefore, it is only necessary to establish a three-dimensional model of the correct assembly of the internal gear and external gear ring. When establish the three-dimensional model of the rotor pump, remove the shaft structure that has little impact on the analysis results to simplify the model and improve the calculation speed. Then, the model is saved as the format of .sat for the future analysis convenience. The three-dimensional model is shown as Figure 3.



(a) Internal gear. (b) External gear ring. (c) Assembly drawing.

Figure 3. Three-dimensional model of the gear rotor pump.

3. Statics analysis of the gear rotor pump

The correct movement of the gear rotor pump is that the internal gear moves to drive the external gear ring to rotate in the same direction, but the internal gear speed is higher than the external gear ring, so there is a speed difference that forms a cavity for pumping oil. The internal gear shaft runs around a fixed shaft center. During operation, the internal gear and the external gear ring will mesh with each other, generating mutual forces. It is necessary to conduct force analysis on the rotor pump and research the dynamic and static characteristics of the structure under load.

There are manufacturing errors, installation errors, and gear bearing deformations in the use of the gear rotor pump. In addition, the contact between gear pairs is complex and difficult to verify experimentally. The contact between gear pairs is generally divided into surface contact and point contact based on the initial situation. However, when the gears are actually engaged, both contact modes are converted into surface contact, with the former having a rectangular contact surface and the latter having an elliptical contact surface [8]. The method of using Hertz in-line contact theory is mature, which utilizes the linear contact relationship between two cylinders to represent it.

The gear rotor pump researched in this paper strictly follows involute meshing. Because the width of the contact area of the involute gear is far less than the curvature radius of the tooth surface at the contact point, the tooth surface of the meshing gear can simplify a pair of rollers.

Based on the Hertz equation, the half width of the Hertz can be calculated as Equation (1).

$$b = \sqrt{\frac{4F_n}{\pi L} \times \frac{\dfrac{1-\mu_1^2}{E_1} + \dfrac{1-\mu_2^2}{E_2}}{\dfrac{1}{R_1} + \dfrac{1}{R_2}}} \tag{1}$$

Where, b is the equivalent half width of the contact surface of the gear. F_n is the normal force. L is the equivalent length of the contact surface of the gear. E_1 and E_2 are the elastic modulus. μ_1 and μ_2 are the Poisson's ratio of the gears [9].

Due to the varying pressure on the contact surface in different places, the distribution of the pressure appears as a semi-elliptical cylinder. Meanwhile, the maximum pressure is located at each point of the initial contact line. It is equal to π/4 of the average pressure. If the contact stress is $\sigma_{H\max}$, the combined force on the contact surface is $\pi\sigma_{H\max}\times bL/2$. The surface stress on the contact surface is balanced with the external force, as shown in Equation (2), and the contact stress is as shown in Equation (3).

$$F_n = \frac{\pi\sigma_{H\max}\times bL}{2} \tag{2}$$

$$\sigma_{H\max} = \frac{2F_n}{\pi bL} \tag{3}$$

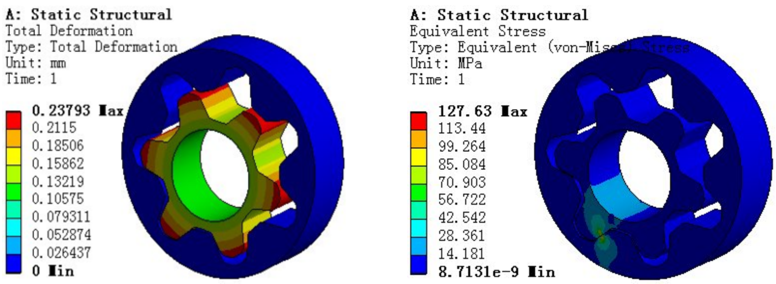
The basic equation for the contact stress is as shown in Equation (4).

$$\sigma_{H\max} = \sqrt{\frac{F_n}{\pi L}\times\frac{\frac{1}{R_1}+\frac{1}{R_2}}{\frac{1-\mu_1^2}{E_1}+\frac{1-\mu_2^2}{E_2}}} \tag{4}$$

The CATIA software is used to establish the three-dimensional model of the gears and simplify them appropriately. Then, the 3D model is imported into ANSYS. In order to carry out finite element analysis, it is necessary to mesh the model for establishing the grid model. The quality of the grid is crucial for handling gear contact simulations. Therefore, the free grid method is used to generate the finite element model of the gear rotor pump, and the grid model is shown in Figure 4.

Figure 4. The grid model of the gear rotor pump.

By adding fixed constraints to the surface of the external gear ring and a torque of 12250N to the internal gear ring, the contact analysis of the gear rotor pump can be carried out. The deformation and the equivalent stress cloud chart are as shown in Figure 5.



(a) Total Deformation cloud chart. (b) The equivalent stress cloud chart.

Figure 5. Results of the finite element analysis.

The contact status is shown as Figure 6. The safety factor is shown as Figure 7.

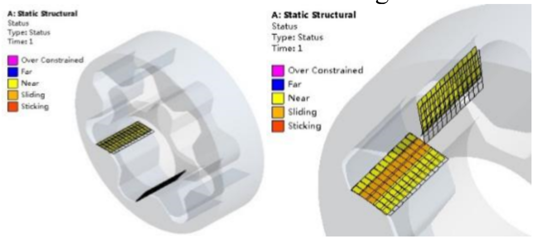


Figure 6. Contact status.

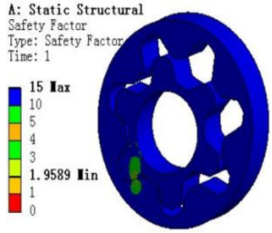


Figure 7. Safety factor.

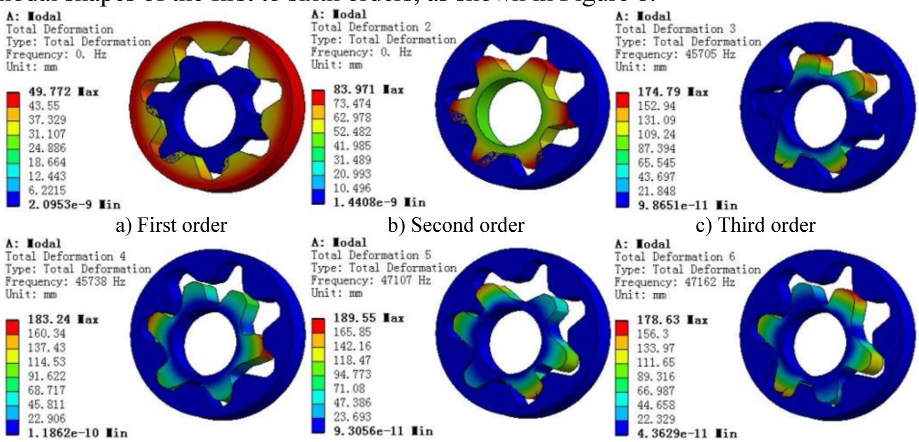
4. Modal analysis of the gear rotor pump

The purpose of the modal analysis is to research the natural frequency, modal shape, amplitude, and other parameters of a structure. The application of the modal technology can help the structure avoid resonance or vibrate at a specified time, and enable engineering technicians to understand how the system responds to different power loads. Of course, other dynamic studies can also solve the determination of control parameters. This paper conducts a constrained modal analysis on the gear rotor pump. Therefore, there is no need to consider any load, only the constraint conditions. The main steps are as follows.

Step 1: Define the interaction relationship. During the modal analysis, it is necessary to release the degree of freedom of rotation around the axis of the internal gear and the external gear ring. At the same time, it is necessary to define the contact attribute in the contact region. Because it is a friction contact, the friction coefficient is set to 0.1.

Step 2: Add a rotating pair. Because the internal gear and the external gear ring can rotate around the axis, it is necessary to add a rotating pair to the internal wall of the internal gear and the external of the external gear ring.

Step 3: Solution. The order of the natural frequency extraction is set to 6. Modal analysis was conducted on the gear rotor pump to obtain the natural frequencies and modal shapes of the first to sixth orders, as shown in Figure 8.



a) First order b) Second order c) Third order

d) Fourth order e) Fifth order f) Sixth order

Figure 8. The first to sixth order modal vibration mode of the gear rotor pump.

From Figure 8, it can be seen that the maximum amplitude of the first order occurs on the external gear ring, while the relative amplitude and deformation of the second, third, fourth, fifth, and sixth order external gear rings are relatively small. The maximum amplitude occurs on the internal gear teeth, and the teeth are deformed. Through the theoretical analysis of the gear rotor pump, the constraint conditions of the rotor pump were determined, and a finite element model of the rotor pump was established. The modal analysis of the gear rotor pump was conducted using the ANSYS Workbench, and the first to sixth order modal shapes were extracted. Through analysis, it can be obtained that under the condition of adding rotating pairs to simulate the motion of the gear rotor pump.

5. Conclusions

This paper uses CATIA software to establish a three-dimensional model of the gear rotor pump and uses ANSYS software to conduct finite element analysis on the established gear rotor pump. Taking the gear rotor pump provided by an automobile enterprise as the research object, perform contact stress analysis on it using finite element analysis software to obtain the equivalent stress and deformation cloud chart. According to the analysis results, it can be seen that the stress concentration point and maximum deformation occur at the position of the rotor pump, providing a basic reference for subsequent optimization. The modal analysis was conducted on the main components of the gear rotor pump. In order to simulate the working conditions, rotating pairs were loaded to obtain the natural frequency and vibration mode diagram under nearly standard working conditions. The research conclusion provides a reference for the structural design of the gear rotor pump.

Acknowledgment

This work was supported by College Students' Innovation and Entrepreneurship Training Program (No. 2023cxcy283).

References

[1] YongChao Y, ZongMei H, LuXiang S. Contact analysis for gears of transplanting mechanism based on ABAQUS finite element method. Hubei Agricultural Sciences; 2021 Jun; 60(12): pp 135-139+171.
[2] Hong J. Contact analysis of spur gear transmission based on ANSYS Workbench. Journal of Heilongjiang University of Technology; 2019 Jun; 19(6): pp 15-19.
[3] ZeRong Y. Finite element analysis of low power racing car gearbox gear based on ANSYS Workbench. Time Agricultural Machinery; 2016 Jun; 43(6): pp 45-47.
[4] HuaXing Z. Contact stress analysis and finite element simulation of involute helical-spur gear transmission. Textile Accessories; 2021 Jun; 48(6): pp 385-389.
[5] YuDa W, HongZhan L, ZhiYong H et al. Calculation and analysis of meshing contact of involute spur gear. Journal of Mechanical Transmission; 2021 Dec; 45(12): pp 41-47.
[6] ChunFeng L. Application of ANSYS software in contact analysis of gears. Mechanical Research & Application; 2021 Apr; 34 (4): pp 139-141.
[7] HengShan L, HuiMin D, DeLun Wang et al. Spur gear set FECM and transmission accuracy analysis with deviation. Modular Machine Tool & Automatic Manufacturing Technique; 2016 Apr; (4): pp 5-8.
[8] JingFang J, YiLing L. Contact strength analysis of gear based on ANSYS. Automobile Applied Technology; 2018 Apr; (8): pp 36-38.
[9] Hong W, Yun W, JiangRu P. Contact load analysis of tractor bearing and gear based on ANSYS. Journal of Agricultural Mechanization Research; 2019 Sep; (9): pp 240-244.