Study on failure of transmission shaft based on finite element and fracture analysis technology

Ming Xia, Ke-lun Zhao*, Hai-xin Zhao and Jing Yang
Guangzhou GRG Metrology &Test co., Ltd., Guangzhou 510627, China
E-mail: zhaokl@grgtest.com

Abstract. Due to the harsh driving environment, the power transmission components of military vehicles are subjected to long-term alternating loads and sudden impact loads. A failure analysis of the transmission shaft of a military truck was carried out. By constructing a mechanical simulation analysis model of the transmission shaft, the dynamic response of the structure under alternating stress and impact load is solved. Then the cause and mechanism of drive shaft failure were analyzed. Using fracture analysis technology to discuss the phenomenon of bearing sleeve cracking, circlip rupture and flange fork deformation. It indicates that caused by overload, foreign matter between the shaft and the needle etc., which ultimately leads to the failure of the universal joint transmission. Finally, the optimal design of the drive shaft is proposed.

1. Preface
Due to the harsh driving environment, the power transmission components of military vehicles are subjected to long-term alternating loads and sudden impact loads [1]. Therefore, the transmission shaft is easily damaged. Performing a failure analysis on it is helpful to gain a deeper understanding of the root cause of its failure and reposition the design ideas. By constructing a mechanical simulation analysis model of the transmission shaft, the dynamic response of the structure under alternating stress and impact load is solved. Then analyze the causes and mechanism of the drive shaft failure. Further use fracture analysis techniques, such as fracture analysis, metallographic analysis, etc., to study its elastic deformation, plastic deformation, rupture or fracture. And consider the influence of time, temperature and other factors, and finally analyze its failure mode. The basic idea is shown in Figure 1.
Figure 1. Flow chart of failure determination method based on simulation and fracture technology

Figure 2. Contact model of elastic bodies

2. Basic Theory

2.1 The Principle of Finite Element Simulation

With the development of computer technology, the engineering application of finite element simulation technology is more and more widely. Compared with traditional methods, finite element simulation technology has the advantages of short period and high efficiency, and can give detailed and complete data.

The transmission shaft is an assembly part, which belongs to a nonlinear contact problem. When solving the contact problem, in addition to satisfying the general equations of elasticity, it is also necessary to satisfy the displacement non-embedding condition in the normal direction of the contact surface and the Coulomb friction law in the tangential direction. There are two elastomers A and B in contact with each other, as shown in Figure 2. Its equilibrium finite element equation in the global coordinate system is:

\[ K_A u_A = P_A + R_A \]

\[ K_B u_B = P_B + R_B \]

Where: \( K_A \) and \( K_B \) are the overall stiffness matrix of elastic bodies A and B; \( u_A \) and \( u_B \) are the node displacement vector of elastic bodies A and B; \( P_A \) and \( P_B \) are the external load vector of elastic bodies A and B; \( R_A \) and \( R_B \) are the contact force vector of elastic bodies A and B.

When the above method is used to solve the contact of the finite element, it is necessary to deal with the definite conditions of the contact \(^{[1]} \) as shown in Table 1.

| Contact State       | Continuous State | Sliding State    | Detached State |
|---------------------|------------------|-----------------|---------------|
| definite conditions | \( R_{ij} = -R_{ij} \) | \( R_{ij} = -R_{ij} \) | \( R_{ij}^2 = R_{ij}^1 = 0 \) |
| \( j = n, t \)      | \( u_{in}^2 = u_{in}^1 + \delta_{in} \) | \( u_{in}^2 = u_{in}^1 + \delta_{in} \) | \( R_{in} = \pm \mu | R_{in} | \) |

In Table 1, \( \delta_{in} \) is the initial gap of the contact point \( i \) in the normal direction; \( \delta_{it} \) is the initial gap of the contact point \( i \) in the tangential direction; \( R_{in} \) is the contact force of the contact point \( i \) in the normal direction; \( R_{it} \) is the contact force of the contact point \( i \) in the tangential direction; \( \mu \) is the friction coefficient of the contact surface; \( u_{in} \) is the displacement of the contact point \( i \) in the normal direction; \( u_{it} \) is the displacement of the contact point \( i \) in the tangential direction.
2.2 Principles of Crack and Fracture Analysis
Different materials and stress states, different action time and temperature, and fractures generated under different environmental conditions will cause specific differences in fracture morphology. In order to carry out accident analysis and take preventive measures, it is very important to analyze various fracture morphology. The research content includes the internal relationship of various factors and reveals the metal fracture mechanism. Therefore, in recent years, fracture analysis technology and analytical instruments have developed rapidly, and a large number of theoretical and practical results have been obtained in fracture analysis. Fracture analysis is divided into macro analysis and micro analysis [4]. Macro analysis refers to observing the fracture with the naked eye, a magnifying glass or a low-magnification optical microscope. The fracture analysis of optical microscopes is collectively referred to as microscopic analysis.

3. Fault Description
The object of this research is the transmission shaft of a military vehicle. During the environmental adaptability test, the universal joint of the hydraulic pump end was damaged or broken, and the hydraulic pump lost its power and stopped working.

Perform visual inspection on the failed parts, as shown in Figure 3 and Figure 4. It can be seen from the figure that a sleeve at the end of a universal joint has a crack at the bottom and protrudes outward, which is macroscopically cracked. It can be preliminarily determined that the structure has been subjected to overload conditions. According to the "slubby" indentation and deformation on the surface of the circlip, it can be preliminarily judged that the failure is caused by the squeeze of the circlip by the broken sleeve. According to the remaining needle squeezing marks on the inner wall of the sleeve and the surface of the cross shaft, it can be preliminarily determined that the structure has been stuck.

Only from the preliminary judgment of appearance, the cause of failure cannot be accurately determined. Based on the finite element analysis method, design problems can be discovered in advance, to determine the strength failure or fatigue failure trend and the weak point of the structure. Based on the fracture failure analysis method, the failure mechanism and influencing factors can be determined more clearly.
4. Finite Element Analysis of Transmission Shaft

4.1 Establishment and Solution of Finite Element Model

The outline drawing of the transmission shaft is shown in Figure 5. The mesh model diagram is shown in Figure 6. See Table 2 for materials.

Table 2. Material parameter list

| Name                          | material | Young's Modulus (GPa) | Poisson's Ratio | Material Density (g/cm³) | Yield Strength (MPa) |
|-------------------------------|----------|-----------------------|----------------|--------------------------|---------------------|
| Flange fork, Spline Shaft     | 45 #     | 210                   | 0.3            | 7.85                     | 355                 |
| Sliding sleeve, Universal joint fork | 40 #     | 211                   | 0.3            | 7.85                     | 335                 |
| Universal joint               | 20Cr     | 207                   | 0.26           | 7.9                      | 540                 |
| Circlip                       | 65Mn     | 211                   | 0.29           | 7.85                     | 430                 |

4.2 Simulation Analysis Results and Judgment

The simulation results are shown in Figure 7-8. The maximum stress is located at the transition between the universal joint and its end bearing, and the maximum stress is 169MPa. The safety factor is 540/169=3.20. The maximum stress of the universal joint fork is located in the area connected to the universal joint, and the value is 130MPa. The safety factor is 335/130=2.58. The maximum stress at the flange fork is located at the connection with the universal joint (as shown in A frame in Figure 7), the value is 120MPa. Safety factor is 2.96. There is a stress concentration area at the flange fork at the root (as shown in frame B in Figure 7).

From the perspective of structural safety factor, the possibility of failure of the flange fork and the universal shaft fork is greater than that of the universal joint. The most likely failure area of the graph edge fork is the two stress concentration areas shown in the figure, either overload failure or fatigue failure. Comparing the location of the fracture failure of the sample (Figure 2), it can be seen that the location of the weak point of failure predicted by the simulation analysis is basically consistent with the actual location of the sample.

5. Analysis of Cracks and Fractures

Based on the results of simulation analysis, through fracture analysis technology, the fracture morphology and metallographic composition of the structure are determined in detail to further determine the specific
cause of structural failure.

5.1 Fracture Analysis
Perform a fracture analysis on the fracture position of the circlip at the I position, as shown in Figure 9. It can be seen from the figure that the source zone, the edge of the end zone and the extension zone all show the shape of shear toughness, indicating the characteristics of overload fracture.

The fracture of the bearing sleeve at position I is analyzed and the results are shown in Figure 10. It can be seen from the figure that the morphology of some fracture area is "along grain" → "river pattern" → "along grain" distributed from the inner wall to the outer wall, some fracture area can be seen as "rock sugar" along grain morphology in the direction of wall thickness. It can be seen that it is mainly a brittle fracture mode.

Analyze the fracture of the bearing sleeve at the E position, as shown in Figure 11. The fracture appearance is intercrystalline fracture from the inner to outer walls of the bottom, the mixed morphology of the main dimples with a little amount of "river patterns" in the core. So the fracture is a mixed fracture mode of toughness and brittleness.

SEM analysis was performed on the surface of the cross axis at the J position, as shown in Figure 12. It can be seen from the figure that there are obvious wear marks and scratches on the surface of the cross shaft.

5.2 Metallographic Analysis
Metallographic analysis of the circlip at the I position is shown in Figure 13. It can be seen from the figure that a small amount of pore defects can be seen in the uncorroded metallography, and the tempered martensite structure can be seen in the corroded metallography.

Metallographic analysis of the bearing sleeve at position E is shown in Figure 14. It can be seen from the figure that the surface material of the part at position E has higher brittleness and lower toughness, because of overheated, which will cause fracture easily under impact load.

Metallographic analysis of the bearing sleeve at position I is shown in Figure 15. It can be seen from the figure that the core is needle-like tempered martensite + a small amount of ferrite structure, and the surface is needle-like martensite structure.

Metallographic analysis of the flange fork at position J is shown in Figure 16. It can be seen from the figure that the metallographic structure is ferrite + pearlite, indicating no obvious structural defects.

Figure 9. The fracture morphology of the circlip at position I
Figure 10. The fracture morphology of bearing sleeve at I position I
6. Conclusion

Through visual inspection, it is predicted that the failure of the flange fork and the universal joint fork will cause a change in the contact mode between the needle bearing and the universal joint. As a result, the needle roller bearings are subject to stress concentration and failure.

The finite element simulation analysis results show that the static strength safety factor of the transmission shaft flange fork and the universal joint fork are both greater than 2.0, which meets the structural strength design requirements of general mechanical products. From the perspective of structural safety factor, the possibility of the failure of the flange fork and the universal shaft fork structure is greater than that of the universal shaft joint. At the same time, it points out the most likely failure area of the flange fork which is the location of the weak point of the structure. The comparison with the failed sample shows
that the weak point of failure predicted by the simulation analysis is basically consistent with the actual failure location of the sample. The above shows the feasibility of this simulation method.

Then the cause and mechanism of drive shaft failure were analyzed by using fracture analysis technology. Phenomena leading to failure are discussed, such as bearing sleeve cracking, circlip rupture and flange fork deformation. It indicates that the reason is overload, foreign matter between the shaft and the needle etc., which ultimately leads to the failure of the universal joint transmission. Finally, the optimal design of the drive shaft is proposed.

Design optimization suggestions: Increasing the root fillet of the universal joint can reduce the stress concentration to improve the strength and fatigue life of the structure. The flange fork root area can be thickened and strengthened to improve its resistance to deformation.

References
[1] Huijuan F 2017 Military vehicle equipment integrated support engineering National (Beijing: Defense Industry Press) 25-40
[2] Huan C, Liangliang C, Chunjie Q and et al 2012 Helical gear contact analysis of high power marine gearbox. Journal of mechanical & electrical engineering 12(12) 1427-30
[3] Qian L, Jinrong W, Zhibin L and et al 2011 Finite element analysis of the contact stiffness for mechanical crowning compensation. Journal of Hydraulic Equipment and Manufacturing Technology. 5(5) 30-3
[4] Henry G and Horstmann D 1990 Macrofractography and microfractography. (Beijing:China Machine Press) 61-70