M. Chen et al. (Eds.)

doi:10.3233/ATDE220483

Simulation Analysis of the Effect of Vibration on Contact Pressure of Electrical Connectors

Yutong LIU [a] and Zhigang KONG [a,1]
[a] *Research Laboratory of Reliability of Electrical Connection and Connector, Beijing University of Posts and Telecommunications, Haidian District, China*

Abstract. As an important component in electronic equipment, the reliability of electrical connectors determines the life of the entire electronic equipment. Vibration is one of the main factors affecting the reliability of electrical connectors, and the contact pressure of connectors is an important indicator of the reliability of connectors. In this paper, ANSYS simulation is used to study the contact pressure of electrical connectors under static and different vibration conditions. The research results show that the contact pressure becomes smaller under the vibration condition than under the static condition. The increase of the vibration amplitude has little effect on the contact pressure; the increase of the vibration frequency has a great influence on the contact pressure. With the increase of the vibration frequency, the minimum value of the contact pressure decreases.

Keywords. Electrical connector, vibration, FE simulation, contact pressure

1. Introduction

An electrical connector is a device that connects two active devices and is used to transmit current or signals. It is an indispensable part of electronic equipment and is widely used in automotive, aerospace, communications, and other fields. The reliability of the electrical connector directly affects the normal operation of the electronic equipment. If the reliability of the electrical connector is reduced, it will lead to short-circuit, open-circuit, and other faults of the electronic equipment circuit [1].

It is unavoidable that electrical connectors are subjected to vibrations in multiple frequency bands during transportation and use, so it is necessary to ensure the reliability of electrical connectors in a vibration environment [2]. Scholars at home and abroad use finite element software to simulate and analyze the electrical connector in the vibration environment and explore the factors affecting the reliability of the electrical connector in the vibration environment. Yu Dazhao et al. used the combination of COMSOL and ANSYS to simulate and analyze the contact parts and obtained the influence of vibration load on the contact resistance [3]. Liu Shaojie used finite element software to explore the electromechanical characteristics of electrical connectors in a random vibration environment [4]. Pan Jun et al. used ABAQUS software to simulate the vibration of the electrical connector, and gave the relationship

[1] Zhigang Kong, Corresponding author, Beijing University of Posts and Telecommunications, Haidian District, China; E-mail: zgkong@bupt.edu.cn

between the life of the contact and the vibration level, to realize the rapid evaluation of the reliability of the electrical connector contact in the vibration environment [5]. Zhu Chongyang et al. combined finite element simulation with failure physical equations to analyze the reliability and optimization of electrical connectors [6]. Pang Hongliang used the finite element optimization module to improve the design of the connector structure, which improved the plugging life of the electrical connector [7].

The contact pressure of the connector is an important indicator of the reliability of the connector. The analysis of the contact pressure of the connector is helpful to find the changing trend of the reliability of the electrical connector under different loads. Therefore, this paper will explore the change of contact pressure of an automotive electrical connector under different vibration conditions.

2. Modeling of Electrical Connectors

A certain type of automotive electrical connector is a reed type. Solidworks is used to build a 3D model of the electrical connector, as shown in figure 1 and figure 2. The contact part between the female end and the male end of the electrical connector is an interference fit. In actual work, the male end pin is inserted into the female end, and the female end reed is elastically deformed by the extrusion of the male end pin, and then the female end reed Clamping is accomplished by reacting to the male pin. The male end and the female end of the electrical connector form an assembly as shown in figure 3.

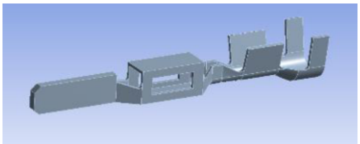

Figure 1. Electrical connector male.

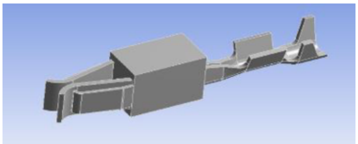

Figure 2. Electrical connector female.

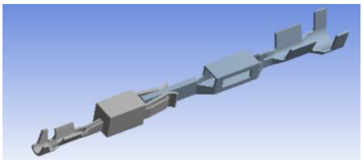

Figure 3. Assembly.

3. Static Analysis of Electrical Connectors

3.1. Theoretical Calculation

Under static conditions, the reed at the female end of the electrical connector is simplified into a cantilever beam mechanical model, as shown in figure 4.

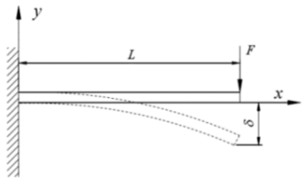

Figure 4. Cantilever beam structural mechanics model.

The relationship between the parameters in this model is as follows [8]

$$F=\frac{3EI\delta}{L^3} \tag{1}$$

$$I=\frac{bh^3}{12} \tag{2}$$

In formula (1): *F* is the contact pressure on the male end after the elastic deformation of the female end reed, the unit is N; *E* is the elastic modulus of the reed, the unit is MPa; *I* is the moment of inertia of the cross-section of the cantilever beam, the unit is mm^4; δ is the maximum deflection (the maximum deformation of the end of the reed), the unit is mm; *L* is the length of the reed, the unit is mm. (2) where: *b* is the width of the reed, *h* is the thickness of the reed, both in mm.

The reeds of the female end of the electrical connector are symmetrical on both sides, and two reeds are superimposed on each side, so two reeds on one side are used for theoretical calculation. The data of the larger reed are as follows: E=110000MPa; δ=0.2mm; L=3.7mm; b=2.8mm; h=0.32mm. The data for the smaller reed is as follows: E=110000MPa; δ=0.2mm; L=2.6mm; b=2mm; h=0.2mm. Bringing the data of the two reeds into (1) and (2) respectively, the sum of *F* calculated from the two formulas is the theoretical value of the contact pressure on the male end, which is 14.972N.

3.2. FE Simulation Analysis

3.2.1. Solve Settings

Import the assembly model into ANSYS for statics simulation analysis. The electrical connector material is $CuSn_4$, and table 1 shows the mechanical properties of the material [9].

Table 1. Mechanical properties of electrical connector materials.

Material Name	Density (kg/m^3)	Young's Modulus (GPa)	Poisson's Ratio	Yield Strength (MPa)	Tangent Modulus (MPa)
$CuSn_4$	8860	110	0.33	345	410

The contact part of the electrical connector can slide relatively and there is friction, so the contact part is set to frictional contact, the friction coefficient is 0.1, and the contact algorithm is set to the augmented Lagrangian method [10]. Tetrahedral mesh is used for mesh division, and the mesh size is 0.2mm. After adding constraints, the settings before solving are completed.

3.2.2. FE Simulation Results

Because the contact part of the electrical connector has a symmetrical structure, it only needs to solve the contact pressure on one side. Insert a reaction force probe into one of the contact surfaces, and obtain the value of the static contact pressure of 15.888N. The direction of the contact pressure is shown by the arrow in figure 5. Compared with the theoretical calculation value of the contact pressure, the error of the FE simulation result is about 6.12%, and it can be considered that the theoretical calculation result is consistent with the FE simulation result.

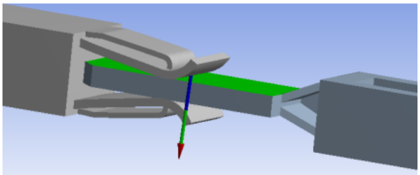

Figure 5. Contact pressure direction.

4. Electrical Connector Vibration Analysis

4.1. FE Vibration Simulation Solution

The solution set for the vibration analysis of this electrical connector is the same as that of the static analysis. A vibration load is applied to the connector in the form of sinusoidal acceleration. The formula for the vibration load is as follows

$$a=A \cdot \sin(2\pi f+\theta)$$

Among them, a is the acceleration, the unit is m/s^2; A is the acceleration amplitude, the unit is m/s^2; f is the frequency, and the unit is HZ. Since the problem of initial acceleration is not involved, it is taken as $\theta=0$. The coordinate system of this electrical connector model in the finite element software is shown in figure 6. The vibration load will be applied in three different directions of X, Y, and Z respectively. This paper will explore the change of the minimum contact pressure under different vibration conditions.

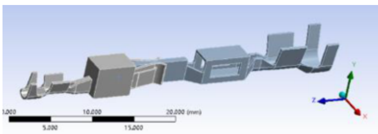

Figure 6. Electrical connector coordinate system.

The FE vibration simulation scheme is divided into two groups, which are the same frequency with different acceleration amplitudes and the same acceleration amplitude with different frequencies. According to GJB1217-2009, the amplitude and frequency conditions of the FE vibration simulation test are selected, as shown in table 2 and table 3, where $g=9.8m/s^2$.

Table 2. The same frequency with different acceleration amplitudes.

Load Application Direction	Load Frequency (HZ)	Acceleration Amplitude(m/s^2)
X	200	5g
X	200	10g
X	200	15g
X	200	20g
Y	200	5g
Y	200	10g
Y	200	15g
Y	200	20g
Z	200	5g
Z	200	10g
Z	200	15g
Z	200	20g

Table 3. The Same acceleration amplitude but different frequencies.

Load Application Direction	Acceleration Amplitude(m/s^2)	Load Frequency (HZ)
X	15g	50
X	15g	200
X	15g	350
X	15g	500
Y	15g	50
Y	15g	200
Y	15g	350
Y	15g	500
Z	15g	50
Z	15g	200
Z	15g	350
Z	15g	500

4.2. FE Vibration Simulation Results and Analysis

Under the conditions of the same frequency and different acceleration amplitudes, the change curve of the minimum contact pressure is shown in figure 7.

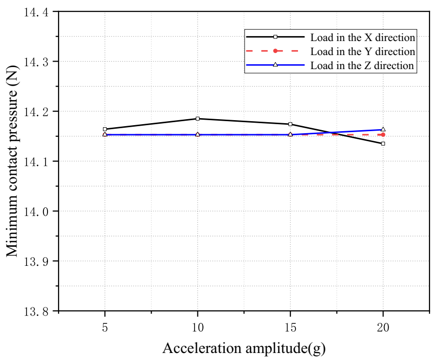

Figure 7. Minimum contact pressure under different acceleration amplitudes.

It can be seen from the FE simulation results that when the load is applied in the X direction, the minimum contact pressure decreases with the increase of the acceleration amplitude; When the load is applied in the Y and Z directions, the minimum contact pressure does not change significantly with the increase of the amplitude, and it remains around 14.13N. Compared with the static contact pressure, the maximum reduction of the contact pressure under this group of vibration conditions is 1.743N, and the reduction is 10.9% of the static contact pressure.

Under this set of vibration conditions, only when the load is applied in the X direction, the minimum contact pressure force decreases with the increase of the acceleration amplitude. This is because the increase of the vibration amplitude in the X direction will lead to an increase in the deformation amount of one side reed and a decrease in the other side reed deformation amount, and the decrease in the reed deformation amount makes the contact pressure on this side smaller.

Under the same acceleration amplitude and different frequency conditions, the change curve of the minimum contact pressure is shown in figure 8.

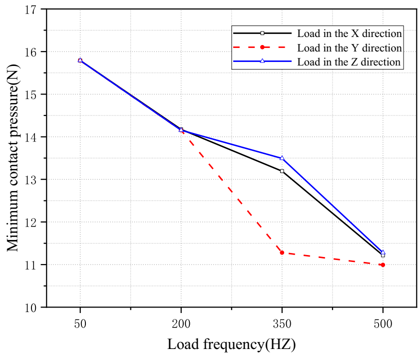

Figure 8. Minimum contact pressure at different frequencies.

It can be seen from the FE simulation results that when loads are applied in the three directions of X, Y and Z, the minimum contact pressure decreases with the increase of frequency. Compared with the static contact pressure, the maximum reduction of the contact pressure under this group of vibration conditions is 4.895N, which is 30.8% of the static contact pressure.

Under this set of vibration conditions, the drop of the contact pressure is significantly higher than that of the previous set of vibration conditions, which also indicates that the vibration frequency has a more significant effect on the minimum contact pressure than the vibration amplitude to a certain extent. This is because at high frequencies, the pins of the male end of the electrical connector interact more frequently with the reeds of the female end, and the load frequency is close to the natural frequency of the electrical connector.

5. Conclusion

This paper takes a certain type of automotive electrical connector as the research object and analyzes the contact pressure of the electrical connector under static and different vibration conditions through ANSYS simulation. The results show that:

(1) The error between the theoretical calculation value and the simulation value of the contact force under the static state of the electrical connector is 6.12%, which proves the rationality of the simulation.

(2) Under the vibration condition where only the acceleration amplitude is changed, the minimum contact pressure decreases compared with the static condition, but the

decrease is smaller. The minimum contact pressure changes smoothly and is only affected by the amplitude change when the load is applied in the X-direction.

(3) Under the vibration condition where only the frequency is changed, the minimum contact pressure decreases compared with the static state, and the decrease is larger. The minimum contact pressure decreases with the increase of frequency, and the minimum contact pressure is obviously affected by the frequency change.

Acknowledgements

The study is financially supported by Postgraduate Teaching Reform Project of Beijing University of Posts and Telecommunications (No. 2021Y038).

References

[1] Jin T. Design of electrical connector reliability test scheme. Environmental Technology. 2020; 38(04): 78-81.
[2] Ren GT. Basic knowledge of electrical connectors (2). Electromechanical Components. 2004; (02): 42-46+53.
[3] Yu DZ, Zhang T. Analysis of the influence of vibration on avionics connectors. Equipment Environmental Engineering. 2021; 18(06):87-93.
[4] Liu SJ. Study on the loose failure mechanism of electrical connectors under multi-axis random vibration. Xidian University, China. 2021.
[5] Pan J, Zhang W, Zhang LB, He QC, Qian P, Chen WH. Vibration reliability test evaluation of electrical connector contacts. Chinese Journal of Mechanical Engineering. 2021; 57(10): 257-266.
[6] Zhu CY, Feng Q and Sun B. Lifetime prediction for electrical connector under the action of random vibration loading. Proceedings of the IEEE 2012 Prognostics and System Health Management Conference, PHM-2012. 2012; Beijing; p. 1-5
[7] Pang HL. Research on plug-in characteristics of electric vehicle connectors. Beijing University of Posts and Telecommunications, China. 2021.
[8] Liu HW. Mechanics of materials II. 5th edition. Beijing: Higher Education Press. 2011.
[9] Zheng F. Quick reference manual for copper and copper alloys. Chemical Industry Press. 2008.
[10] Lu N. Finite element simulation and its application in electrical connection technology. China Machine Press, China. 2020.