Structural Strength Analysis of High Current Connector for Rail Transit

Ruobing Kong¹ and Zhigang Kong²,*

¹,² Research Laboratory of Reliability of Electrical Connection and Connector, Beijing University of Posts and Telecommunications, Haidian District, China

*Corresponding author email: zgkong@bupt.edu.cn

Abstract. Aiming at a high current connector for rail transit, using ANSYS Workbench software for simulating the established connector model under the condition of maintaining the original assembly relationship. Analyzing the overall structural strength of the connector, and obtaining the actual stress distribution and deformation of the electrical connector with various contact relations. The error between the simulation results and the theoretical results of the bolt is within 10%. The maximum stress of each component is less than the yield strength of the material, and the plastic deformation don’t occur under the normal working condition. The overall structure of the connector meets the structural strength requirements.

Keywords: Electrical connector; Assembly relationship; Virtual thread; Finite element analysis.

1. Introduction

As an important supporting electrical component of high-speed train, rail transit electrical connector completes the function of electrical connection and signal transmission between different circuit systems. Its reliability directly affects the normal operation of vehicles. If the connector fails or has potential safety hazards, the on-off of electrical lines between vehicles will be affected [1]. Therefore, it is particularly important to ensure the reliability of electrical connector structure.

In the past, the simulation research of electrical connector mainly focused on the key component of connector terminal. The research on the overall structural strength of electrical connector was few [2-4]. However, an electrical connector has many components which exist multiple assembly relationships. The interaction between components will also affect the overall structural strength of the electrical connector. Therefore, this paper will analyze the overall structural strength of a high-speed railway electrical connector, and study the stress distribution and deformation of the electrical connector with multiple contact relationships in the actual assembly state.

2. Basic Structure and Finite Element Model of Connector

As shown in Figure 1, the connector is composed of shell, insulating table, terminal and structural accessories. The core element terminal is composed of rigid element male and elastic element female. The fit interference between the two contact surfaces is 0.20 mm. The head of the male is slotted, and two internal spring coils prevent excessive deformation. The material parameters of each component are shown in Table 1.
Due to the complex structure of the connector, the original model needs to be simplified in order to reduce the amount of simulation calculation [5]. In addition, in order not to change the assembly situation of the whole model, the matching relationship between components in the original model is not modified, but the contact interface between components is reasonably treated in the contact setting of the Workbench Static Structural [6]. The simplified model and its key dimensions are shown in Figure 2.

3. Parameter Setting

3.1. Setting of Contact Parameters

For the connector selected in this paper, the contact surface which may have relative slip between components is set as nonlinear Friction contact, the other contact relationships are set as linear type of No Separation or Bound [7], with a total of 56 pairs of contact relationships. Because the components of the electrical connector are not allowed to penetrate each other, all the contact pairs are set to Symmetric contact behavior. And the contact algorithm is set as the Augmented Lagrange, and the Normal Stiffness is set to 0.01.
3.2. Interface Treatment and Contact Geometry Correction

3.2.1. Interface Treatment

ANSYS Workbench provides three interface treatment methods [8]. The characteristics of three methods are shown in Table 2.

**Table 2.** Comparison of Interface Treatment methods.

| Method of Interface Treatment | Characteristic |
|------------------------------|----------------|
| Add Offset, No Ramping       | It can specify the positive and negative distance to allow the offset of contact surface. The positive value indicates that the contact pair approaches or increases the interference; A negative value indicates that the contact pair is away from or reduces the interference. The applied interference will be applied to the contact pair in the first sub step. |
| Add Offset, Ramped Effects   | The method is the same as Add Offset, No Ramping. The difference is that the interference amount applied is applied to the contact pair in several sub steps |
| Adjusted to Touch            | When there is a gap or penetration between the contact surfaces, the gap or penetration is eliminated, so that the contact surface just contacts the state without force. |

According to Table 2, different methods are adopted to set the contact interface between the finite element model components of the electrical connector. The specific settings are as follows:

a) The interfaces between cable clamp and cable gland, cable gland and cap nut, steel balls and connecting sleeve are set as Adjusted to Touch respectively.

b) The interfaces between cable gland and connector socket (connector plug) are set as Add Offset, No Ramping, and the offset equals -1.5 mm, and the virtual thread is set on the contact surface.

c) The interface between male and female terminal is set as Add Offset, No Ramping, and offset equals 0 mm, so as to maintain the original fit interference between the contact pairs.

3.2.2. Contact Geometry Correction

Two methods to simulate the stress of thread connection are available by ANSYS Workbench [9]. One is to establish a real thread model; Another is to simplify thread modeling, the thread area is replaced by the smooth cylindrical surface with corresponding diameter. And the setting of thread contact type can be divided into two methods. The first is to set the thread contact type is Bound; The second is to set the thread contact type is Non Separation, and customize the detailed thread by the virtual thread function. This paper uses the virtual thread method. The specific setting process is as follows:

a) Setting the thread contact area as a smooth cylindrical surface with corresponding diameter, which has been installed in 3.2.1.b);

b) Four pairs of contact surfaces of cable gland and connector plug (connector socket), cap nut and connector plug (connector socket) are set as No Separation contact type;

c) Creating thread starting and ending point, as shown in Figure 3;

d) Defining thread parameters, as shown in Figure 4.

3.3. Mesh Generation and Boundary Condition Setting

The grid division results of electric connector are shown in Figure 5. The grid number is 128012, the Average Element Quality is 0.80 and the Average skewness is 0.27, which meet all the requirements of connector grid division quality.

![Figure 3. Setting of thread starting point and end point.](image)

![Figure 4. Setting of thread parameter.](image)
In order to simulate the actual working condition of the connector, Fixed Support is imposed on the inner surfaces of the four bolt holes of the connector socket. And according to the test requirements, the bolt pretension applied to the threads of connector socket and connector plug are 400 N, the bolt pretension applied to M5 ordinary thread the threads is 3684 N. The application of constraints and loads is shown in Figure 6.

4. Simulation Results and Analysis

4.1. Simulation Results and Analysis of Connector Shell

As shown in figure 7 to figure 10, the stress of the connector shell components are mainly distributed at the area of threaded connection. The maximum stress values of connector socket, connector plug, socket cable gland and plug cable gland are 2.89 MPa, 2.86 MPa, 3.94 MPa, 80.85 MPa respectively, which are far less than the 6061-T6 aluminum alloy’s yield strength of 259 MPa, so the electrical connector shell components meet the structural strength requirements, there is no plastic deformation.
4.2. Simulation Results and Analysis of Connector Terminal

As seen from figure 11, because the inner diameter of the male terminal is larger than the outer diameter of the female terminal in the matching state, the male is squeezed and deformed, and the maximum deformation is 0.18 mm. From figure 12, the stress of male is concentrated near its root, the maximum stress is 123.40 MPa, less than the T2-Y’s yield strength of 300 MPa. Figure 13 shows that the maximum stress of female is 14.45 MPa, which is far less than the yield strength of T2-Y. Therefore, when the maximum fit interference between the male and the female terminal is 0.20 mm, the overall deformation of these belong to the elastic deformation stage of the material, which meets the structural strength design requirements of the electrical connector.

As seen from Figure 14, the maximum stress of the M5 bolt occurs at the connection between the bolt and the plug cable gasket. The value is 273.09 Mpa, which does not exceed the A2-70’s yield strength of 450 MPa. Therefore, the bolt connection meets the structural strength requirements.

5. Comparison between Theoretical and Simulation Results

According to the handbook of *MECHANICAL DESIGN* [10], the calculation formula of stress cross-sectional area of external thread is (1), and the strength check formula of ordinary solid screw is (2).

\[ A_s = \frac{\pi}{4} \left( \frac{d_2 + d_3}{2} \right)^2 \]  
\[ \sigma = \frac{F}{A_s} \]  

Where:  
- \( d_2 \) - The basic dimension of the thread pitch diameter, mm;  
- \( d_3 \) - The basic dimension of the thread minor diameter (\( d_1 \)) minus 1/6 of the original triangle height of the thread (\( H = \frac{\sqrt{3}}{2} P \)), that is: \( d_3 = d_1 - \frac{H}{6} \) mm;  
- \( P \) - The Pitch, mm;  
- \( F \) - The bolt pretension, N.
According to GB/T196-2003 Standard for Basic Dimensions of Common Threads, \( d_1 \) is 4.459 mm, \( d_2 \) is 4.675 mm, \( P \) is 0.5 mm. Through calculating, \( A_1 = 14.2 \text{mm}^2 \). Then according the formula (1) and the formula (2), the maximum stress on the M5 bolt is 259.45 MPa. The simulation results show that the maximum stress of bolt is 273.09 MPa. Therefore, the simulation calculation error of bolt connection is 5.26%.

6. Conclusion
In this paper, taking a certain type of rail transit electrical connector as the research object, the overall structure of the connector is analyzed by ANSYS simulation. The results show that:

a) The stress of contact terminal is concentrated at male terminal’s root, and the maximum stress is 123.40 MPa, which is less than the yield strength of T2-Y material.

b) The maximum stress values of connector socket, connector plug, socket cable gland and plug cable gland are 2.89 MPa, 2.86 MPa, 3.94 MPa and 80.85 MPa respectively, which are far less than the 6061-T6 aluminum alloy’s yield strength of 259 MPa. The connector shell components meet the structural strength requirements, and the plastic deformation do not exist.

c) The error between the simulation value and the theoretical value of the M5 ordinary bolt is less than 10%, and the maximum stress value of other components is less than the yield strength of the material, there is no plastic deformation under the normal working state. The overall structure of the connector meets the structural strength requirements.

Based on the research content of this paper, the structural strength of the connector in the dynamic environment can be simulated. Exploring whether the connector can resist the vibration and impact load in the external environment, and estimating the fatigue life of the connector.

Acknowledgements
The study is financially supported by National Key Research and Development Project of Ministry of Science and Technology (No.2018YFF01011604) and Postgraduate Teaching Reform Project of Beijing University of Posts and Telecommunications (No. 2021Y038).

References
[1] Ding YQ and Kang JC 2016 Structure Simulation Analysis of Vehicle Connector Terminal vol 07 (Changsha: Auto Electric Parts) p 51-53
[2] CHEN ZK and FAN XZ 2017 Structural Strength Analysis of A High Frequency Connector vol 04 (Beijing: Structure & Environment Engineering) p 10-17
[3] ZHAO WT and Li T 2019 Simplified Modeling of Screw Connection Simulation Analysis vol 02 (Xi’an:Heavy Machinery) p 69-72
[4] Bruce A and Samuel C 2013 Electromagnetic Radiation Resulting from PCB/high-density Connector Interfaces vol 55 (IEEE Transactions on Electromagnetic Compatibility) chapter 4 p 614-623
[5] Wang JG and Ren MQ 2020 Structural Simulation and Strength Analysis for Trailer Bogie Frame of High-Speed Train Based on CATIA and ANSYS Workbench vol 13 (Recent Patents on Mechanical Engineering) p 266-279
[6] Prakash A and Singh D 2020 Static Structural Simulation Analysis of Leaf Spring Using Ansys Workbench vol 8 (International Journal of Innovative Technology and Exploring Engineering) chapter 8 p 201-206
[7] Wang T 2015 Influence of Dynamic Characteristics of Connector on Fatigue Life (Beijing: Beijing University of Post Telecommunication)
[8] Lu N 2020 Finite Element Simulation and Its Application In Electrical Connection Technology (Beijing: China Machine Press)
[9] Ravichandran G 2020 Finite Element Analysis of Weld Thermal Cycles Using ANSYS (Florida:CRC Press) p 7-8
[10] PU LG and CHEN GD 2013 Mechanical design - 9th Edition (Beijing: Higher Education Press)