Structural Simulation and Fatigue Life Analysis of the Step Attenuator

Zhigang Kong1,* and Peng Liu2
1Research Laboratory of reliability of electrical connection and connector, Beijing University of Posts and telecommunications, Haidian District, Beijing, China
2Research Laboratory of reliability of electrical connection and connector, Beijing University of Posts and telecommunications, Haidian District, Beijing, China

*Corresponding author

Abstract: In this paper, the internal reed of a type of wideband coaxial step attenuator was studied. The research unit was modeled, and the finite element ANSYS-Workbench simulation software was used for simulation analysis to clarify the stress status of the reed and determine the dangerous position. Based on the theory of structural fatigue damage, the nominal stress method is used to predict the theoretical life of the reed, and N-code fatigue simulation software is used for fatigue simulation. The result shows that the maximum stress of reed connector occurs at the fixed end of the reed, which is the dangerous part. The theoretical calculation value of maximum stress is 318.54MPa, the simulation value of finite element is 395.53MPa. The theoretical prediction of fatigue life is 8.65e6, and the simulation life of N-code is 7.003e6.

Keywords: Simulation; Fatigue; Reliability; Reed.

1. Introduction
With the continuously improvement of microwave frequency and bandwidth, the frequency and level loads of microwave test instruments have increased. In order to protect the expensive test instruments and equipment in related application fields, the wideband coaxial step attenuators have been widely used. However, the reliability of the wideband coaxial step attenuators at present studied is mainly affected by the following four aspects: the design of the attenuator circuit of the attenuation unit, the structural stability and fatigue of the internal reed connector, reliability during connection, that is, the electrical contact reliability between the reed and the attenuation unit, the stability and accuracy of the implementation structure[1][2].

This paper focuses on the structure and reliability of the reed connector in a wideband coaxial stepping attenuator. The stress distribution of the reeds as well as the dangerous position was determined and the fatigue life of the reed was predicted according to theoretical calculation and finite element simulation [3].

2. Model Establishment of the Step Attenuator
According to the setting attenuation decibels, the action of the ejector rod is driven by the electromagnetic relay under the control of the electrical signal, and the corresponding reeds switch from the straight through plate to the attenuation circuit, which are driven by the ejector rod. Then the reed and the attenuation plate coating are in stable contact. The signal is attenuated by the attenuation circuit finally. It means that the role of the reed structure in the attenuator is the key factor that determines its reliability and life based on the working principle. The wideband coaxial step attenuator has multiple
attenuation units, and its structure of the attenuation is completely the same. The first level attenuation unit, and each level attenuation unit consists of a relay, the ejector rod, reed connector, the fixed stage of the reed, attenuation unit (attenuation circuit and attenuation plate coating) and pass-through circuit. The Solidworks software is used for modelling, as shown in Fig.1.

Figure 1. Single attenuation element model.

3. Model Theoretical Calculation and Finite Element Simulation

3.1. Model Mechanics Calculation

As shown in the calculation model of Fig.2, the reed can be simplified into a cantilever beam. The cantilever beam is subjected to two concentrated forces. According to the calculation of material mechanics and theoretical mechanics, the stress distribution, maximum stress, and maximum stress position of the reed are obtained.

In the Fig.3, $f_1$ is the ejector force, and $f_2$ is the reaction force of the attenuation plate coating on the reed. According to the theory of material mechanics [4], the shear diagram and bending moment diagram are obtained respectively. Firstly, the support reaction force and torque at the fixed end of the cantilever beam are obtained. Supposing the reaction force of the fixed end support is $T$ and the torque is $M$. Then shear equation $f(x)$ is solved.

$$\begin{align*}
T &= f_2 - f_1 \\
M &= f_1a - f_2l\\
F_{S_1}(x) &= f_2 - f_1 \quad (0 \leq x \leq a) \\
F_{S_2}(x) &= f_2 \quad (a \leq x \leq l)
\end{align*}$$

(1)

According to the shear force equation (2), the shear force diagram is obtained. Then the bending moment equation is solved.

$$\begin{align*}
M_1(x) &= f_1a - f_2l + (f_2 - f_1)x \quad (0 \leq x \leq a) \\
M_2(x) &= f_2x - f_2l \quad (a \leq x \leq l)
\end{align*}$$

(3)
According to the bending moment equation (3), the bending moment diagram is obtained. Based on the principle of material mechanics, the maximum normal stress of the cantilever beam (the reed) occurs at the maximum bending moment. The maximum bending moment is at \( x = 0 \) or \( x = a \), which is substituted into the bending moment equation (3).

\[
\begin{align*}
M_1(0) &= f_1a - f_2l \\
M_2(a) &= f_2a - f_2l.
\end{align*}
\]

(4)

When the contact between the reed and the plate of the damping plate reaches a stable state, it is assumed that the reed and the plate of the damping plate are completely bonded. The superposition principle is used to solve the force \( f_1 \) on the reed at this time.

When only \( f_1 \) affects, set the deflection \( \omega_b^1 \) at point B and the deflection \( \omega_c^1 \) at point C. The calculation equations for the deflection and rotation angle of the cantilever can be obtained.

\[
\omega_b^1 = \frac{1}{3EI} f_1a^3. 
\]

(5)

\[
\omega_c^1 = \frac{1}{6EI} f_1a^2(3l - a) .
\]

(6)

When only \( f_2 \) affects, the deflection \( \omega_b^2 \) at point B and the deflection \( \omega_c^2 \) at point C can be obtained.

\[
\omega_b^2 = \frac{1}{3EI} f_2a^2(3l - a) .
\]

(7)

\[
\omega_c^2 = \frac{1}{6EI} f_2a^3.
\]

(8)

According to the principle of superposition, the formula (10) is obtained.

\[
\begin{align*}
\frac{1}{3EI} f_1a^3 - \frac{1}{6EI} f_2a^2(3l - a) &= \omega_b \\
\frac{1}{6EI} f_1a^2(3l - a) - \frac{1}{3EI} f_2a^3 &= \omega_c.
\end{align*}
\]

(9)

The force \( f_1 = 0.151N \) is obtained by substituting \( a = 4.49\text{mm}, l = 7.89\text{mm}, \omega_c = 0.55\text{mm}, f_2 = 0.0588N(6g \text{ force}) \) into formula (9).

The bending moments which are \( M_2(a) = -2 \times 10^{-4}N \cdot m \) and \( M_1(0) = 2.14 \times 10^{-4}N \cdot m \) are obtained by substituting \( f_1 = 0.151N, f_2 = 0.0588N, a = 4.49\text{mm}, l = 7.89\text{mm} \) into the above bending moment equation (4). Comparing the absolute size \( |M_2(a)| < |M_1(0)| \), the maximum normal stress occurs at the fixed end of the cantilever beam, that is, the end where the reed is in contact with the fixed base, as shown in fig.2 at point A.

According to the formula (10) for maximum normal stress in material mechanics, the maximum stress \( \sigma_{\text{max}} \) is obtained, which is \( 3.1854 \times 10^8\text{Pa}(318.54\text{MPa}) \).

\[
\sigma_{\text{max}} = \frac{M_{\text{max}}}{W_z}.
\]

(10)

In the formula (10), \( W_z \) is the bending cross-section coefficient, and the \( M_{\text{max}} \) is the maximum bending moment. According to the rectangular cross section of the reed, \( W_z \) is obtained, where \( b = 1.12\text{mm}, h = 0.06\text{mm}, y_{\text{max}} = 0.03\text{mm} \). The \( W_z \) is \( 6.72 \times 10^{-13}\text{m}^3 \).

3.2. Model Finite Element Simulation

In practical engineering, the finite element simulation software is frequently used to perform simulation analysis on the engineering model. ANSYS-workbench is used to perform simulation analysis of the reed in this paper [5]. The simulation analysis model is modeled in SolidWorks as shown in Fig.1. At the same time, the two parts of the through-circuit and the attenuation circuit are suppressed to reduce the amount of calculation and increase the convergence of the simulation.

According to the materials of each component of the wideband coaxial step attenuator, the corresponding materials of the model are assigned, which are shown in the Table 1.
Table 1. Part parameters.

| Part name                | Material | Density (g/cm³) | Young's modulus (MPa) | Poisson's ratio |
|--------------------------|----------|-----------------|-----------------------|-----------------|
| Reed                     | C17200   | 8.25            | 1.28e5                | 0.3             |
| Reed fixing table        | PPE      | 1.08            | 2550                  | 0.35            |
| The ejector rod          | PAI      | 1.44            | 1.55e5                | 0.39            |
| Attenuation sheet coating| Au       | 19.32           | 82500                 | 0.42            |

According to the load and constraint of the wideband coaxial step attenuator, the simulation model was loaded and constrained. The upper surface of the attenuation film coating was fixed support, the surface under the fixed platform was fixed support, and the bottom surface of the ejector rod was loaded with displacement load.

The stress distribution of the reed is shown in Fig.3, with the maximum value of 395.53MPa, which is located at the end of the reed contacting with the reed fixed platform. The stress value can reach more than 360MPa at the action position of the ejector rod on the reed, that is, the dumbbell shaped opening of the reed. In the case of the impact of the ejector rod, the stress concentration occurs in the opening position of the reed. At the same time, due to the impact of the ejector rod, this part is constantly bent up and down during the actual working process and subject to cyclic loading, which is prone to fatigue cracks. This is the reason why the reeds are prone to fatigue fracture at this part.

![Stress cloud diagram of the reed.](image)

4. Reed Fatigue Life Analysis

The reed has the risk of fatigue fracture during long-term service. First, the theoretical fatigue life prediction of the reed is performed, and then the n-code fatigue simulation analysis software is used to perform fatigue analysis and life prediction of the reed.

4.1. Prediction of Reed Fatigue Life

In this paper, the fatigue life of the reed is predicted. First, the S-N curve of component is obtained through revising the S-N curve of the material, and then the fatigue life of the component is estimated. The correction equation is shown in equation (11):

\[ S_a = \frac{\sigma_a}{K_f} \varepsilon \beta C_L. \]  \hspace{1cm} (11)

In the formula, \( \sigma_a \) corresponds to the stress of the S-N curve of the material, \( S_a \) corresponds to the stress of the S-N curve of the structure, \( K_f \) is the fatigue gap coefficient, \( \varepsilon \) is the size coefficient, \( \varepsilon = 1 \), \( \beta \) is the surface state coefficient, \( \beta = 1 \), and \( C_L \) is the loading mode, \( C_L = 1 \). The fatigue gap coefficient can be obtained through Peterson's publicity, that is:

\[ K_f = 1 + \frac{K_T - 1}{\varepsilon^2 + \rho^2}. \] \hspace{1cm} (12)

In formula (12), \( \rho \) is the root radius of the notch of the structural member. According to the opening size of the reed, \( \rho \) is 0.23; \( a \) is the material constant, which is 0.8; \( K_T \) is obtained by the finite element simulation method, which is 1.5. Then \( K_f \) is calculated, which is 1.12. Take the parameter into the equation (11) to find \( S_a = 0.89 \sigma_a \) [6][7].
The solid line curve of the Fig.4 is the S-N curve of the material C17200 beryllium bronze, and the dotted curve is the S-N curve of the reed [8].

![Figure 4. S-N curve of C17200 beryllium bronze and the reed.](image)

According to the above theoretical calculations, the maximum stress of the reed under the action of the ejector rod is 318.54 MPa. Since the reed is continuously switched up and down in practical working conditions between the attenuator and the through circuit. And it is subject to equal proportion of load, the load ratio is $R = -1$. So $S_a = \frac{1}{2}(S_{max} - S_{min}) = 318.54 MPa$.

According to the corrected S-N curve in Fig.4, it is known that the predicted fatigue life is $8.65\times10^6$.

4.2. Reed Fatigue Life Simulation Analysis

N-code fatigue simulation software is engineering integration fatigue design simulation software. It can realize fatigue design, fatigue life prediction, vibration fatigue simulation, solder joint fatigue simulation, etc. The n-code implements simulation result input, solution, result input, and post-processing through a modular solution method. It is easy to operate with complete parameters and high reliability of simulation results. At the same time, n-code has been jointly developed with ANSYS, and users can directly implement professional fatigue simulation under ANSYS.

![Figure 5. Life cloud diagram and hotspot diagram of simulation results.](image)

In this paper, through finite element simulation, the result file is input into N-code, and a fatigue simulation module is established. The fatigue simulation results of the reed are obtained, and the reed fatigue life is $7.00\times10^6$, as shown in Fig.5[9][10].

5. Conclusion

(1) The mechanical calculation model and the finite element simulation model are established. Through mechanical calculation, the maximum stress value is solved, which is 318.54MPa, and the dangerous position is determined,

(2) Through the finite element simulation, the maximum stress position is the same as the mechanical calculation. The maximum value of the simulation result is 395.53MPa. The causes of fatigue fracture of the reed are stress concentration and cyclic loading;
(3) The fatigue life of the reed is theoretically solved by the nominal stress method, which is $8.65 \times 10^6$. And through the n-code fatigue simulation software, the fatigue life of the reed is simulated and solved, and the simulation life value is $7.00 \times 10^6$.

Acknowledgements
The study is financially supported by National Key Research and Development Project of Ministry of Science and Technology (No.2018YFF01011604).

References
[1] Zhao Lihua, Li Yuan. Design and application of program-controlled step attenuator in communication test equipment [J]. Computer knowledge and technology, 2014,10 (10): 2461-2463
[2] Bo Tang. Design and Implementation of 2.4mm Programmable Step Attenuator [D]. Xi'an University of Electronic Science and technology, 2007.
[3] Wenfeng Zheng. Design and Implementation of 2.4mm 11dB Programmable Step Attenuator [C]. Chinese Institute of Electronics. Proceedings of 2015 National Microwave and Millimeter Wave Conference.
[4] Sun Xunfang. Mechanics of Materials, 5th edition. Beijing: Higher Education Press [M], 2009.
[5] Shusheng Fu. ANSYS Workbench 17.0 Numerical Simulation and Explained by Example [M]. Beijing: People's Posts and Telecommunications Press, 2018.
[6] Weixing Yao. Structural fatigue life analysis [M]. Beijing: Science Press, 2019.
[7] Tianhai Chen. Research on Fatigue Life Prediction of Miniature Connectors Based on Nominal Stress Method [J]. Electromechanical Components, 2009, 29 (04): 20-24.
[8] Editorial Committee of China Aviation Materials Manual. China Aviation Materials Handbook, Volume IV. Beijing: China Standard Press.
[9] Congcong Wen, Qiang Yang, Yiming Zhang, Zhili Sun. Simulation calculation model of contact fatigue life of military aviation electrical connector [J]. Mechanical Design and Manufacturing, 2018 (S1): 51-54.
[10] Jin Dan, Wang Wei, Tian Dajiang, Lin Wei. Finite element analysis of fatigue life of notched parts under non proportional load [J]. Journal of mechanical engineering, 2014,50 (12): 25-29.