Reliability Parameter Optimization of Electromagnetic Relay for Aerospace Products Based on 3D Thermal Stress Field

Cheng Mu, Wei Song*, Tianli Hui
Beijing Spacecrafts Ltd, Beijing, China

*Corresponding author: weisong@bjspacecrafts.org

Abstract. The reliability of thermal and solid coupling relay is an important factor in aerospace products. A refined model based on the internal constitutive characteristics of the device is established, and the thermal solid coupling relationship curve is established. The three-dimensional temperature field and thermal stress field are analyzed by using the finite element analysis software ANSYS. Through the simulation of the thermal field and thermal stress field under different parameters, the design factors affecting the internal thermal distribution of the device are preliminarily analyzed, which provides the parameter basis for the reliability optimization design of the relay.

Keywords: Electromagnetic relay; reliability; ANSYS.

1. Introduction
Electromagnetic relay is a kind of basic electrical control components, which can automatically control the circuit output from a long distance. It has the functions of automatic regulation, conversion circuit, signal transmission, executive control, system power distribution, circuit isolation, etc. Electromagnetic relay is widely used in control system, communication system, power system and other automatic electrical systems. Because of its high conversion depth, multi-channel synchronous switching, large input-output ratio and strong anti-interference ability, electromagnetic relay has become an indispensable basic component of industrial control, aerospace, weapon equipment and other fields.

As a kind of control switch element, the reliability of the whole system is affected by the vibration, impact, explosion, noise and other mechanical forces in practical application. Due to the limitation of development time and development strength, the product design of electromagnetic relay industry in China mainly relies on experience and reference to similar foreign products. The manufacturing process mainly depends on manual assembly and debugging. Therefore, the domestic and foreign similar products are relatively low in all aspects, especially in the anti-vibration performance. In this paper, taking an aerospace product electromagnetic relay as an example, the finite element analysis software ANSYS is used to analyze the three-dimensional temperature field and thermal stress field. Through the simulation of thermal field and thermal stress field under different parameters, the design factors affecting the thermal performance of the device are preliminarily analyzed.
2. Introduction to thermal stress analysis

Due to the mismatch of thermal expansion coefficients between different structures or different parts of the same structure, the expansion or contraction degree of each other is not consistent during heating or cooling, which leads to the generation of thermal stress. The process of coupling analysis depends on the coupled physical field, and its analysis methods can be divided into two categories: sequential coupling and direct coupling.

2.1. Thermal stress analysis unit

In the indirect method, the conventional thermal element is used for thermal analysis, and then the thermal element is transformed into the corresponding structural element, and the obtained node temperature is applied to the model as the body load, and then the structural stress analysis is carried out. Therefore, there is the problem of conversion between thermal element and structural element in the whole analysis process.

2.2. Basic steps of thermal stress analysis

For indirect methods, different databases and result files are used. Fig. 1 shows the data flow chart of thermal stress analysis by indirect method. Each database contains appropriate entity model, element, load, etc. One result file can be read into another database. However, the unit and node numbers must be consistent in the database and result file.

![Data flow chart of thermal stress analysis](image1)

The LDREAD command connects different physical environments in coupled field analysis, so that the analysis results in the first physical environment are transferred to the next physical environment as loads.

3. Thermal stress analysis of electromagnetic relay

Taking jgx-0613m relay as an example, the mathematical model and finite element model of jgx-0613m relay are established. The distribution of temperature field and thermal stress are obtained by using ANSYS finite element software.

The internal structure of the relay is shown in Fig. 1. These layered surfaces are filled with gas, and a thin alloy shell is covered on the outer surface of the device. In the inner part of the device, four chips are heating bodies. There is heat conduction between the layer structures and the surface of the device is convective heat dissipation with the environment.

![Internal structure of relay](image2)
3.1. Thermal stress analysis process of relay

3.1.1. Establish the finite element model. In this paper, APDL language is used for parametric modeling. Parametric modeling technology is a set of technology to extract the relevant feature information from the model and provide model generation, information reading and editing. By changing the quantitative information in the model into adjustable parameters, users can assign different values to the variable parameters. After the parametric modeling of electromagnetic relay is completed, the generated model is shown in Figure 3.

![Finite element model](image)

Figure. 3 finite element model

3.1.2. Defining material properties. ANSYS uses three commands to control element types and attributes, among which type controls element type, real controls geometric parameters and mat controls element material properties. The material properties defined in this paper are shown in Table 1.

| Material          | Thermal conductivity (W/m·°C) | Coefficient of thermal expansion (10⁻⁶/°C) | Elastic modulus (MPa) | Poisson's ratio | Thickness (mm) |
|-------------------|-------------------------------|------------------------------------------|------------------------|-----------------|----------------|
| Silicon chip      | 85                            | 2.98                                     | 140                    | 0.26            | 0.5            |
| Pbbsn solder      | 39                            | 29                                       | 10                     | 0.3             | 0.2            |
| Conductor         | 48                            | 5.7                                      | 119                    | 0.3             | 0.1            |
| Ceramic substrate | 15                            | 5.7                                      | 255                    | 0.29            | 0.64           |
| Base              | 20                            | 5.1                                      | 141                    | 0.23            | 1.5            |

3.1.3. Mesh generation. After the establishment of the finite element model, it is necessary to establish the mesh model and control the mesh size. After the relay specifies the attributes, the required mesh size is specified, and then the finite element mesh of the model can be obtained. The results are shown in Fig. 4.

![Grid generation](image)

Figure. 4 grid generation
3.1.4. Apply load to solve the problem

(1) In the three-dimensional model, the thickness of the chip is 0.04cm, which is very thin compared with the whole device. In this paper, the chip is regarded as a uniform heater. The power dissipation of each chip is 0.325w, which is converted into heat generation per unit volume to $3.63 \times 10^7 (W^3)$.

(2) When the convection load is applied on the surface of the shell, the convection coefficient on the horizontal plane is $8.437 (W/m^2\cdot K)$, and the vertical surface is $12.152 (W/m^2\cdot K)$. The ambient temperature of convection is 25 °C and the constant temperature of bottom plate is 32.2 °C.

3.2. Simulation results

3.2.1. ANSYS results. At the end of the solution, ANSYS post processor can be used to view and check the solution results. Isoline and vector diagram can be used to display the results. The cloud pictures of steady-state thermal analysis results of electromagnetic relay are shown in Fig. 5 and Fig. 6.

![Overall temperature nephogram](image1)

**Figure. 5** overall temperature nephogram

![Internal temperature nephograms](image2)

**Figure. 6** internal temperature nephograms

Because the chip is the main part of electromagnetic relay heating, the temperature rise in the process of device operation is mainly due to the role of local heat source chip. Therefore, the highest
temperature region of the model is on the chip. The heat generated by the chip is mainly transmitted through two ways: internal and external channels. First of all, the thermal resistance of the chip itself will be encountered, followed by the conduction thermal resistance of solder layer and conductor. After the heat flow reaches the ceramic substrate, it overcomes the conduction thermal resistance of the substrate and the conduction thermal resistance of the base, so as to reach the outer surface of the base. In addition, part of the heat generated by the heat source passes through the internal space of the model to the inner surface of the shell in the form of convection. However, the main way of heat transfer to the outer surface of the chip is to overcome the external heat transfer resistance. According to the profile of temperature distribution, most of the heat flow is transferred from the chip to the bottom of the model along a certain diffusion angle. The simulation results show that the thermal path from the chip to the bottom of the model is the main way of heat dissipation.

3.2.2. Thermal structural coupling. After the solution of thermal analysis is completed, re-enter the pre-processor to convert the thermal unit to the corresponding structural unit. The switch element type dialog box appears in the ANSYS display window, and select the structural element from the drop-down list box.

![Switch Elem Type](image)

**Figure. 7** conversion unit type dialog box

When the thermal element is converted into structural element, the material properties in structural analysis, such as elastic modulus, linear expansion coefficient and Poisson's ratio, are setted; the displacement boundary conditions are given, and the y-direction degree of freedom of the bottom node of the model is assumed to be zero. The temperature data obtained from the heat conduction analysis are read in as heat load. The results of equivalent stress distribution are shown in Fig. 8 and Fig. 9.

![Distribution of Equivalent Stress](image)

**Figure. 8** distribution of equivalent stress of the whole body
The results of finite element analysis show that the maximum equivalent thermal stress occurs at the four corners of the solder layer. At the interface between the chip and the solder layer, because the thermal expansion coefficient of the chip is 2.85 ppm, the thermal expansion coefficient of the solder layer is 20 ppm, there is a mismatch between the chip and the solder layer. In the heating process, the thermal expansion of the chip is less than that of the solder layer, and for the model, the high temperature region is mainly concentrated in the chip and its vicinity, which makes the contact surface between the chip and solder layer the largest thermal stress area in this model.

4. Conclusions
In the increasingly fierce competition environment, the market demands to shorten the production cycle of products. One of the key parts is the shortening of product design cycle. The finite element analysis software ANSYS is used to analyze the three-dimensional temperature field and thermal stress field. Through the simulation of thermal field and thermal stress field under different parameters, the design factors affecting the thermal performance of the device are preliminarily analysed. After numerical simulation analysis, it is found that the maximum equivalent thermal stress occurs at the four corners of the solder layer. If the thermal stress is too large, it will lead to spalling, fatigue damage, mechanical fracture and permanent deformation of the device. Therefore, the internal thermal stress should be reduced to improve the reliability of the device.

References
[1] Huimin Liang, Huimin Ren, Xuerong Ye, et al. Research on Reliability Tolerance Analysis Technology of aerospace electromagnetic relay [J]. Chinese Journal of Aeronautics, 2005, 18 (1): 65-71
[2] Jie Deng. Robust design of aerospace electromagnetic relay [D]. Harbin Institute of Technology, 2010
[3] Yingji Yin. Research on dynamic characteristic test and reliability evaluation method of aerospace electromagnetic relay [D]. Harbin Institute of Technology, 2007
[4] Electromagnetic relay design, Harbin Institute of Technology, 2014.