Simulation of Heat Dissipation in a Closed Enclosure Based On Flotherm

Weibo Que¹, Qiang Zhang¹, Dehuai Zeng¹* and Chenxue Wang²,a

¹Laboratory of Optical and Electrical Engineering Technology, Shenzhen University, Guangdong 518060, PR China
²Shenzhen Key Laboratory of Advanced Manufacturing Technology for Mold &Die, Shenzhen University, Shenzhen 518060, Guangdong, PR China

*Corresponding author e-mail: 50042500@qq.com, awangchenxue2016@email.szu.edu.cn

Abstract. As an important method to measure the reliability of electronic products, thermal simulation is widely used in product life prediction and reliability optimization. In this paper, the relationship between the working temperature and working performance of power components and the importance of thermal simulation analysis are discussed. The basic theory of thermal analysis software Flotherm is expounded. The functional characteristics and application range of Flotherm software are introduced. The chassis power device is designed. The boundary condition setting, grid setting, and result processing of the simulation model are introduced in detail. The results illustrate that the maximum temperature of the power components can be guaranteed within the specified temperature range, which satisfies the temperature requirements for reliable operation.

1. Introduce

As electronic devices flourish toward miniaturization, high power consumption, and high density, the contradiction between heat flux density and heat dissipation performance of electronic chips has become increasingly prominent. On the one hand, most of the input power of electronic equipment becomes thermal power consumption, and the working performance of electronic products is closely related to the temperature at work. When the thermal power generated by the input power is converted, the thermal energy is short-lived. When it is difficult to dissipate, it will increase the temperature of the internal power components of the electronic product. Since the power components have certain limit operating temperature limits, when the temperature limit is exceeded, the working state may be unstable or even damaged. Therefore, in order to ensure the normal operation of the power components, the electronic device must meet certain structural design requirements. Thermal performance requirements. The traditional method is to process and test the sample, and then continuously improve the product according to the test feedback. In turn, the product development cycle and cost increase. If the thermal simulation analysis is integrated into the product design stage, the simulation software is used to analyze the temperature field distribution of the product model in real time, and the model is refined or optimized for the heat dissipation structure, and the maximum temperature in the model is reduced. The problem of heat accumulation and heat dissipation keeps the...
maximum temperature in the model within a reasonable range. In order to shorten the product development cycle and improve the success rate of the product, it has laid a good foundation [1-2].

2. Type of heat transfer
Heat transfer mainly has three basic ways of heat conduction, heat convection and heat radiation [3]. The heat conduction follows Fourier's law: the heat passing through a given area per unit time in heat conduction is proportional to the cross-sectional area perpendicular to the heat conduction direction and its temperature change rate. The specific formula is as follows [4]:

\[ \Phi = -\lambda A \frac{\partial t}{\partial x} \]  

Where: a negative sign indicates that the direction of heat transfer is opposite to the direction in which the temperature rises; \( \Phi \) is the heat flux in W; \( \lambda \) is the thermal conductivity in W/(m·°C); \( A \) is perpendicular to the direction of heat flow Heat transfer area in m²; \( \partial t/\partial x \) is the rate of temperature change in the direction of heat flow in °C/m.

Thermal convection follows Newton's law of cooling [4]: When there is a temperature difference between the surface of the object and the surrounding environment, the amount of heat lost per unit time is proportional to the temperature difference and the surface area of the object's heat dissipation. The proportional coefficient is the heat transfer coefficient. Its calculation formula is as follows:

\[ \Phi = h_cA\Delta t \]  

Where: \( h_c \) is the surface heat transfer coefficient, the unit is W/(m·°C); \( A \) is the convective heat transfer surface area of the object, the unit is m²; \( \Delta t \) is the temperature difference between the surface temperature of the object and the temperature of the fluid, the unit is °C.

3. Theory foundation of thermal simulation
The basic calculation theory of thermal analysis software FLOtherm includes energy conservation equation, momentum conservation equation and mass conservation equation. The general form of the governing equation for fluid flow is as follows:

\[ \frac{\partial \rho \Phi}{\partial t} + \nabla (\rho \mathbf{V} \Phi) = \nabla (\Gamma \nabla \Phi) + S \]  

Each of the equations is an unsteady term, a convection term, a diffusion term, and a source term. Apply the mass conservation equation directly to the Cartesian coordinate system as follows:

\[ \frac{\partial \rho \Phi}{\partial t} + \frac{\partial \rho u \Phi}{\partial x} + \frac{\partial \rho v \Phi}{\partial y} + \frac{\partial \rho w \Phi}{\partial z} = 0 \]  

The momentum conservation equations in each direction in a Cartesian coordinate system are as follows [4]:

X direction:

\[ \frac{\partial (\rho u)}{\partial t} + \frac{\partial (\rho u u)}{\partial x} + \frac{\partial (\rho u v)}{\partial y} + \frac{\partial (\rho u w)}{\partial z} = -\frac{\partial P}{\partial x} + \frac{\partial}{\partial x} \left( \mu \frac{\partial u}{\partial x} \right) + \frac{\partial}{\partial y} \left( \mu \frac{\partial u}{\partial y} \right) + \frac{\partial}{\partial z} \left( \mu \frac{\partial u}{\partial z} \right) + S_u \]  

Y direction:
The performance on the energy conservation equation is as follows [4]:

$$\frac{\partial (\rho u)}{\partial t} + \frac{\partial (\rho u u)}{\partial x} + \frac{\partial (\rho u v)}{\partial y} + \frac{\partial (\rho u w)}{\partial z} = -\frac{\partial p}{\partial x} + \frac{\partial}{\partial x} \left( \mu \frac{\partial u}{\partial x} \right) + \frac{\partial}{\partial y} \left( \mu \frac{\partial u}{\partial y} \right) + \frac{\partial}{\partial z} \left( \mu \frac{\partial u}{\partial z} \right) + S_u \tag{6}$$

$$\frac{\partial (\rho v)}{\partial t} + \frac{\partial (\rho uv)}{\partial x} + \frac{\partial (\rho vv)}{\partial y} + \frac{\partial (\rho vw)}{\partial z} = -\frac{\partial p}{\partial y} + \frac{\partial}{\partial x} \left( \mu \frac{\partial v}{\partial x} \right) + \frac{\partial}{\partial y} \left( \mu \frac{\partial v}{\partial y} \right) + \frac{\partial}{\partial z} \left( \mu \frac{\partial v}{\partial z} \right) + S_v \tag{7}$$

The performance on the energy conservation equation is as follows [4]:

$$\frac{\partial (\rho T)}{\partial t} + \frac{\partial (\rho u T)}{\partial x} + \frac{\partial (\rho v T)}{\partial y} + \frac{\partial (\rho w T)}{\partial z} = \frac{\partial}{\partial x} \left( \lambda \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( \lambda \frac{\partial T}{\partial y} \right) + \frac{\partial}{\partial z} \left( \lambda \frac{\partial T}{\partial z} \right) + S_T \tag{8}$$

Where: \(u, v, w\) are the velocity components of the fluid in the x, y, and z directions; \(\rho\) is the fluid density; \(\mu\) is the fluid motion viscosity coefficient; \(p, T, t\) represent the pressure, temperature, and time, respectively; \(C_p\) is Pressure specific heat capacity; \(\lambda\) is the thermal conductivity; \(ST\) is the internal heat source and viscous dissipation term of the fluid; \(Su, Sv, Sw\) represent the source term of momentum in each speed direction per unit time and unit volume.

4. Overall design

The chassis is made of low-carbon steel sheet by stamping. The outer dimensions (L×W×H) of the chassis are 465mm×455mm×190mm. It consists of one computer motherboard, one power module and two fans. The structure is shown in Figure 1. The computer motherboard mainly contains one CPU processor, four data exchange boards and one graphics processor. The power module is placed at the bottom of the chassis, and the motherboard is placed on the side of the chassis. The heat consumption of the working components on the main board is shown in Table 1. The distribution of power components on the main board is shown in Figure 2. To ensure that each working unit has a good heat dissipation effect to maintain normal working conditions, in front of and behind the chassis. The left side is provided with rectangular vents of equal width and the same width. At the same time, two axial fans of the same specification are placed behind the chassis to enhance the air convection around the chassis, so that the internal air is heated outward and the external cold air enters the chassis. It enhances the heat exchange between the interior of the cabinet and the surrounding environment.

![Fig. 1 Chassis structure form](image)
5. Fan and heat sink

The heat dissipation schemes commonly used in the thermal design include natural convection, forced air cooling, and liquid cooling. Considering the design characteristics of the computer and the electronic components of the internal high-density heat flow, the axial cooling fan is used for forced air cooling. Forced air cooling is stable, easy to maintain, and economical, making it the preferred solution for most electronic equipment cooling systems. Two axial fans of the type 109R0612D402 are used here, and the fan is in the form of a draft. When selecting a heat sink, it is necessary to fully consider the distribution pattern of the components of the heat sink, the specific production process, the fin spacing and specifications, the different joining processes of the fins and the substrate, and the flow velocity of the air in the air duct. According to the thermal analysis, the fin thickness can be reduced to increase the fin spacing under the condition that the number of fins is kept constant, and the heat dissipation effect of the heat sink can be enhanced to a certain extent. However, limited to economic costs and manufacturing levels, the minimum fin thickness cannot be less than 0.8 mm. The size of the fin spacing also affects the heat dissipation efficiency of the heat sink. The too small fin spacing is not conducive to air flow, which will increase the wind resistance. At the same time, the surface heat transfer of the fins may interfere with each other, and local heat accumulation will occur around the fins. The phenomenon, heat dissipation efficiency is reduced. If the fin spacing is too large, the overall heat exchange surface area may not meet the heat dissipation requirement in a short time, resulting in heat accumulation of the radiator. On the other hand, increasing the fin height can effectively increase the overall heat exchange area and improve the heat dissipation efficiency to some extent. However, in previous studies, it was found that the heat dissipation efficiency of the heat sink increases with the increase of the fin height. However, when the height of the fin reaches the critical value, the heat dissipation efficiency of the heat sink will gradually decrease. Therefore, the fin specifications on the heat sink of the present scheme are 60 mm × 23 mm × 2 mm, and the fin pitch is set to 5 mm.

Table 1

| position | Num | Tolerance temperature (°C) | Heat consumption/pie (W) | sum(W) |
|----------|-----|-----------------------------|--------------------------|-------|
| A        | 1   | 85                          | 65                       | 65    |
| B1-B4    | 4   | 85                          | 4                        | 16    |
| C        | 1   | 85                          | 5                        | 5     |
| Rest     | 19  | 85                          | 0.2                      | 3.8   |
| sum      |     |                             |                          | 89.8  |
6. Simulation model
Thermal simulation analysis and calculations were performed using Flotherm developed by FLOMERICS Software UK for electronic heat dissipation. In the software core design theory and solver calculation method, Flotherm not only adopts mature and perfect computational fluid dynamics and numerical analysis simulation technology, but also combines rich experimental data and empirical formulas of electronic heat dissipation. Compared with similar software, Flotherm has significant advantages in heat dissipation simulation of electronic systems. The software itself can establish a parametric virtual model according to the actual physical model, and can have mature applications in all aspects of chip level, printed board level, equipment level and system level. According to the modeling features and computational characteristics of Flotherm software, under the premise of ensuring the accuracy of the model simulation results, the simulation model of this scheme can be appropriately simplified, mainly as follows: (1) It is assumed that there is no radiation factor; 2) Ignore the low relative power consumption of the device, replace the geometrical structure with a regular stereo, and influence the heat dissipation on the chamfer, fillet, threaded hole, convex and concave features and irregular plane in the simulation model. Very small structures and features are necessary for simplification. The simplified model is shown in Figure 3. The outside air enters the interior of the chassis under the action of two axial fans behind the chassis, and is exchanged with the heat sink of the working unit and its internal parts to take away the work unit. Part of the heat and eventually escapes from the vents on the side of the chassis.

![Fig. 3 Model of thermal analysis](image)

7. Meshing
After the model is built and simplified, it can mesh. The density of the mesh has a direct impact on the accuracy of the simulation results and the amount of software calculation. In the case of a small amount of calculation, the local meshing technique can be used, that is, the mesh of the power component region is further finely divided. To achieve more accurate simulation results. According to the spatial distribution of the model and the expected temperature distribution requirements, the mesh density corresponding to the model is obtained, so as to ensure the accurate and reliable simulation results of the model, the calculation amount can be effectively reduced and the solution can be shortened. Time. For the experimental model, the mesh model obtained is shown in Figure 4. The number of cells and the maximum aspect ratio are 2464800 and 7.656234.
8. Results and analysis
The model of Fig. 3 is used for calculation. The ambient temperature is set to 30 °C, the inlet air inlet is 1 atm atmospheric pressure, and the heat loss of each component is taken as the value in Table 1. The residual curve and the monitoring point temperature curve after the model simulation calculation are completed. As shown in Fig. 6, it can be seen that the residual curve and the temperature curve of the monitoring point tend to be stable after about 500 iterations, and the iterative calculation converges [6]. After the software simulation calculation is completed, the Plot Editor window is opened, and the model post-processing window of the FLOtherm software is obtained. The temperature distribution of the model simulation can be obtained in the temperature field distribution cloud map, as shown in FIG. 7.

The simulation results show that the highest temperature inside the device appears on chip A, the maximum temperature is about 57.4 °C, and the maximum temperature of the data exchange board and graphics processor are lower than 70 °C. In the actual test, the temperature sensor is placed on the surface of the power component of the main board, and the device is placed in an incubator at 30 °C for one half hour, and then continuously energized to perform temperature measurement. After measurement, the maximum temperature on the main board is 55 °C, which is very different from the simulation result of Flotherm. The reason for the error may be that the simulation software ignores the radiation heat transfer effect and natural convection of the power components and the surrounding air, resulting in software simulation results. Slightly higher. It shows that CFD software has good applicability in model simulation analysis.
Fig. 5 Residual diagram

Fig. 6 Temperature monitoring point curve
9. End
On the whole, compared with the experimental test, the simulation software can be used to build the model and numerically solve the calculation, which can simulate the internal temperature field of the power component under different structural forms more quickly and intuitively, so that the designer can fully and accurately Master the overall heat distribution of the equipment under different conditions, so as to comprehensively evaluate the advantages and disadvantages of various heat dissipation schemes, and solve possible heat dissipation problems in time, helping designers to select design schemes that are more in line with heat dissipation requirements, so that the equipment is in a better work. Status. This improves the design efficiency and reduces the prototype cost of the prototype.

Acknowledgments
This work was supported by the Shenzhen Basic Research Project of China. (No. JCYJ20170302142203605). The authors would like to thank the colleagues for their essential contribution to the work.

References
[1] SU Shiming, LI Wei. Simulation and Analysis of Electronic Equipment Chassis Cooling [J]. Photoelectric Technology Applications, 2013, 28(3): 64-67. (in Chinese)
[2] QIU Chengti, ZHAO Dunshu, JIANG Quanxing. Structure Design Principle of Electronic Equipment [M], Nanjing: Southeast University Press, 2005. (in Chinese)
[3] YANG Shiming, TAO Wenquan. Heat Transfer [M]. Beijing: Higher Education Press, 2006. (in Chinese)
[4] NIU Dongke, JING Xiaoyi, ZHANG Xiangwei, et al. Thermal Simulation and Analysis of Electronic Circuits Based on FLOtherm [J]. Modern Electronic Technology, 2015, 38(6): 16-19. (in Chinese)
[5] ZHOU ju. Simulation Research on Thermal Analysis Method of Closed Electronic Equipment [J]. Computer Simulation, 2012, 29(11): 416-419. (in Chinese)
[6] FANG Zhiqiang, FU Guicui, GAO Zexi. Application of Electronic Equipment Thermal Analysis Software[J]. Journal of Beijing University of Aeronautics and Astronautics, 2003,29(8): 737-740. (in Chinese)

Fig. 7 Temperature distribution cloud of power components