Temperature rise simulation and thermal strain study of ring main unit

Xiang Wang[1], Linsen Mu[2], Enlin Li[2], Anxin Wang[2] and Mingge Wu[1,2,3,*]

[1] College of Mechanical and Electrical Engineering, Wenzhou University, Wenzhou325035, China

[2] Wecome Intelligent Manufacturing Co., Ltd., Wenzhou325000, China

[3] College of Mechanical Engineering, Nanjing University of Aeronautics and Astronautics, Nanjing210016, China

* Corresponding Author Email: wmg7810@wzu.edu.cn

Abstract. The internal structure of the Ring Main Unit is compact, the gap between the components is limited, and the heat dissipation performance is poor. Working in a high temperature environment for a long time will lead to structural deformation and aging, which will affect the service life and insulation performance, and may even lead to electrical accidents. In order to better solve such problems, this study uses the finite element multi-physics coupling simulation model. According to the characteristics of the Ring Main Unit, the internal components and the body are simplified, the calculation amount is reduced and the simulation accuracy is improved. In ANSYS Workbench, through the coupling calculation of steady-state heat and transient heat, the temperature distribution of the Ring Main Unit circuit under the condition of external temperature is obtained. The temperature change and distribution are further used for the thermal stress and strain analysis of the structural field to obtain the thermal strain distribution map of the conductive circuit under the influence of temperature, which provides theoretical support for the internal structure design of the new Ring Main Unit.

Keywords: Ring Main Unit, temperature distribution, multi-physical field coupling simulation, finite element simulation.

1. Introduction

Ring main unit, also known as insulated switchgear, is the key equipment in the field of power transmission and distribution. It is mainly composed of load switches, buses and circuit breakers. It is widely used in distribution stations and box substations in load centers such as urban houses, high-rise buildings and factories. At present, the research on the temperature rise of the ring main unit focuses on multi-physical field coupling analysis and internal structure optimization.

In the field of research of all parties, researchers of a certain model of overhead line switchgear cabinet carried out a series of temperature rise tests, and the data were analysed and compared in detail, and the factors affecting the temperature distribution inside the switchgear cabinet were summarized [1]. In the literature [2] a mathematical model for the estimation of the temperature rise of components installed on the distribution network during a short-circuit fault is presented. The model was applied to the common disconnecting switches of the network and the characteristics of the results were plotted according to the different forms of the asymmetrical part of the short-circuit current waveform and the initial value of the short-circuit DC current component. Guan's team considered the contact resistance and electromagnetic repulsion by establishing an equivalent contact bridge between the contact interfaces in the switchgear. The electromagnetic force and temperature obtained from the electromagnetic-thermal coupled field calculations are used as load inputs in the thermal-force coupled field analysis. The validity of the computational model is demonstrated by temperature rise experiments and comparison with other closed formulations. The dynamic behaviour of plug-in connectors under short-circuit conditions is also analysed [3]. Lijun Wang, Rui Wang, Xiaolin Li, and Shenli Jia used the electromagnetic fluid temperature multiphysics coupling method to model and simulate the 40.5kV medium voltage cubic gas insulated switchgear, and by comparing it with the experiment, it was found that the temperature rise changes significantly under certain

pressure [4]. The researchers explored the temperature field and thermal stress distribution of the dynamic and static contacts inside the ring main unit through finite element simulation, which provides a basis for the study of circuit breaker life. These studies provide valuable insights and methods to improve the performance and reliability of ring mains cabinets, reduce the temperature rise and extend the life of the equipment. Andre Mueller and other scholars have investigated and compared the geometrical characteristics of radiators with each other from the point of view of radiators in electrical equipment. Evaluating the effects of material selection, total surface area, temperature uniformity and maximum surface temperature, it was concluded that with increasing surface area, implementation of geometrical features and high thermal conductivity materials can reduce the maximum surface temperature and improve the overall heat dissipation [5]. Wang and colleagues proposed a MiNET deep learning model (MDLM) within the UPIOT environment, aiming to detect faults and diagnose electrical ring network cabinets. Through validation, MDLM demonstrated an impressive recognition accuracy of 99.1%, significantly surpassing other methods [6]. In another study, Jin Lei utilized a magnetic-flow-thermal coupled analysis method to investigate the steady-state thermal performance of GIS busbars. Based on the temperature rise simulation results, optimizations were made to the conductor and tank diameters [7]. Robby employed an RMU temperature rise simulation model to determine the temperature rise at critical locations within the RMU. This information was then used to establish the maximum duration of continuous overload for internal RMU components [8]. Li Fei and colleagues described a two-dimensional model to represent traditional rectangular busbars in a specific high-current switchgear model. They applied the finite element method (FEM) for electromagnetic field calculations, obtaining the distribution of busbar conductor current density and power losses. Considering the impact of proximity effects and skin effects, they designed an irregular busbar that improved the performance compared to the traditional rectangular busbar [9]. Other research efforts focused on optimizing the internal structure of ring network cabinets, effectively reducing temperature rise levels by enhancing component dimensions and structural designs. Wang and others conducted temperature rise simulations using theories related to eddy current fields, airflow fields, and temperature fields. They found that temperature rise was significant under natural convection conditions in ring network cabinets. To mitigate this, they introduced forced air cooling and improved internal contact structures, successfully lowering temperature rise levels [10]. These optimization methods also encompassed temperature rise simulations for switchgear of different sizes and structural designs aimed at enhancing heat dissipation.

In this study, a simplified three-dimensional ring network cabinet model is proposed. The steady-state thermal and transient thermal simulation analysis is carried out in the thermal analysis module of ANSYS workbench. The steady-state temperature distribution of the ring network cabinet under the condition of external temperature load and the relationship between the temperature rise of the ring network cabinet and the change of time are studied. The heat source and the higher part of the temperature rise are found. Finally, the thermal stress is simulated by thermal structure coupling. The thermal strain nephogram of each component under the action of thermal stress is studied.

2. Establishment of finite element model

2.1 Ring Main Unit finite element calculation mathematical model

2.1.1. Control equation of temperature field

The heat transfer equation of temperature field is usually based on the law of heat conduction (Fourier heat conduction law). In one-dimensional case, the heat transfer equation of temperature field can be expressed as:

$$\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2} \tag{1}$$

T is the temperature inside the object, t is the time, x is the space coordinate, α is the thermal diffusion coefficient. This equation describes the change of temperature with time and space, and the right side expresses the relationship between heat conduction rate and temperature gradient.

In the more general three-dimensional case, the temperature field heat transfer equation can be written in the following form:

$$\rho c \frac{\partial T}{\partial t} = \nabla(K \cdot \nabla T) + Q \tag{2}$$

ρ is the density of the object, c is the specific heat capacity, K is the thermal conductivity, and Q is the heat source term in the volume. The equation describes that the change of temperature field is affected by heat conduction, heat source and heat capacity.

The thermal stress control equation: After the conductor accepts the temperature load, thermal stress will be generated, which will cause deformation under the action of thermal stress. This process can be expressed by the thermal stress control equation.

$$\nabla\sigma + F = \nabla(\alpha\Delta + \rho c \frac{dT}{dt}) \tag{3}$$

∇ is the gradient operation symbol, σ is the stress tensor, α is the linear expansion coefficient, ΔT is the temperature change, ρ is the density of the object, c is the specific heat capacity, F is the external force.

2.2 The establishment of the physical model of the Ring Main Unit

The research object is a certain type of Ring Main Unit, which is composed of cabinet body and gas tank, mechanism box, instrument box, cable box and grounding device. The external cabinet material of the Ring Main Unit is mainly aluminum plate, and the interior is composed of instrument box, mechanism box, gas box cable box and grounding device. Its structure is shown in Figure 1.

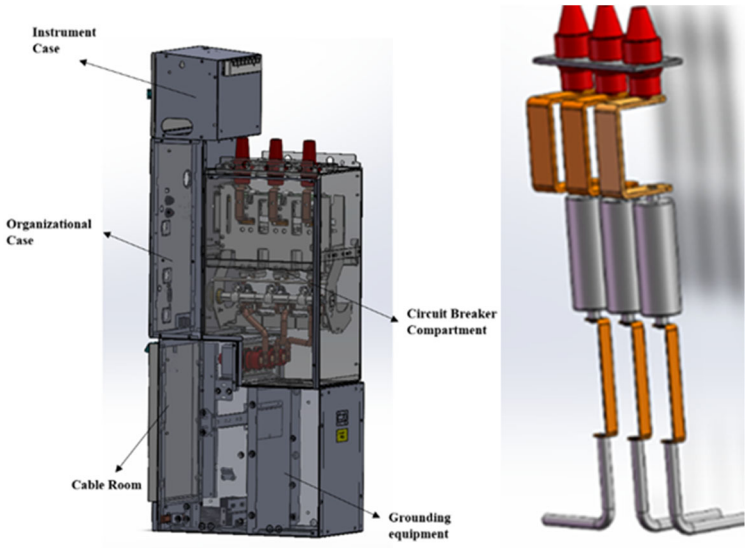

Figure 1. Ring Main Unit (left) and Ring Main Unit conductive circuit simplification (right)

The purpose of this study is to use the simplified three-dimensional Ring Main Unit model established on SolidWorks to import it into the thermal analysis module of ANSYS workbench for steady-state thermal and transient thermal simulation analysis : to study the temperature distribution cloud diagram of the model under the steady-state temperature load and the relationship between the temperature rise and time of the Ring Main Unit, to find the heat source and the higher temperature rise part, and finally to obtain the thermal stress. The thermal structure coupling simulation is carried out to study the thermal strain cloud diagram of each component under the action of thermal stress, and to study the degree of thermal strain of the conductive circuit of the Ring Main Unit under the action of temperature.

By reading the relevant literature and the Ring Main Unit test manual, the main body of the conductive circuit is composed of insulation sleeve, partition board, busbar, vacuum circuit breaker and busbar. The material properties are defined on ANSYS, and the specific parameters are shown in the following table.

Table 1. Material parameter table

parts	material	Thermal conductivity (W/m°C)	Thermal expansion coefficient (1/°C)
Insulation sleeve	Plastics	0.65	9.909E-05
Partition plate	Structural steel	60.5	1.2E-05
Busbar row	Copper alloy	401	1.8E-05
Vacuum circuit breaker	Aluminum alloy	175	2.3E-05
Branch bus	Copper alloy	401	1.8E-05

3. Simulation analysis

3.1 Steady-state thermal simulation

According to the given boundary conditions, the temperature of the branch bus reaches 400 °C in the steady-state thermal simulation setting. This temperature is transmitted from the branch bus to the surface of the conductive circuit, and convection heat dissipation is performed on the surface. The convection coefficient was set to 5W / (m ² • °C) and the ambient temperature was 20 °C. The simulation time is 10 seconds. Under the given boundary conditions, the distribution of temperature in the whole system can be obtained by steady-state thermal simulation.

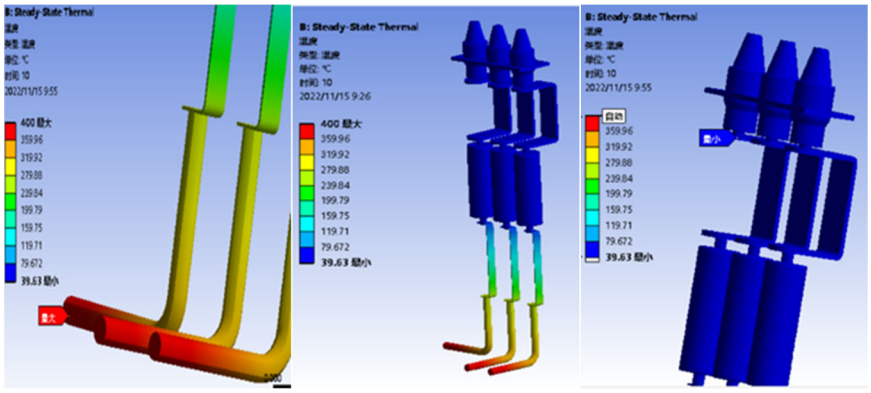

Figure 2. Steady-state temperature distribution cloud chart

After 10 seconds of temperature analysis, it is observed that the maximum temperature range in the whole system is 279.88 °C to 400 °C, which is mainly concentrated in the area of the branch bus and the lower bus. The main reason for this temperature distribution phenomenon is that the lower bus carries the Joule heat generated by the current transmitted by the cable chamber. Therefore, during the operation of the system, the highest temperature appears in the lower bus part. On the other hand, it is found from the diagram that the main bus and insulation sleeve of the circuit breaker are the lowest temperature areas in the system. Under the time limit of the solution, the heat is not further transferred, resulting in a minimum temperature of 39 °C in the region.

This conclusion shows that during the operation of the system, the lower bus is the region that withstands the highest temperature, while the main bus of the circuit breaker and the insulation sleeve are relatively low temperature regions. The characteristics of this temperature distribution are closely related to the relationship between current transmission and thermal energy conversion. In order to ensure the normal operation and safety of the system, we should focus on and effectively manage the temperature of the lower bus part, and take corresponding measures to control the temperature within an acceptable range.

3.2 Transient thermal simulation

The temperature results obtained from the steady-state thermal simulation are transmitted to the transient thermal module to study the temperature change of the system in a short time. In this process, the convection coefficient on the surface of the conductive loop is maintained at 5W / (m ²·°C), the ambient temperature is 20 °C, and the boundary condition of the simulation time is 10 seconds. Firstly, the results of steady-state thermal simulation need to be input into the transient thermal module as the initial temperature condition. In this way, from the steady-state temperature distribution of the system, we can observe how the temperature changes with time during the simulation time.

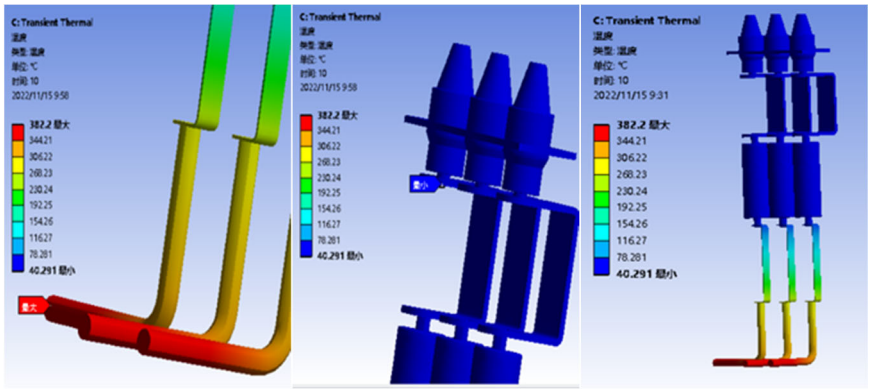

Figure 3. Temperature distribution nephogram of transient temperature field

Within ten seconds of the simulation, the overall temperature of the conductive circuit rises rapidly when the Ring Main Unit starts and the load changes rapidly. The maximum temperature reaches 382.2 ° C, and the position of the maximum temperature is at the lower bus part. With the passage of time, the heat is gradually transferred, and the branch bus continues to receive the heat transferred by the lower bus. The insulation sleeve and other devices with the same initial temperature have obvious temperature rise. The result of this temperature change is a direct reflection of heat transfer in the conductive circuit. Through transient thermal simulation, the temperature changes in different parts of the system can be understood in more detail and the impact of heat transfer on system performance can be evaluated. These results are of great significance for optimizing system design, improving temperature rise and predicting system performance.

3.3 Transient thermal simulation

According to the actual working conditions, the temperature load of the vacuum circuit breaker and the bus part under the gas pressure of 1200 Pa, the support around the baffle and the temperature load of the conductive circuit under the transient temperature will be considered, and the simulation will be carried out within 10 seconds to understand the thermal strain under the thermal load.

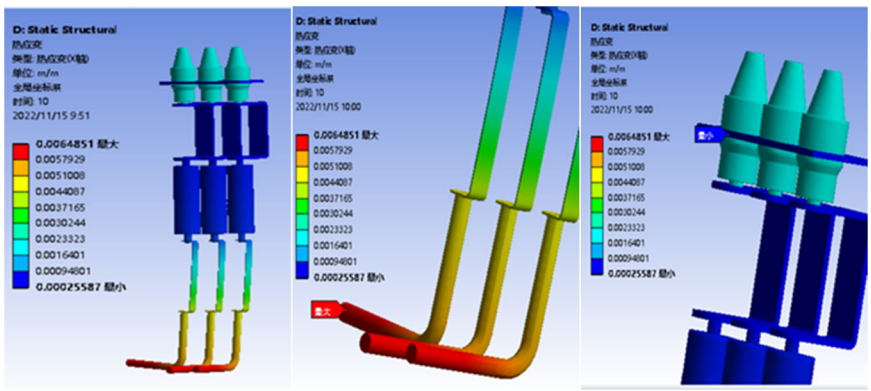

Figure 4. Thermal strain cloud diagram

After thermal simulation analysis, the thermal strain simulation focuses on the gas pressure and baffle support of the vacuum circuit breaker and the busbar part under actual working conditions. Under the action of temperature load, the lower busbar and the branch busbar of the conductive circuit are subjected to large thermal stress load. Therefore, the thermal strain of this part is more obvious. The same as the temperature distribution, as the temperature decreases, the degree of thermal strain is smaller. In the part of the insulated bushing, due to the dual load of the support and temperature of the baffle part, this part also has a certain degree of stress deformation.

4. Conclusion

For the steady-state thermal simulation analysis, the lower bus is the region that withstands the highest temperature, while the main bus and insulation sleeve of the circuit breaker are relatively low temperature regions. The characteristics of this temperature distribution are closely related to the relationship between current transmission and thermal energy conversion. In order to ensure the normal operation and safety of the system, we should focus on and effectively manage the temperature of the lower bus part, and take corresponding measures to control the temperature within an acceptable range.

Through transient thermal simulation, the temperature changes in different parts of the system can be understood in more detail. The result of this temperature change is a direct reflection of the heat transfer in the conductive loop and an assessment of the impact of heat transfer on system performance. Therefore, we can focus on the areas with large temperature changes to find the influencing factors. These results are of great significance for optimizing system design, improving temperature rise and predicting system performance.

According to the simulation results of thermal strain, it can be clearly seen that there is obvious thermal strain in the vacuum circuit breaker and the bus part, as well as the lower bus and the branch bus part of the conductive circuit. These thermal strains are caused by gas pressure, baffle support and temperature load. When the temperature decreases, the degree of thermal strain gradually decreases, which means that the stability and reliability of the system are higher under low temperature conditions. This is essential to ensure the long-term performance of the vacuum circuit breaker and the bus part. In addition, a certain degree of stress deformation is also found in the insulation bushing part, which is caused by the support and temperature double load of the baffle part. Therefore, in the design and manufacture of insulating bushings, these factors need to be considered to reduce stress deformation and improve the durability of the material.

Acknowledgment

This work was sponsored by the China Postdoctoral Science Foundation (2022M721604).

References

[1] L. Penghao, "Temperature Rise Test and Analysis of High Current Switchgear in Distribution System," China New Technol New Prod., vol. 24, no. 16, 2016.

[2] D. Polykrati, C. G. Karagiannopoulos, and P. D. Bourkas, “Thermal effect on electric power network components under short-circuit currents,” Electr. Power Syst. Res., vol. 72, no. 3, pp. 261–267, 2004

[3] X. Guan, N. Shu, B. Kang, and M. Zou, "Multiphysics Analysis of Plug-In Connector Under Steady and Short Circuit Conditions," in IEEE Transactions on Components, Packaging and Manufacturing Technology, vol. 5, no. 3, pp. 320-327, March 2015.

[4] Lijun Wang, Rui Wang, Xiaolin Li, and Shenli Jia, "Simulation Analysis on the Impact of Different Filling Gases on the Temperature Rise of C-GIS," presented at [Conference Name], [Year].

[5] Mueller, C. Buennagel, S. Monir, A. Sharp, Y. Vagapov, and A. Anuchin, "Numerical Design and Optimisation of a Novel Heatsink using ANSYS Steady-State Thermal Analysis," in 2020 27th

International Workshop on Electric Drives: MPEI Department of Electric Drives 90th Anniversary (IWED), Moscow, Russia, 2020, pp. 1-5.

[6] Y. Wang, J. Yan, Q. Sun, J. Li, and Z. Yang, "A Mobile Nets Convolutional Neural Network for GIS Partial Discharge Pattern Recognition in the Ubiquitous Power Internet of Things Context: Optimization, Comparison, and Application," in IEEE Access, vol. 7, 2019, pp. 150226-150236.

[7] Jin Lei et al., "A 3-D Steady-State Analysis of Thermal Behavior in EHV GIS Busbar," Journal of Electrical Engineering & Technology, vol. 11, 2016, pp. 781-789.

[8] Ouerdani et al., "Temperature rise simulation model of RMU with switch-fuse combinations for future load profiles," presented at CIRED 2021 - The 26th International Conference and Exhibition on Electricity Distribution, Online Conference, 2021.

[9] Size Li et al., "multi-physical field coupled analysis of heavy-current irregular busbar," presented at 2018 13th IEEE Conference on Industrial Electronics and Applications (ICIEA), 2018.

[10] L. Wang, W. Zheng, L. Wang, J. Lin, X. Li, and S. Jia, "Electromagnetic-Thermal-Flow Field Coupling Simulation of 12-kV Medium-Voltage Switchgear," in IEEE Transactions on Components, Packaging and Manufacturing Technology, vol. 6, no. 8, Aug. 2016, pp. 1208-1220.