Analysis of vibration characteristics of mounting plate for molded case circuit breaker

Ying Pan1, Zhixian Tang and Zhi Cheng

School of Mechanical and Automotive Engineering, Shanghai University of Engineering Science, No. 333, Longteng Rd., Shanghai, China, 201620

1 E-mail: pying_gclx@sues.edu.cn

Abstract. This paper presents a framework to analyze the vibration characteristics of mounting plates for molded case circuit breakers (MCCB). Two different three-dimensional models for mounting plate I and II are established in ANSYS Workbench simulation platform through reasonable simplifications. The preceding six-order natural frequencies are obtained by means of modal analysis. Based on the experimental data of modal analysis, random vibration analysis of mounting panel I and II is conducted. It is concluded that mounting plate I is easy to deform permanently when its 3δ stress exceeds the yield limit of material, while mounting plate II meets the strength requirements and is better suited to test requirements.

1. Introduction

With the rapid development of new electrical technologies, the dependability requirements for MCCB and other electronic products are becoming more and more stringent [1]. As the main electrical equipment in the distributing systems, MCCB play the important role in connection and disconnection with the main circuits.

So far, there are many test methods and platforms for research on reliability index of MCCB2-4. In [2], the dynamics simulation calculation of circuit breaker for equipment was carried out with the software of ADAMS to research on vibration-resistant property. Meanwhile, the transient dynamics simulation was calculated with ANSYS, and the calculation results in various frequency and acceleration conditions can be obtained. In [3], a circuit breaker (CB) aging failure model based on condition monitoring was presented. This model was inserted into a basic CB Markov model and applied in substation reliability assessment. The proposed model can accurately incorporate the wearing status and aging process of CBs. In order to solve the problem of failure mechanism of low voltage electrical apparatus under the condition of vibration shock, the simulation analysis of the failure of the circuit breaker was carried out based on the finite element method in [4]. According to the simulation analysis, the stress distribution of the circuit breaker was obtained. The feasibility of the research method of the failure mechanism of vibration shock was verified, which provides an effective method for the structural design and material optimization of circuit breaker.

It is noted that test environments in above literatures are static. Random vibration is a very common circumstance which the vast majority of cases in rockets or vehicles may frequently encounter. When random vibration occurs, MCCB may appear to loose parts or suffer friction, which can significantly affect the reliability of product. In addition, random vibration within a long time will cause fatigue failure and lead to incorrect operation or release, which can degrade performance of MCCB.
This paper presents a framework to analyze the vibration characteristics of mounting plates for MCCB under a dynamic condition. The rest of this paper is organized as follows: Section 2 introduces two different three-dimensional models for mounting plate I and II, which are established by ANSYS Workbench simulation platform. Random vibration analysis of mounting panel I and II is conducted in section 3. Section 4 concludes the paper.

2. Modal analysis for mounting palates

There will be a lot of problems such as line elements missing and plane elements missing when the model is directly imported into ANSYS software through CAD software. ANSYS provides a good modeling environment with rich modeling instructions, and there are also functions which some other CAD software does not supply, such as beam modeling, enclosure operation, spot welding set and so on.

To avoid data loss during the process of modeling, this paper utilizes Design Modeler model of ANSYS to model the mounting plates. Models of mounting plates are reasonably simplified by removing subordinate factors like fillets, chamfers, bolts and holes to improve the efficiency of modeling while maintaining precision of analysis.

This section discusses two kinds of mounting plates, the mounting plate I can be used to install a number of different types of circuit breaker, its simplified model is shown in Figure 1 (a). Mounting plate I is specifically designed to install the 3VT circuit breaker, and its simplified model is given in Figure 1 (b).

![Figure 1. Simplified model of mounting plate I and II. (a): mounting plate I (b): mounting plate II.](image)

The material of mounting plate I is made of structural steel, the elastic modulus is 210GPa, the density is 7850kg/m³, and the Poisson's ratio is 0.3. The material of II is mainly made of aluminum alloy. The elastic modulus is 710GPa, the density is 2770kg/m³, the Poisson's ratio is 0.33.

2.1. Finite element models for mounting plate I and II

Contact is a very common phenomenon in assembly engineering such as bolt connection, assembly and tight coordination. It is one of the most frequently used nonlinear characteristics by Workbench users [5]. Contact analysis usually consume huge computing resources, so it is critical to form a reasonable model [6]. Contact pairs used in workbench are composed of the target surface and the contact surface, different contact types will have significant impacts on the accuracy of contact [7, 8]. Take mounting plate II for example to illustrate the finite element analysis, Mounting plate II is assembled by combining a plurality of parts together, in order to simulate actual working conditions,
real contact model is adopted to carry out finite element analysis. Except the contact between bolt and threaded hole, other contact pairs are set to be binding type. There does not exist stress concentration in the modal calculation process in structural dynamics. Therefore, mesh is uniformly refined and is not be partial encryption. Figure 2 shows the mounting plates after meshing.

![Figure 2: Finite element models of mounting plate I and II. (a): mounting plate I (b): mounting plate II.](image)

After meshing, mounting plate I has 36602 nodes and 18004 units, while mounting plate II has 106905 nodes and 22030 units. Because the two mounting plates are fixed on the shaking table test, fixed constraints are applied to mounting plate I and four bolt holes of mounting plate II.

2.2. Modal analysis for mounting plate I and II

In the actual structure, the preceding six-order natural frequencies are generally concerned. The vibration response to structure is the largest when the external excitation frequency recloses with one of the preceding six-order natural frequencies. There is a structure damping in the actual state, although the damping is small, the structure damping effects the high frequency vibration. Figure 3 to Figure 8 show the preceding six-order modes for mounting plate I and II.

From simulations, we learn that the first order natural frequency of mounting plate I is 6.97Hz, its vibration mode is swinging back and forth in the XY plane, and the maximum vibration occurs at the top. The second order natural frequency is 30.39 Hz, its vibration modes is torsion deformation in the XY plane. The third order natural frequency is 56.26Hz, its mode is blending deformation in the XY plane. The fourth order natural frequency is 112.02Hz, its mode is about a substantial twist in the XY plane and a slight twist in the XZ plane. The fifth order natural frequency is 134.18Hz, its vibration mode is torsion in both XY plane and XZ plane. The sixth order natural frequency is 146.59Hz, its mode is twist form left to right in the XY plane and swinging up and down in the XZ plane.

The first order natural frequency of mounting plate II is 177.63Hz, its mode is a vertical plate with the vibration gradually increasing from bottom to top, and swinging horizontally in the XY plane. The second and third natural frequency is 322.12 Hz and 380.03Hz respectively, its mode is a vertical plate with a slight twist in the XY plane. Strongest vibration occurs at the top of the two corners. The fourth order natural frequency is 549.87Hz, its mode is a fixed plate with its left part’s vibration gradually increasing from the lower to the upper. The fifth order natural frequency is 707.62Hz, its mode is a vertical plate swinging in both XY plane and XZ plane. The sixth order frequency is 883.54Hz, its mode is a fixed plate with twist deformation in the left.
Figure 3. First order modes of mounting plate I and II. (a): mounting plate I (b): mounting plate II.

Figure 4. Second order modes of mounting plate I and II. (a): mounting plate I (b): mounting plate II.

Figure 5. Third order modes of mounting plate I and II. (a): mounting plate I (b): mounting plate II.
Figure 6. Fourth order modes of mounting plate I and II. (a): mounting plate I (b): mounting plate II.

Figure 7. Fifth order modes of mounting plate I and II. (a): mounting plate I (b): mounting plate II.

Figure 8. Sixth order modes of mounting plate I and II. (a): mounting plate I (b): mounting plate II.
Through the comparative analysis of modal analysis for mounting plate I and II, the preceding six natural frequencies of mounting plate I are less than 150Hz, it has great amplitude of bending and torsion, which have a great influence on the subsequent tests. The preceding six natural frequencies of mounting plate II are beyond 150Hz, then do not coincide with the vibration frequency of the circuit breaker.

3. Random vibration analysis for mounting plates

Based on the modal analysis, ANSYS Workbench is used to analyze the random vibration of the mounting plates. In the ANSYS simulation, simulations are based on the standard TB/T 3058-2002 in the shock and vibration test of the locomotive and rolling stock equipment.

By adding vertical (Z direction), transverse (X direction), longitudinal (Y direction) directions to mounting plates, we obtain the calculation results of stress distribution of power spectra as well as 3δ displacements in accordance with the directions. The results are shown in Figure 9 to Figure 12.

The results of ANSYS analysis of random vibration are probability values, Figure 9(a) shows that the maximum stress under 3σ are no more than 700.3Mpa when mounting plate I is under the random loading. This value is greater than the yield strength of material, which means that the probability of the model occurs under the stress more than 700.3MPa is a small probability event. Hollowed part of the mounting plate I prone to permanently deform. From Figure 10(a), we know that the 3σ deformation of edges of hollowed part in the X direction is 0.041mm, from Figure11(a)we know that 3σ deformation occurred in the mounting plate I in the Y direction of Y is 2.553mm and this occurs in the middle part of the edge. From Figure 12(a) we know that 3σ deformation of mounting plate I in the Z direction is 73.896mm, and this deformation takes place at the top of mounting plate I.

As shown in Figure 9(b), the 3σ stress of mounting plate II under random load is 8.933MPa, which is far less than the yield strength of the material. It occurs in the four mounting holes at the bottom. From Figure 10(b), the 3σ deformation in the X direction is 3.36e-3mm, which occurs at the top of the vertical plate. From Figure 11(b) we know that deformation in the Y direction is 2.55e-3mm, which occurs at the bottom of bolt. Figure 12(b) shows that the 3σ deformation in the Z direction is 3.10e-2mm, which occurs at the top of the vertical plate. From above we learn that deformations occurs in the mounting plate II under the condition of random vibration are very small, hence, the accuracy of the test results is guaranteed.

(a) Figure 9. Stress of mounting plate I and II. (a): mounting plate I (b): mounting plate II.
Figure 10. Displacement in X direction of mounting plate I and II. (a): mounting plate I (b): mounting plate II.

Figure 11. Displacement in Y direction of mounting plate I and II. (a): mounting plate I (b): mounting plate II.

Figure 12. Displacement in Z direction of mounting plate I and II. (a): mounting plate I (b): mounting plate II.
4. Conclusions
This paper analyzes the modal analysis of the mounting plates by ANSYS Workbench. Based on modal analysis of the workpiece, random vibration analysis is carried out. Conclusions are obtained as follows:

1. The preceding six order natural frequencies of mounting plate I are within 150Hz, and its amplitude of bending and torsional deformation is significant, while the preceding six natural frequencies of the mounting plate II are beyond 150Hz, so molded case circuit breaker vibration test is not easy to be excited for plate II.

2. The 3δ stress of the mounting plate I is greater than the yield limit of the material, and the permanent deformation is very easy to happen. The analysis results of the mounting plate II meet the strength requirements.

3. According to the simulation results, the performance of the mounting plate II is good, so the mounting plate II should be modeled when designing the mounting plate.

Acknowledgments
This work was supported by capacity building Program for local colleges in Shanghai (14110501200).

References
[1] Wang XL 2017 Summary of the current development of vibration environmental testing equipment and testing technology Engineering Technology (1) 00278-00278
[2] Zhang H and Liu HY 2015 Simulation and Experimental Research on Vibration-Resistant Property of Circuit Breaker for Equipment Electrical & Energy Management Technology
[3] Zhong J, Li W and Billinton R 2015 Incorporating a Condition Monitoring Based Aging Failure Model of a Circuit Breaker in Substation Reliability Assessment IEEE Transactions on Power Systems 30(6) 1-9
[4] Huang R, Miao XD and Center E T. 2016 Simulation Analysis on Failure of Low Voltage Electrical Shock and Vibration Journal of Shanghai Second Polytechnic University
[5] Zhong GS, Ao LP and Zhao K 2012 Influence of explosion parameters on wavelet packet frequency band energy distribution of blast vibration Journal of Central South University 19(9) 2674-2680
[6] Debayan D, Saurav R and Anush K. 2015 Design and finite element analysis of connecting rod using Solidworks and ANSYS workbench International journal of research in Engineering and Advanced Technology 4(3) 88-96
[7] Niu C, Zhang J and Chen D 2006 Dynamic simulation of operating mechanism in low-voltage moulded-case circuit breaker IEE PROCEEDINGS-GENERATION TRANSMISSION AND DISTRIBUTION 153(4) 451-0
[8] Park H J, Jung H K and Ro J S 2016 Analysis and design of diverse electromagnet type actuators for moulded case circuit breaker IET Electric Power Applications 10(9) 849-857