Finite Element Analysis of Monorail Brake Caliper Based on Solidworks

Hu Li [a], Yunxiang Wang [b], Kunpeng He [c]

[1]School of Mechanical and Electronic Engineering, Shan Dong University of Science and Technology, Qing dao 266590, China

[a] skd_fox@foxmail.com, [b] 211244942@qq.com, [c] 695684093@qq.com

Abstract

The brake mechanism is an important mechanism for monorail cranes and plays an important role in the safety and stability of the entire monorail hoist system. The three-dimensional model of the monorail suspension brake caliper was established using Solidworks finite element software. The finite element analysis was verified and the optimization scheme was proposed. Through the analysis and research in this paper, it provides reference data and theoretical basis for other similar devices.

Keywords

Monorail Brake; Caliper;Solidworks;finite element analysis.

1. Introduction

Monorail cranes are inevitably affected by many factors such as working space and geological conditions due to their working in the special environment under coal mines. On the one hand, the brake system is directly exposed to high dust gas due to the large concentration of dust gas in the well. In the environment, this puts a strict explosion protection requirement on the braking system. Therefore, the selection of brake friction materials is quite special. On the other hand, underground tunnels are relatively narrow, with many ramps and poor working conditions. This requires that the overall size of the braking system for monorail hoists should not be too large, so as to avoid affecting normal operation. At the same time, under the premise of mine safety first, the safety and reliability of the braking system must be guaranteed.

The brake is the main brake actuator of the brake system and is the brake component used to create a tendency to hinder monorail movement or movement. It plays the role of deceleration and parking during driving, maintains relatively stable speed on the downhill, and realizes the parking function for a long time on the ground or on the slope.

The People's Republic of China Coal Industry Standard MT/T591-1996 has a clear performance requirement for a monorail crane braking system:

The brake blocks used must not be made of plastic or resin-reduced material and must be made of materials that will not detonate or burn when braking.

The centrifugal release device of the braking device shall have two sets of equal structural performance, and it must be ensured that at least one of the two is always engaged with the rail surface during operation.

The braking force of each transportation device shall not be less than 1.5-2 times of the rated traction force. When the brake shoe wears to the limit specified by the manufacturer, it must still have a rated braking force.

The braking device must be provided with both manual and automatic control devices and should have the following properties:

When the running speed exceeds 15% of the maximum speed, the brake can be automatically braked. When the maximum speed is not higher than 2m/s, it is allowed to brake automatically when the running speed exceeds 30% of the maximum speed.

The value of the idle time for the braking device must not exceed 0.7 s.

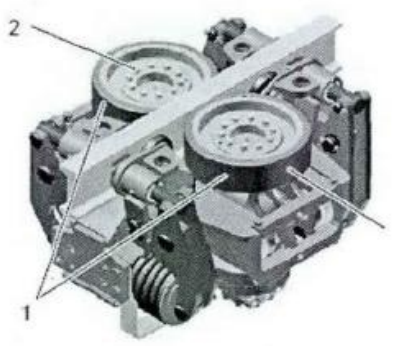
When running at the maximum design speed at the maximum load and maximum gradient, the braking distance does not exceed the 6s stroke corresponding to this speed, and the braking deceleration does not exceed 5m/s at the minimum load and the maximum gradient.

The anti-knocking property of the brake device.

The brake block used in the braking device shall not cause explosion of dangerous gas under the coal mine when braking at the maximum load and the specified maximum design gradient.

2. Braking Structure Design

At present, the domestic multi-purpose brake structure is a clamp-type hydraulic mechanism (three-bar type), using the Scharf hydraulic brake mechanism, as shown in Figure 1.2:



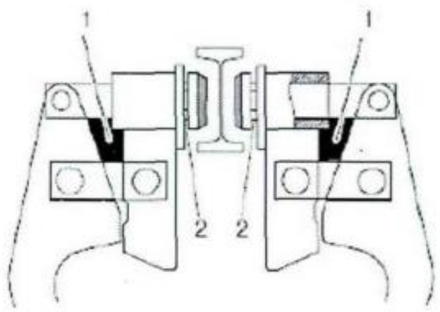


Figure 1.2. Scharf hydraulic brake mechanism

In this braking mode, the spring is sleeved on the hydraulic cylinder body, reducing the space layout. At the same time, because of the greater pressure provided by the hydraulic system, the lower link can be designed to be relatively short and the overall structure is small.

3. Calculation and Selection of Springs

The brake mechanism adopts spring brake, so after determining the basic structure of the brake mechanism, it is necessary to calculate and select a suitable spring. Since the cylindrical spiral compression spring has the advantages of simple structure, convenient manufacture and good safety, the cylinder spiral compression spring is selected. In addition, it is necessary to calculate the braking force and stiffness of the spring, select materials, and calculate the minimum diameter of the spring. The specific work is described below.

Calculation of spring braking force and stiffness

The brake mechanism rocker can be reduced to a lever, and the spring brake force can be calculated using the lever principle. When the braking mechanism is braked, the force and deformation of the brake spring are shown in Figure 2.1 and Figure 2.2 respectively:

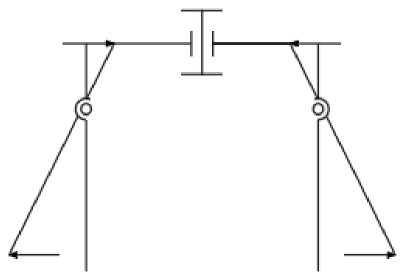


Figure 2.1. Brake spring force diagram at brake sluice

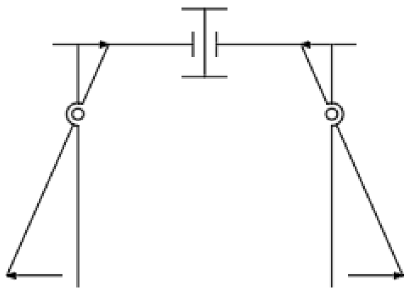


Figure 2.2. Brake Spring Distortion Diagram at Gate Brake

In the design of the brake structure, in order to reduce the pressure on the spring and the hydraulic cylinder, a 1:3 leverage mechanism was used in this design.

Material selection

In order to enable the brake spring to work reliably, the brake spring material must have a high elastic limit and fatigue limit, and should have sufficient toughness and plasticity, as well as good heat treatability. With reference to commonly used spring materials, primary hot-rolling (60Si2Mn A) can be selected.

According to Table 16-2 of the 8th Edition of the "Mechanical Design", the type II spring is a spring loaded with 103-105 impact loads. The allowable shear stress of 60Si2Mn A material is 640MPa, the shear modulus is 78500MPa, the tensile strength is 1880MPa, =1200MPa. According to the spiral ratio C value commonly used in the manual, the C value can be selected between 4-16. This design primary election C=5.

Curvature coefficient:

$$K=\frac{4C-1}{4C-4}+\frac{0.625}{C}=1.32 \tag{1}$$

The minimum diameter of spring wire:

$$d \geq \sqrt{\frac{8KFC}{\pi\tau}}=28.6\text{mm} \tag{2}$$

F-brake required spring force, N.

Query the diameter of the spring in Table 4-2-6 of the Mechanical Design Handbook, Version 3 , $\mathrm{d}=20$. Therefore, the diameter of the spring $\mathrm{d}=Cd=96mm$ .According to the "Mechanical Design and Use Manual" third edition of the general aid spiral compression spring size and parameters 4-2-11 query: $\mathrm{d}=16mm$, $D=85mm$, $F_n=11970N>8572N$, meet the design requirements.

4. Model Establishment and Finite Element Analysis

In order to judge the reliability of the structural dimension design, the program uses the Simulation module in Solidworks to complete the static analysis of the main structure of the brake device. In accordance with the material properties of the added parts (brake lever, brake arm, and carbon structural steel Q235 used in the frame, the yield strength is 240 MPa, the friction plate is made of bronze, and its yield strength is 239 MPa), the static analysis process of the established model is finally completed by defining the sequence of links, defining contact sets of components, adding fixtures, applying external loads, performing meshing, and solving the analysis sequence.

Because the model loads, materials, and structures have symmetry, the symmetry can be used to analyze only a quarter of the structure. The solution model is shown in Figure 3.1. Define the contact surface groups of the brake block and the guide rail, the gasket and the tie rod shaft, and set the contact type of the component as the non-penetration type, and establish the symmetrical restraint and the fixed restraint of the rack and the guide rail respectively. Since only the stress and strain of the main components of the brake end state are analyzed, a constant load can be applied to the lower end of the brake arm. The horizontal force of 30 kN is used to represent the spring pressure. When dividing the mesh, the curvature-based mesh is selected, the mesh is generated by the variable elements, which is advantageous for obtaining accurate results at the fine features of the geometry. The running analysis of the constructed example can give the model of stress, strain and displacement. As shown in Figure 3.2, the maximum von Mises stress is used as the yield failure criterion.

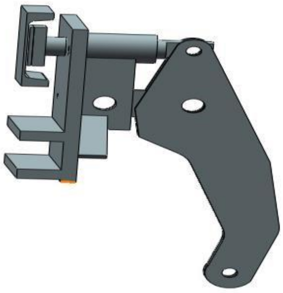


Figure 3.1. Simplified model

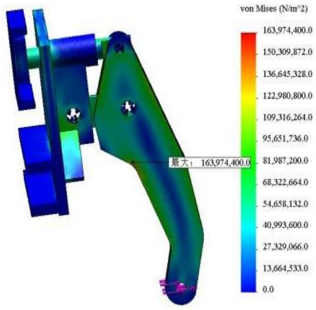


Figure 3.2. von Mises stress plot

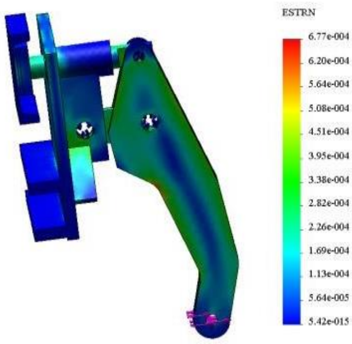


Figure 3.3. Strain Diagram

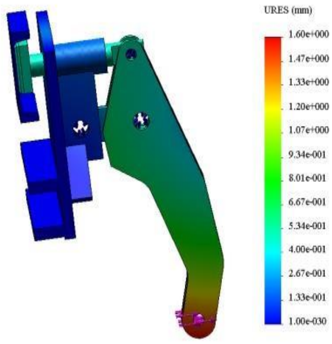


Figure 3.4. Displacement diagram

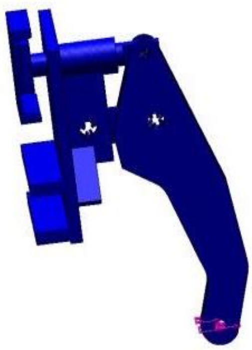


Figure 3.5. Distribution of safety area

5. Finite Element Analysis Results

It can be seen from the stress diagram 3.2 that the maximum stress value of this structure is 163.9 MPa, which is lower than the yield strength of the part material. Due to stress concentration, the maximum stress occurs at the corner of the brake arm. Considering that the area is small and the stress is not greater than the yield limit, its influence on the structure can be ignored. From the strain map 3.3, the maximum strain appears in the maximum stress region, with a maximum of 0.67 mm. It can be seen from the combined displacement diagram that the maximum displacement occurs at the connection of the spring tray and the brake arm, and the maximum displacement is 1.6 mm, which is relatively small with respect to the spring travel. Figure 3.5 shows the distribution of safety zones based on the maximum von Mises stress for yield failure criteria. Red represents non-safe areas and blue represents safe areas. Since the maximum stress of the structure does not exceed the yield limit of the material, the area is displayed in blue, and the lowest safety factor appears in the maximum stress area, but the safety can still be guaranteed.

6. Conclusion

In the past, the analogous method was usually used to design the monorail braking system components. Then the strength and stiffness of key components were checked using the material mechanics formula. Although this kind of method is safe and reliable, it takes time and effort. Here, the finite element method is used to perform finite element analysis on monorail lifting brake calipers, which is not only fast but also simple and intuitive. In this paper, through the finite element analysis of monorail hoisting brake calipers, the rationality of monorail hoisting brake calipers is proved, and the scheme of equipment optimization is also proposed, which provides reference data and theoretical basis for other similar equipment.

References

[1] Yan Yanjun. Design and performance study of underground monorail crane braking system[D]. Master thesis of Shandong University of Science and Technology, 2011.

[2] Liu Yangyang, Gu Daoda, Zhang Lixiang. Design and Analysis of Monorail Crane Braking Mechanism [J]. Coal Technology, 2015, 34(03), 253-254.

[3] ANSYS 16.0 Introduction to Finite Element Analysis Advanced Proficient/CAE Application Alliance Group [M]. Beijing: Mechanical Industry Press, 2016, 4.

[4] Zhou Ning. ANSYS Mechanical Engineering Application [M]. Beijing: China Water Power Press, 2006.

[5] Jiang Hong, Chen Yu, Wang Zhi et al.. SolidWorks finite element analysis example analysis. Beijing: Mechanical Industry Press, 2007.

[6] Li Renxian. Foundation of Finite Element Method [M]. Beijing: National Defense Industry Press, 2004.

[7] The People's Republic of China Coal Industry Standard MT/T883-2000, Monorail Diesel Locomotive [M]. Beijing: Coal Industry Press, 2004.

[8] Zheng Licheng. Design and Analysis of Key Components of Monorail Crane Brake[J]. Coal Mine Electromechanical, 2017, 12(02), 72-74.