PAPER • OPEN ACCESS

Finite Element Analysis of Combination Condition of ZF6400/19/32 Hydraulic Support

To cite this article: Wenbin He *et al* 2019 *IOP Conf. Ser.: Mater. Sci. Eng.* **490** 052006

IOP Conf. Series: Materials Science and Engineering **490** (2019) 052006 doi:10.1088/1757-899X/490/5/052006

Finite Element Analysis of Combination Condition of ZF6400/19/32 Hydraulic Support

Wenbin He, Zhijun Chen, Jinguang Du*, Shaotai He, Hengyang Yao

Henan Key Laboratory of Intelligent Manufacturing of Mechanical Equipment, Zhengzhou University of Light Industry, Zhengzhou, China

*Corresponding author e-mail: dujinguang@zzuli.edu.cn

Abstract: The stress distribution of the ZF6400/19/32 hydraulic support under combined conditions is a challenging problem since the complexity of the working condition. In this paper, We studied the stress distribution characteristics of hydraulic supports under the combined conditions of the canopy bending & the base horizontal loading, the canopy hind torsion & the base bending. And the dangerous and weak areas of the hydraulic support under various working conditions are determined. The original design scheme of the ZF6400/19/32 hydraulic support is improved and optimized based on the analysis results. The research provides useful reference for the design of the hydraulic support, and improves the hydraulic support safety and reliability in the actual work.

1. Introduction

The hydraulic support is used to manage the activities of the roof, floor, control working face and the gob rock stratums mainly, which provides a safe and reliable working space for the personnel and equipment in the coal mining face [1]. Undering actual working conditions, the hydraulic support mainly bears the pressure of surrounding rock, the supporting force of the ground and the internal load of the hydraulic support column.

A positive four-bar linkage chock-shield hydraulic support is studied in this paper, which is composed of the box-shaped welded components such as canopy, caving shield, base and linkage, these components are connected by pin shaft. The integral support resistance is provided by internal loading of the hydraulic support column [2]. The front column socket of the base and canopy is designed with inclined structure, which can increase the contact area between hydraulic cylinder and column socket, the column head and socket. It has the advantages of large coal mining space and even stress, etc.

At present research, the structural strength of hydraulic support is mostly calculated under single load case by the general specification of MT312-2000. The mechanical behavior of the structure under typical working conditions such as canopy bending, canopy torsion, base bending and base torsion are studied respectively. The single load case can not accurately reflect the actual stress situation of the support in the mine. Practice research shows that the combined working conditions of the canopy bending & the base horizontal loading, the canopy front torsion & base bending, the canopy hind torsion & base bending are more severe, which are easy to damage the hydraulic support structure, and are more in line with the actual complicated conditions in the mine [3]. Referring to the test standard of the hydraulic support in an enterprise, the combined working conditions of the canopy bending & the base horizontal loading, the canopy hind torsion & base bending are simulated based on ABAQUS platform in this paper. The stress distribution characteristics of hydraulic support under two working conditions are analyzed, and the dangerous areas with stress concentration and structural damage are determined. On this basis, the design scheme is improved and optimized to ensure the safety of the hydraulic support during service.

2. Geometric Modeling of hydraulic support

Due to the structure of the hydraulic support is complex, it is modeled by Solidworks [4], and imported into ABAQUS in Parasolid format. The model of hydraulic support needs to be simplified before analysis.

(1) The functional accessories that do not affect the structural strength of the components are omitted. Including but not limited to the following parts: pipe clamp, hanging ring, disc, baffle seat, ring, nameplate [5].

(2) Small holes distributed in areas with low stress levels and without large stress concentrations can be simplified. The remaining holes should retain the original structure as far as possible.

(3) The weld chamfers, casting fillets less than 10 mm and transition arc length less than 30 mm are simplified in actual structure.

(4) In order to simulate the welding relationship between the components in the welded structure, the Boolean merging operation is performed on components.

The simplified model is shown in Fig. 1.

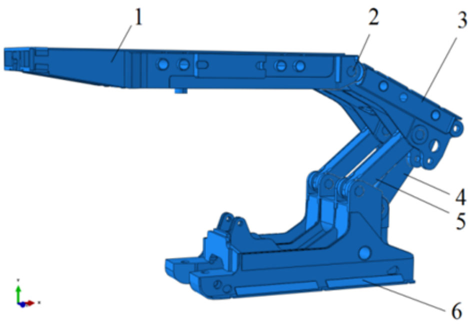


Fig. 1. Simplified geometric model of the hydraulic support
1. Canopy 2. Pin shaft 3. Caving shield 4. Rear linkage 5. Front linkage 6. Base

3. Finite element modeling of hydraulic support

3.1. Material and property

The main material used in the bracket structure is Q460, 16Mn, etc. The Poisson's ratio and elastic modulus of the two materials are similar. The parameters are shown in Table 1.

IOP Conf. Series: Materials Science and Engineering **490** (2019) 052006 doi:10.1088/1757-899X/490/5/052006

Table 1. Material properties

elastic modulus /MPa	Poisson's ratio	Density /t·mm^{-3}
2.1×10^5	0.28	7.85×10^{-9}

3.2. Establishing hinged relations between components

In Fig. 1, the hydraulic support is assembled by the components, such as canopy, caving shield, base, front linkage and rear linkage, these parts are connected by pin shaft. The connection relationship between components should accurately reflect the way and path of force transmission when the hydraulic support is carried. There are two ways to establish hinged relations.

Firstly, Using the beam element as the axis pin, and the constraint relationship between the beam and nodes near the hole is established on the pin-hole section of the beam element [6]. Through this constraint, the force transmission relationship between the axis pins and beam is realized. This method linearizes the nonlinear problem, and expedites calculation speed, but the pin-hole position calculation is not accurate [7].

Another method is to establish the axis pin substantial contact model. The contact area and position of the axis pin and pin-hole vary with load, this belongs to a typical boundary nonlinearity problem. The force acts on the axis pin and pin-hole can be truly simulated by substantial contact model.

This study used the second method to deal with the hinged relationship between components.

3.3. Displacement boundary condition at the block

According to the requirements of the test specification, the combined working condition of the canopy bending & the base horizontal loading is shown in Fig. 2. There are Two rectangular long blocks are arranged at the canopy front and rear column sockets respectively, and a rectangular and a triangular long stop blocks are arranged at the canopy front end and base rear end respectively.

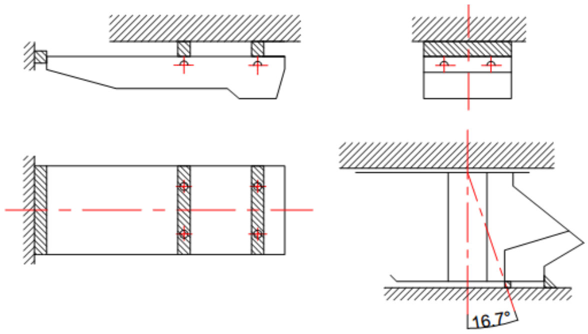


Fig. 2. Schematic diagram of the block position in working condition of the canopy bending & the base horizontal loading

For the combined working condition of the canopy hind torsion & base bending, as shown in Fig. 3. A rectangular long and a short blocks are respectively arranged on the canopy front and rear end, and two rectangular long blocks are respectively arranged on the base front and rear end (Block specific section and location size refer to MT312-2000 standard).

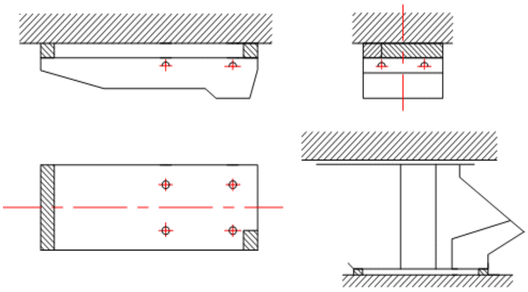


Fig. 3. Schematic diagram of the block position in working condition of the canopy hind torsion & base bending

To simulate the contact state between the block and component, the contact surfaces between them are solid modeled by face-to-face contact, and the friction coefficient is 0.2. The contact surface of the block, stop block and experimental bench is fully constrained to limit its three translational degrees of freedom.

3.4. Applying load

The hydraulic support is loaded inside the column, and the magnitude of load is determined according to the working resistance of the column. The working resistance of a single column is P=1600 kN, and 1.2P is used as the calculation load [8]. The center point of the column cross section is taken as the control point, and the structural distribution coupling constraints is applied to the column socket nodes and control point, and the motion of each node on the column socket is weighted by the constraint. So that the resultant force and resultant moment applied to the column socket nodes are equivalent to the force and moment applied to the control point. The interaction between the column, canopy and base as shown in Fig. 4.

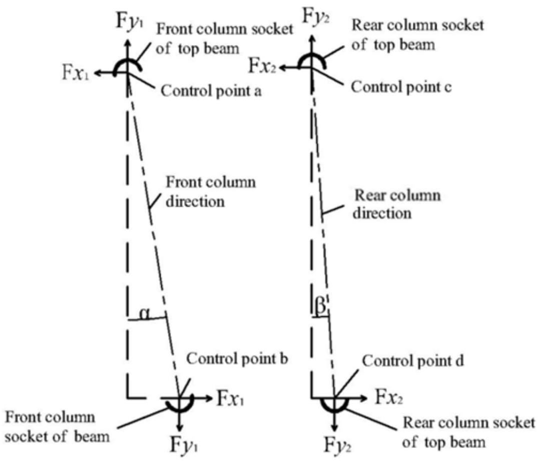


Fig. 4. Force loading schematic diagram

Inclination Angle of the front column

$$\alpha = \tan^{-1}\frac{x_b - x_a}{y_a - y_b} \quad (1)$$

Where

x_a、y_a—— x and y axis coordinates of control point a, mm;

x_b、y_b—— x and y axis coordinates of control point b, mm。

Inclination Angle of the rear column

$$\beta = \tan^{-1}\frac{x_d - x_c}{y_c - y_d} \qquad (2)$$

Where

x_c、y_c—— x and y axis coordinates of control point c, mm;

x_d、y_d—— x and y axis coordinates of control point d, mm。

Horizontal force at the control point of the front column socket:

$$Fx_1 = 1.2P\sin\alpha \qquad (3)$$

Vertical force at the control point of the front column socket:

$$Fy_1 = 1.2P\cos\alpha \qquad (4)$$

Horizontal force at the control point of the rear column socket:

$$Fx_2 = 1.2P\sin\beta \qquad (5)$$

Vertical force at the control point of the rear column socket:

$$Fy_2 = 1.2P\cos\beta \qquad (6)$$

Under the combined conditions of the canopy bending & base horizontal loading, the canopy hind torsion & base bending, according to the space geometric relationship obtain: α=10.9°, β=5.32°, Fx_1=363.06 kN, Fy_1=1885.44 kN, Fx_2=178.56 kN, Fy_2=1911.73 kN, applying each component force to the control point and completing the internal loading.

3.5. Meshing

Meshing is a crucial step for numerical simulation analysis. The quality of mesh directly affects the accuracy of the analysis result [9]. The basic principle of mesh discretization is to reflect the true stress state of the structure. The solid models of the hydraulic support components belong to box-shaped structure, which are welded by plates. In order to simulate the stress distribution of the model accurately, the high-order C3D10M element is used to mesh, the mid-node is retained, and the mesh element size is 27 mm. Grid encryption for the column socket, small hole, high stress area and interest area to improve the calculation and analysis accuracy. The number of grid elements is 779663, and the number of nodes is 1372425 in the integral model. The finite element model as shown in Fig. 5.

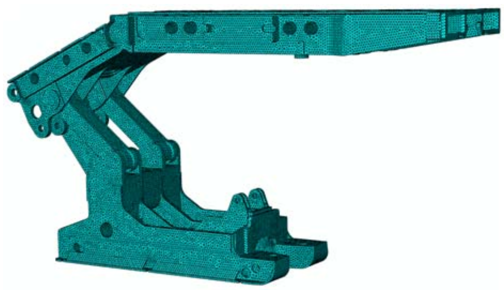


Fig. 5. finite element model

4. Result Analysis

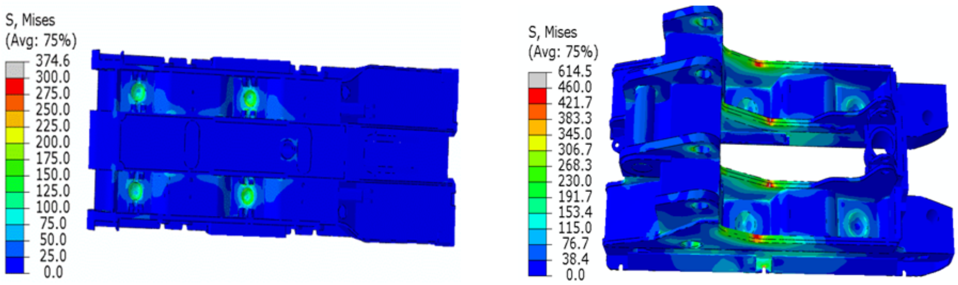
4.1. Canopy bending & base horizontal loading

The stress distribution of the canopy under the canopy bending & base horizontal loading is shown in Fig. 6(a). The bending deformation of the canopy is small, and the maximum stress distributed in the contact position between the canopy and block, so little attention is paid to it. The stress distribution is

IOP Conf. Series: Materials Science and Engineering **490** (2019) 052006 doi:10.1088/1757-899X/490/5/052006

uniform at the column socket, and the stress value is far less than the yield limit of 460 MPa, which will not cause damage.

The stress distribution of the base under the canopy bending & base horizontal loading is shown in Fig. 6(b). The maximum stress value of the base is 614 MPa, which exceeds the yield limit of the material, and distributed in the semi-circular hole of the base edge. The high stress value distributed in the arc transition of the main rib, which is tensile stress and close to the yield limit of the material, and easy to cause structural damage. In order to reduce the stress concentration, the semi-circular hole at the base edge should be strengthened, the radius of the transition arc should be increased, and the height of the main rib should be increased.



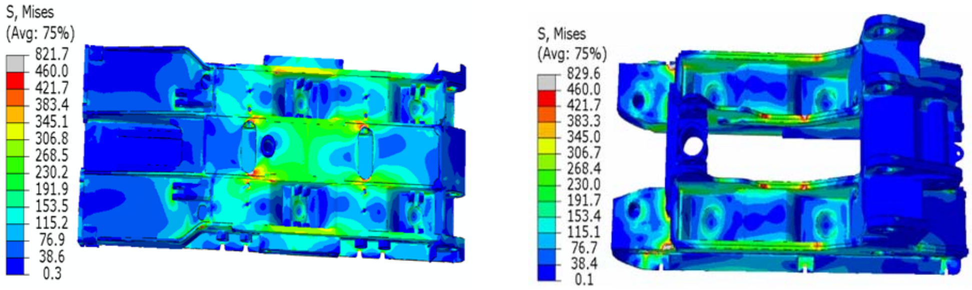
(a) The stress distribution of the canopy **(b)** The stress distribution of the base

Fig. 6. The stress distribution of the canopy and base under the canopy bending & base horizontal loading

4.2. The canopy hind torsion & base bending

The stress distribution of the canopy hind torsion & base bending is shown in Fig. 7(a). The high stress area is concentrated around the hole in the cover plate, the stress value is greater than the material yield limit, and the local reinforcement should be performed here. The stress of the side panel middle is close to the material yield limit. In order to increase the strength of the side panel, and avoid stress concentration, the box in the side panel should be widened.

The stress distribution of the base under the canopy hind torsion & base bending is shown in Fig. 7(b). The maximum stress value of 829 MPa, which is higher than the material yield limit, and located at the contact position between the bridge and front cover plate. The stress values of the semi-circular hole at the base edge, the arc transition in the main rib, the hole in the base side panel and bridge outside is also high. These areas are dangerous, and the local reinforcement need to be implemented to reduce the stress value.



(a) The stress distribution of the canopy **(b)** The stress distribution of the base

Fig. 7. The stress distribution of the canopy and the base under the canopy hind torsion & base bending

5. Conclusion

According to ZF6400/19/32 hydraulic support simulation analysis under two kinds of severe combined working conditions, the obtained results are summarized as follows:

(1)The area where the canopy failure may occur is around the hole in the cover plate. In order to effectively reduce the stress and meet the strength requirement, a flitch plat with the thickness of 10~15 mm is added at this location.

(2)The dangerous area of the base is located at the contact area between the bridge and cover plate, the transition arc of the middle rib, the semi-circular hole at the base plate and the hole in the front end of the side panel. In order to improve the safety factor of the base, it is necessary to add flitch plat, increase the transition arc radius and select high-strength materials in these high stress areas.

(3) Based on the simulation results, the original design scheme of the hydraulic support is improved. The simulation result shows that the improved design scheme can meet the strength design requirement of the hydraulic support under combined working conditions.

References

[1] Xia Qiuzhong. Finite Element Analysis and Mechanics Performance of Hydraulic Excavator’S Key Part[D]. Liaoning Technical University, 2007.
[2] Huang Hejiang. Research and application of rapid installation technology for high rise fully mechanized coal mining face[J]. Energy Technology and Management, 2012, 3(03): 119-142.
[3] Hu Min, Cui Jianghong, Cao Bide. Finite Element Analysis of ZF7200/18/35 Hydraulic Support[J]. Coal Mine Machinery, 2009, 30(05): 87-89.
[4] Yang Pengfei, Yang Guigui, Peng Yangxi, et al. Design reliability analysis and application of high concentration cementation stowing hydraulic support for coal mine[J]. coal engineering, 2016, 48(01): 24-27.
[5] Yan Honghong, Li Xiao, Li Yongtang. FEA of large mining height hydraulic support[J]. Mining & Processing Equipment, 2012, 40(11):17-20.
[6] Yan Zhiqin, Liu Yanping. Analysis of Caving Coal Hydraulic Support Finite Element Simulation Based on ANSYS[J]. Coal Technology, 2016, 35(8): 265-266.
[7] Chen Leiping. Finite element analysis of whole hydraulic support[D]. Xi'an University of Technology, 2007.
[8] Qin Runze.The Stress Testing And Finite Element Analysis to the Caving Hydraulic Support[D]. Xi`an University of Science and Technology, 2012.
[9] He Wenbin, Yao Hengyang, Ma Jun, et al. Strength Finite Element Analysis of Full Frame of Transition Hydraulic Support[J]. Coal Technology, 2017, 36(10): 223-225