Finite Element Modeling and Structural Parameter Optimization of Ball Hinge Rubber for Commercial Vehicle Thrust Rod

Wang Shuai[1,a], Luo Kai[2,b], Shen Guowei[3,c], Gao Sasa[1,d,*]

[1]College of Mechanical & Electrical Engineering, Shaanxi University of Science and Technology, Xi'an, 710021, China

[2]Technical Quality Department, Xi'an Oude Rubber & Plastic Technology Co, Ltd, Xi'an, 710201, China

[3]Technology Center, Xi'an Deshi Vehicle Components Co, Ltd, Xi'an, 710000, China

[a]1916215085@qq.com, [b]346738442@qq.com, [c]313913219shen@163.com, [d]gaosasa@sust.edu.cn

[*]Corresponding author

Keywords: Rubber ball joint; Constitutive model; Finite element analysis; Parameter optimization; Crack propagation

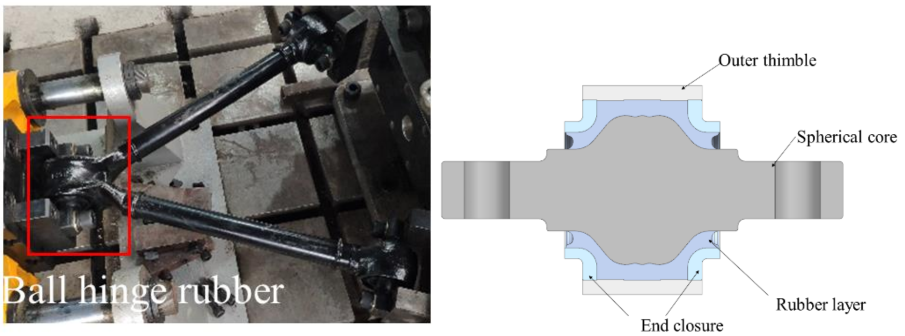
Abstract: When the thrust rod of commercial vehicles is under the ultimate load, rubber extrusion or even cracks may occur in the rubber layer, which affects the operation safety of the vehicle. In order to reduce the failure probability of the thrust rod and extend its service life, this paper simulated the working condition of the thrust rod ball-hinged rubber by finite element method, revealed the relationship between the maximum stress of the thrust rod ball-hinged rubber and its structure, and proposed an optimization scheme. Firstly, based on the stress-strain data of rubber material, the parameters of elastoelastic constitutive model of rubber material were fitted in ABAQUS software. Secondly, the finite element model of ball hinge rubber was established to reduce the maximum working stress of rubber layer by changing the parameters of rubber layer thickness and its edge shape, so as to extend the service life of thrust rod. The results show that reducing the thickness of the rubber layer can effectively reduce the maximum stress of the rubber layer, but the stress concentration is not significantly improved, and the crack spreads to the inner surface of the rubber layer, and changing the edge shape of the rubber layer to convex, not only can greatly reduce the maximum stress, but also effectively alleviate the stress concentration, but the rubber layer edge of the outer surface of the tear. Finally, the optimal combination scheme of the two structural parameters was determined by orthogonal test, which greatly reduced the maximum stress and improved the stress concentration, reduced the crack propagation length, and no tearing occurred on the inner and outer surfaces of the rubber layer. The effectiveness of the orthogonal experiment optimization scheme is proved, which can lay a theoretical foundation for the optimization design of thrust rod ball hinge rubber.

1. Introduction

As the core component of the balance suspension of commercial vehicle, the ball hinge structure of the thrust rod is made of rubber with superelastic characteristics. This material makes the thrust rod play an important role in the functions of transferring force, limiting displacement and isolating vibration, and has a greater impact on the handling stability of the vehicle. Considering that heavy commercial vehicles need to deal with a variety of complex driving road conditions and harsh use environment, the performance and stability of the thrust rod is particularly important, which is a key factor to ensure that the vehicle can run safely and smoothly under various conditions. As a major force transmission and vibration isolation component, the main failure forms of the thrust rod are rubber extrusion, cracking, etc., and there are relatively few problems such as ball core fracture and wear, which bring great losses to users, manufacturers and vehicle manufacturers [1-3]. Therefore, more and more people begin to pay attention to the safety performance of the thrust rod. At present, experts have carried out more research on the thrust rod assembly. Feng Guoyu [4], Wang Qian [5], Li Zan [6] and Li Lei [7] proposed methods to improve the material and structure of the ball core, the process flow and the jacket assembly through finite element analysis, so as to extend the service life of the thrust rod assembly. Ke Jun et al. combined finite element method and genetic algorithm to propose a multi-objective optimization method, which extended the safe working time of the thrust rod by 7 times [8]. Shi Wenku et al. carried out dynamic simulation of V-shaped thrust rod under the ultimate load through Adams software, obtained the ultimate working load of thrust rod, and proposed optimization schemes such as grooving on the ball core [9]. Cao Zhou et al. took the thrust rod in automobile rear suspension as the research object [10]. Aiming at the problems of rubber falling off and cracking, Cao et al. improved the phenomenon of rubber falling off by modifying the structure and material of the end cover [10]. Yang Shuhan took locust hind leg and foot pad as the bionics research object, optimized the working face of thrust rod ball hinge, and made four parameters, namely radius, height, spacing and number of groups of protrusion [11].

2. Thrust Rod Ball Hinge Rubber Model

As a key component in the suspension system of commercial vehicles, the thrust rod is mainly divided into type I and type V. The thrust rod mainly includes ball-hinged rubber, circlip and rigid jacket components [12]. The core research point of this paper is to analyze the ball hinge rubber in the thrust rod. The ball hinge rubber mainly consists of the ball core, the end cap, the rubber layer and the outer sleeve. Figure 1 shows the installation position of the thrust rod ball hinge rubber and the structure diagram of the ball hinge rubber.



(a) (b)

Figure 1: Schematic diagram of thrust rod ball joint rubber a) Actual installation position of ball hinge rubber, b) Schematic diagram of ball hinge rubber model.

3. Thrust Rod Ball Hinge Rubber Finite Element Model

3.1. Superelastic Constitutive Model of Rubber Material

Ball-hinged rubber is an important part of thrust rod assembly, and the rubber layer is an important part of bearing multi-directional loads. In order to accurately characterize the stiffness of thrust rod ball-hinged rubber, an accurate constitutive model should be established. In order to accurately describe the mechanical properties of rubber materials in the process of tension and compression, a constitutive model based on strain energy density function was used for further study and analysis. In this study, the stress-strain parameters of rubber materials were measured by tensile test, and the data were fitted by ABAQUS software to obtain the corresponding hyperelastic constitutive parameters. In the MTS series 1 kN electronic universal test machine, according to the provisions of GB/T 528-2009, the uniaxial tensile test method is used to test the rubber sample. Figure 2 shows the uniaxial tensile test equipment. The rubber specimen should be prepared as a type I dumbbell specimen with a thickness of 2.0 mm $\pm$0.2 mm and a total length of 115 mm, in which the length of the test area of the specimen is 25.0 $\pm$0.5 mm. In order to ensure the accuracy of the test, the sample should be flat without burrs and other defects.

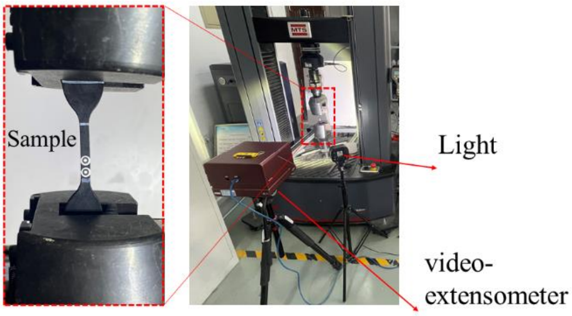


Figure 2: Uniaxial tensile test equipment.

Figure 3: Test piece that fractured after the test.

In order to reduce the error of the test, each group of tests is carried out three times to ensure the reliability of the results. In the process of installing the sample, it is necessary to symmetrically clamp the sample on the upper and lower holder of the universal testing machine, so that the tension is evenly distributed on the rubber cross section, and then paste mark points in the test area of the rubber sample in order to measure the deformation length of the sample and record the length change of the sample. According to the regulations of GB/T528-2009, the moving speed of the upper gripper was set to 500 mm/min±50 mm/min, and the sample was pulled off after the test, as shown in Figure 3.

The data obtained from the test are processed and the force and displacement data are converted into stress-strain curves. The stress-strain data were imported into the material module of ABAQUS software for fitting, and the least square method was used to fit the data. Four strain energy functions, Yeoh, Mooney, Neo-Hookean and third-order Odgen, were used to calculate the constitutive model parameters of rubber materials.

As can be seen from Figure 4, the third-order Odgen model has a high agreement with the test data. Therefore, the third-order Odgen strain energy function is adopted in the constitutive model of ball-hinged rubber [13]. Odgen model is a phenomenological constitutive model based on continuum mechanics theory, and its strain energy density function is expressed as follows:

$$W=\sum_{i=1}^{N}\frac{2\mu_i}{\alpha_i^2}(\overline{\lambda}_1^{\alpha_i}+\overline{\lambda}_2^{\alpha_i}+\overline{\lambda}_3^{\alpha_i}-3)+\sum_{i=1}^{N}\frac{1}{D_i}(J-1)^{2i} \tag{1}$$

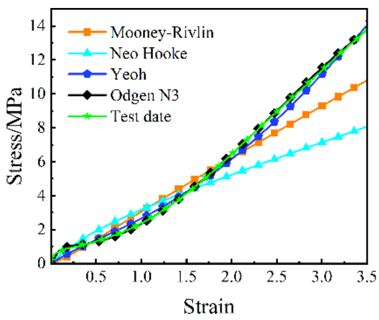
Where W is the strain energy per unit volume; μ_i, α_i and D_i are the material parameters of rubber, which are fitted by ABAQUS software, as shown in Table 1. λ_i is the main elongation ratio of rubber; J is the elastic volume ratio.

The finite element model of dumbbell rubber sample was established by ABAQUS software, and the mesh was divided by hexahedral reduced integral hybridization unit (C3D8RH). The rubber constitutive model parameters fitted were input into the material model, and the general static analysis step was set, because during the tensile test, the sample was pulled apart after 28 s at the set tensile speed. The analysis step time was set to 28 s, and a fixed load was applied to the left end of the rubber sample, and a velocity load was applied to the right end, with the size of 500 mm/min. The finite element model is shown in Figure 5, and the finite element model is used for tensile test simulation analysis.

As can be seen from Figure 6, the stress-strain curve obtained by finite element simulation has good coincidence with the curve obtained by experiment, and the third-order Odgen hyperelastic constitutive model can be used to accurately describe the hyperelastic characteristics of rubber parts [14].

Table 1 Third Odgen model parameters.

Constitutive Model	μ_n	α_n	D_n
Third Odgen	μ_1=-0.7953	α_1=-2.1836	D_1=0
	μ_2=-3.7935	α_2=4.2577	D_2=0
	μ_3=9.3347	α_3=-8.2814	D_3=0



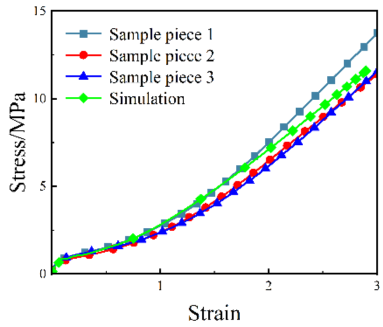


Figure 4: Stress-strain curves of rubber materials under different constitutive models. (Left)

Figure 5: Dumbbell shaped rubber finite element model. (Middle)

Figure 6: Experimental and simulation results of stress-strain curves of specimens. (Right)

3.2. Ball Hinge Rubber Finite Element Model

The 3D model of ball hinge rubber was imported into HYPERMESH software and meshed. In order to improve the calculation accuracy, a hexahedral grid was adopted, as shown in Figure 7, where the rubber layer grid element type was C3D8RH and the rest grid element type was C3D8R.

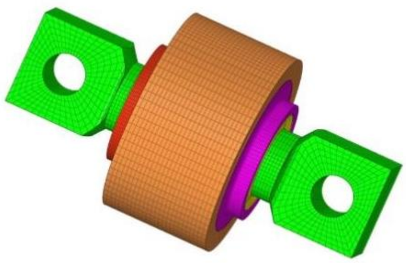


Figure 7: Spherical joint rubber mesh mode.

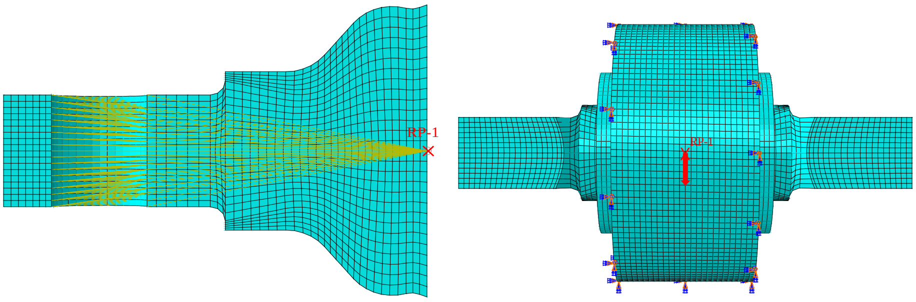
The mesh model was imported into ABAQUS software, material properties were assigned to each part, rubber material parameters were defined according to Table 1, and the remaining rigid parts were defined according to the material parameters described in Table 2.

Table 2: Linear material parameters.

Name of parts	Material type	Material parameter	Numerical value
End closure Outer thimble	45	Density /(kg/mm^3)	7.85E-6
		Young modulus /(N/mm^2)	2.1E5
		Poisson's ratio	0.3
Spherical core	40Cr	Density /(kg/mm^3)	7.85E-6
		Young modulus /(N/mm^2)	2.06E5
		Poisson's ratio	0.3

3.3. Boundary Conditions

The thrust rod ball-hinge rubber has a complex structure, including nonlinear material rubber and linear material steel, etc. In order to accurately predict the stiffness of the thrust rod, it is necessary to simulate the working condition of the thrust rod under the ultimate load. In this paper, the ultimate load is used to simulate the ball hinge rubber, and the solution method is dynamic implicit analysis.



(a) (b)

Figure 8: Boundary condition setting a) Handle hole coupling Settings, b) Boundary condition setting.

(1) Define displacement boundary conditions

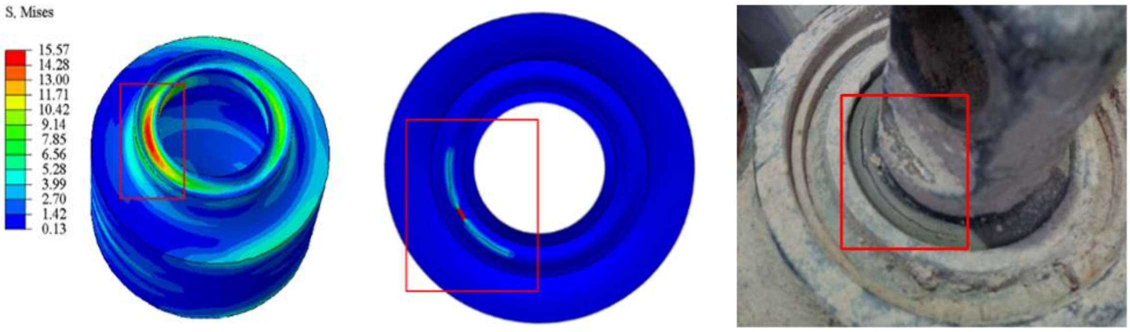
During the operation of the thrust rod, there is friction contact between the rubber and the end cap and the rigid jacket, and the friction coefficient is 0.001. For ball-hinge structures, because the rubber is vulcanized with the end cap and the ball core, the contact relationship between them is defined as binding.

(2) Define the load boundary conditions

In order to simulate the deformation process of the ball hinge under the extreme load, all the nodes in the handle holes on both sides of the ball core are coupled to the center of the ball core. The coupling mode is shown in Figure 8(a). A radial load is applied to the coupling point of the handle of the ball core, with the size of 150 kN. Then a fixed constraint is applied to the outer side of the external sleeve, as shown in Figure 8(b).

3.4. Finite Element Model Verification

For ball-hinge rubber, the most common failure forms are rubber extrusion and cracking. The stress state of the rubber layer obtained by finite element simulation is shown in Figure 9(a). It can be seen from the figure that there is a phenomenon of stress concentration at the circular arc of the rubber layer, and the maximum stress reaches 15.57 MPa, which is caused by the rubber being extruded and leaking out. Cracks are easy to occur here. The prefabricated crack in the stress concentration area is 5 mm in length and 2 mm in depth, which is shown as the red part in Figure 9 (b), and the crack propagation is simulated according to the setting mode mentioned above. Figure 9 (b) shows the finite element simulation result of crack propagation, the red area is the prefabricated crack, and the green area is the crack propagation trend. The crack propagation length is 6.5 cm. In actual work, the crack location of the rubber layer of the ball hinge is shown in Figure 9(c), which is also located at the circular arc of the rubber layer, which is consistent with the finite element simulation results, verifying the accuracy of the finite element model.



(a) (b) (c)

Figure 9: Verification of accuracy of finite element model a) Simulated stress cloud map, b) Finite element simulation, c) Actual failure mode.

4. Parameter Optimization of Ball Hinge Rubber

In view of the phenomenon of cracks in the rubber layer, two improvement measures are proposed: reducing the thickness of the rubber layer (the diameter of the outer sleeve and the end cap will change with it) and changing the edge shape of the rubber layer to study the relationship between the maximum stress and the structure of the ball hinge rubber under working conditions, so as to avoid the stress concentration of the ball hinge rubber layer and reduce the length of crack propagation.

4.1. Influence of Rubber Layer Thickness of Ball Hinge

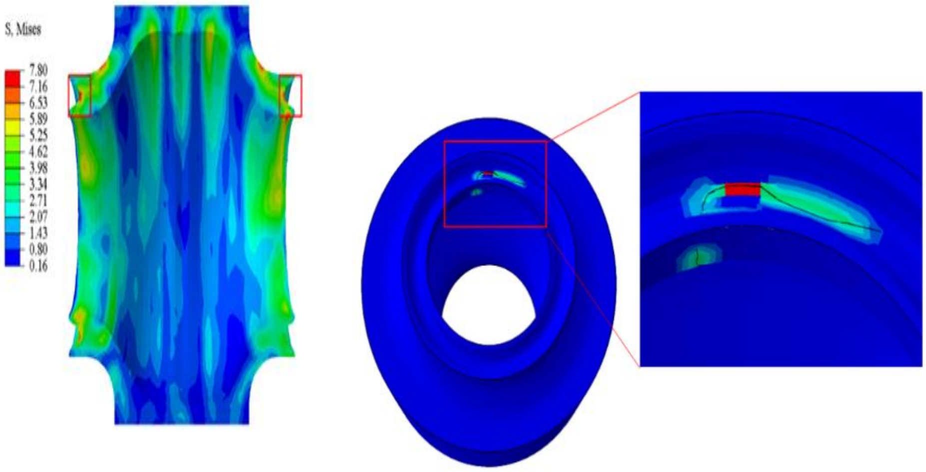
Considering the assembly problem, reducing the thickness of the rubber layer requires changing the inner diameter of the outer sleeve and the diameter of the end cap. The working condition of the thrust rod was predicted by finite element simulation, and the maximum stress value of the ball hinge rubber was obtained, as shown in Table 3.

The 5 subschemes in Table 3 show that the maximum stress value of the rubber layer has a certain relationship with the reduction of the radial dimension of the rubber layer. Appropriately reducing the thickness of the rubber layer can improve the stress state of the rubber layer and increase its life. When the diameter is reduced by 4 mm, the maximum working stress is 7.80 MPa, and the reduction rate is 49.90%. At this time, continuing to reduce the diameter will not reduce its maximum stress, but because the rubber layer is too thin, the stress increases at the contact with the end cap and the outer sleeve, and it is easier to fail. Therefore, the optimal reduction of the thickness of the rubber layer is 4 mm.

Table 3: The variation of max working stress with the decrease of rubber layer thickness.

Decrease /(mm)	Maximum working stress /(MPa)	Amount of variation /(%)
0	15.57	0
1	14.03	-9.89
2	11.71	-24.79
3	8.76	-43.74
4	7.80	-49.90
5	8.3	-46.69

Figure 10(a) shows the stress cloud diagram of the rubber layer under working load after the thickness of the rubber layer is reduced by 4 mm. It can be seen that the maximum stress of the rubber layer has decreased compared with the original structure, but the stress concentration still exists and cracks are easy to occur. Figure 10(b) shows the finite element simulation results of crack propagation when the thickness of the rubber layer decreases by 4mm. It can be seen that although the length of crack propagation becomes shorter, only 3cm, the crack extends to the inside of the rubber layer, which makes the ball hinge rubber more likely to fail.



(a) (b)

Figure 10: Rubber layer outer diameter reduced by 4 mm a) Simulated stress cloud map, b) Crack propagation finite element simulation results.

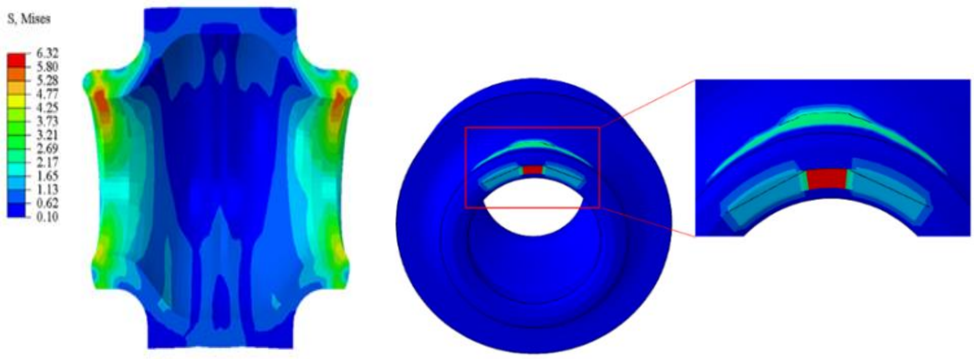
4.2. Influence of Rubber Layer Edge Shape

On the basis of the original model, the initial thickness of the rubber layer was adopted, and the edge of the rubber layer was changed from concave to convex. In order to ensure that the rubber thickness at the contact point between the rubber layer and the ball core remained unchanged, the convex radius was set to 4.25mm, 4.5mm, 4.75mm and 5mm respectively, and the maximum stress value of the ball hinge rubber of the thrust rod was obtained, as shown in Table 4. As can be seen from Table 4, changing the edge of the rubber layer to convex can reduce the maximum stress of the rubber layer, but with the change of the radius of the edge convex, its maximum stress will also change, because the shape of the rubber edge is changed to convex back, and the rubber has a larger buffer area to absorb the load when it is squeezed. Under the premise of ensuring that the remaining rigid elements do not change, the modification of the rubber edge convex radius will appear a value that makes the stress minimum. When the convex radius is 4.75mm, the maximum working stress is 6.32MPa and the reduction is 59.41%.

Table 4: The variation of maximum working stress with increasing outer convex radius of rubber layer edge.

Convex radius /(mm)	Maximum working stress /(MPa)	Amount of variation/(%)
Primary structure	15.57	0
4.25	6.83	-56.13
4.5	6.61	-57.55
4.75	6.32	-59.41
5	7.19	-53.82

Therefore, when the convex radius is set to 4.75mm, the mechanical properties of the rubber layer are the best. Figure 11(a) shows the stress cloud map when the outer convex radius of the rubber layer edge is 4.75mm. It can be seen that the stress concentration has improved and the maximum stress has decreased significantly. Figure 11(b) shows the finite element simulation results of crack propagation when the convex radius of the edge is 4.75mm. At this time, the crack propagation length is only 4cm, but the outer surface of the edge of the rubber layer is torn, which is not conducive to the safe work of the ball hinge rubber.



(a) (b)

Figure 11: Edge convex radius is 4.75 mm a) Simulated stress cloud map, b) Crack propagation finite element simulation results.

4.3. The Relationship between the Maximum Stress of Ball Hinge Rubber and Structure

On the basis of the above single structure parameter test optimization, with the thickness reduction of the rubber layer and the radius of the edge convex as the investigation factors and the maximum stress of the rubber layer as the investigation index, four levels with better optimization effect were selected from each of the two levels, and a two-factor four-level orthogonal test was designed to optimize the maximum working stress of the rubber layer. Orthogonal test factors and levels are shown in Table 5.

Table 5: Orthogonal test factors and levels.

Level	A Thickness reduction of rubber layer	B Edge convex radius
1	3 mm	4.25 mm
2	4 mm	4.5 mm
3	5 mm	4.75 mm
4	6 mm	5 mm

Table 6: Orthogonal experiment scheme and results.

Test number	A	B	Maximum stress of rubber layer /MPa
1	1	1	4.874
2	1	2	4.96
3	1	3	4.866
4	1	4	4.886
5	2	1	5.303
6	2	2	5.368
7	2	3	4.870
8	2	4	5.198
9	3	1	5.274
10	3	2	5.339
11	3	3	4.940
12	3	4	5.162
13	4	1	5.029
14	4	2	5.150
15	4	3	5.034
16	4	4	5.129

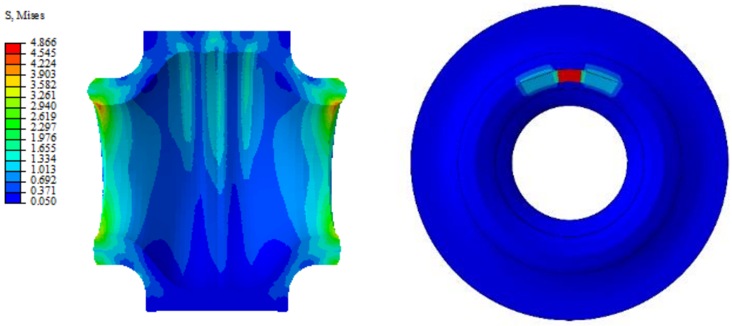
Under the principle of neat comparability and uniform dispersion, L16(42) orthogonal test table [15] was selected according to factors and levels, and the orthogonal test scheme and results were shown in Table 6.

Range analysis was carried out based on the orthogonal test results. According to the calculated range value R, the significant degree of influence of the thickness reduction of the rubber layer and the convex radius of the edge on the maximum stress of the rubber layer was evaluated, and the results were shown in Table 7.

Table 7: Range analysis table.

Range analysis	A	B	Significance
K1	19.586	20.48	
K2	20.739	20.817	
K3	20.715	19.71	
K4	20.342	20.375	
k1	6.529	6.827	
k2	6.913	6.939	
k3	6.905	6.570	
k4	6.781	6.792	
R	0.384	0.369	A>B

Range analysis can evaluate the influence degree of each level on the investigation index. The higher the range value R, the greater the influence of the factor on the investigation index. It can be seen from Table 7 that the thickness reduction of rubber layer has a greater impact on the maximum stress of rubber layer. Due to the smaller the maximum stress of the rubber layer, the longer the service life and better performance of the ball joint rubber, so we should make the k1 and k2 indicators as small as possible.



(a) (b)

Figure 12: Stress nephogram at optimal combination a) Simulated stress cloud map, b) Crack propagation finite element simulation results.

Therefore, the optimal combination level obtained in this test is A1B3, that is, the thickness of the rubber layer is reduced by 3mm, and the edge shape is the convex radius of 4.75mm. At this time, the maximum stress of the rubber layer is only 4.866 MPa, which is reduced by 68.75% compared with the original structure. Figure 12(a) is the stress cloud map of the optimal combination, and it can be seen that there is no obvious stress concentration area in the rubber layer, and the maximum stress is greatly reduced. Figure 12(b) shows the finite element simulation result of crack propagation with the optimal result after orthogonal test. At this time, the length of crack propagation is only 2 cm, and the crack has no tendency to expand to the inner and outer surfaces. It is proved that the optimization scheme can reduce the length of crack propagation and the stress of rubber layer, so as to extend the life of rubber of thrust rod ball hinge.

5. Conclusions

In this paper, a finite element model of ball-hinged rubber for a heavy commercial vehicle thrust rod was constructed, and the optimization effect of the thickness of the rubber layer and the edge shape of the rubber layer on the ball-hinged rubber was analyzed and studied. The main conclusions were drawn as follows:

(1) The super elastic constitutive model of rubber material was determined by uniaxial tensile test. The finite element model of ball-hinged rubber was established by adopting third-order odgen strain energy function for the constitutive model of rubber layer material. The maximum stress distribution position and crack propagation path of rubber layer were obtained through simulation, which were the same as the actual crack generation position.

(2) Through finite element simulation, it can be seen that reducing the thickness of the rubber layer can effectively reduce the maximum stress of the rubber layer, but the stress concentration is not significantly improved. Although the length of crack propagation is shorter, the crack will expand to the inside of the rubber layer, which will make the ball hinge rubber more likely to fail. Changing the edge shape of the rubber layer to convex can not only reduce its maximum stress, but also effectively alleviate the stress concentration. Although the crack propagation length is only 4 cm, the outer surface of the rubber layer edge is torn, which is not conducive to the safe work of the ball hinge rubber.

(3) Through the orthogonal test, it is found that the rubber layer thickness reduction has a greater influence on the rubber layer stress, and the optimal combination is determined: the rubber layer thickness is reduced by 3 mm, the rubber edge shape is convex, and the radius is 4.75 mm. At this time, the rubber layer stress is the smallest, only 4.866 MPa, and the crack propagation length is reduced. There was no tendency to extend to the inner and outer surfaces of the rubber layer.

Acknowledgment

The authors appreciate greatly the financial support from the Key Research and Development Plan of Shaanxi Province, China (No. 2023-YBGY-346).

References

[1] HAO H, WANG H W, OUYANG M G. Predictions of China's passenger vehicle and commercial vehicle stocks [J]. Journal of Tsinghua University (Science and Technology), 2011, 51(6): 868-872.
[2] ZHANG H H, CHEN K S, ZHANG J. Static & dynamic analysis of v-type propelling rod in air suspension system [J]. Tractor & Farm Transporter, 2009, (3): 40-42.
[3] ZHANG J R, LI J L, DENG Y, et al. Strength design of thrust bar for 40 t heavy-duty vehicle equalizing suspension [J]. Automobile Technology, 2008, (3): 19-22.
[4] FENG G Y. Performance optimization and fatigue life prediction of commercial vehicle thrust rod [D]. Changchun: Jilin University, 2017.
[5] WANG Q. Structure optimization and durability research of heavy commercial vehicle thrust rod [D]. Changchun: Jilin University, 2013.
[6] LI Z. Structure improvement and performance study of thrust rod on commercial vehicle [D]. Changchun: Jilin University, 2015.
[7] LI L. Structure design and fatigue life study of rubber bushing for commercial vehicle thrust rod [D]. Changchun: Jilin University, 2017.
[8] KE J, ZU H F, SHI W K. A multi-objective optimization method for the spherical hinge of a thrust rod based on the finite element method and genetic algorithm [J]. Automotive Engineering, 2020, 42(02): 178-183.
[9] SHI W K, KE J, WANG Q, et al. Limit load analysis and spherical hinge structure optimization for v-type thrust rod [J]. Journal of Xi'An Jiao Tong University, 2013, 47(10): 132-136.
[10] CAO Z, WANG Y, LIANG J, et al. Failure analysis of v-type thrust bar of commercial vehicle [J]. Automobile Applied Technology, 2021, 46(14): 69-72.
[11] YANG S H. Bionic design and finite element analysis of the working surface of the ball hinge of the thrust rod of the automobile [D]. Changchun: Jilin University, 2023.
[12] HOU K, SHEN Y Q, QIAO X. Design and research on thrust rod of hollow sphere pin [J]. Automobile Applied Technology, 2020, 45(18): 66-67.
[13] Alexander E. Ehret. On a molecular statistical basis for Ogden's model of rubber elasticity [J]. Journal of the Mechanics and Physics of Solids, 2015, 78.
[14] FAN Z R, ZHANG J, HUO Z B, et al. Finite element simulation of static stiffness of annular nitrile butadiene [J]. Materials for Mechanical Engineering, 2024, 48(01): 93-98.
[15] LIU R J, ZHANG Y W, WEN C W, et al. Study on the design and analysis methods of orthogonal experiment [J]. Experimental Technology and Management, 2010, 27(09): 52-55.