Short communication

Finite element analysis of spherical bearing based on Abaqus

Abstract: Aiming at QZ12500GD spherical bearing, Abaqus finite element analysis software is adopted for modeling analysis, vertical pressure, horizontal thrust and vertical tension are analyzed, and the deformation and stress of the bearing are studied. The model parameters and modeling method of spherical bearing in finite element software analysis are obtained. The analysis results show that this kind of bearing form has good deformation and bearing capacity under the design bearing capacity and meets the design requirements.
Key words: spherical bearing; Stress; Strain; finite element analysis

Introduction

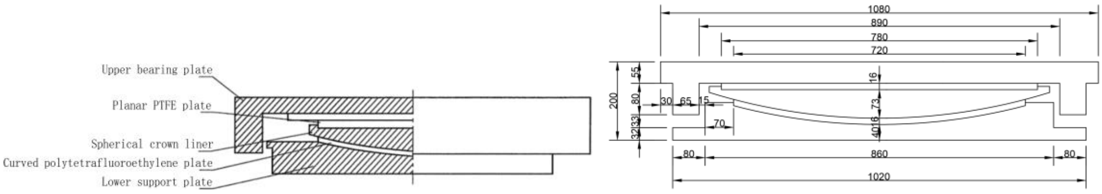
The painful lessons of previous earthquakes show that bridges are an important part of lifeline engineering, and once they are seriously damaged in earthquakes, they will cause huge economic and social losses. Since the Wenchuan earthquake in 2008, more and more attention has been paid to the earthquake resistance of bridges, and the earthquake resistance of bearings is an important part of the earthquake resistance of bridges.

The spherical bearing is one of the most widely used bearing forms at present. By setting spherical friction pairs, the spherical bearing makes the bearing rotate flexibly. After overcoming the sliding friction between the spherical cap steel plate and the spherical PTFE plate, the bearing rotates immediately, and the magnitude of its rotating torque has nothing to do with the rotation angle, which adapts to the multi-directional large-angle bridge [1~3], reduces the earthquake action on the structure, consumes seismic energy, makes the stress on the structure more reasonable, and protects the safety of the whole structure. It not only has the characteristics of large bearing capacity and displacement of basin rubber bearing, but also can better meet the needs of large corner of bearing, so it is widely used in highway bridges, municipal bridges and rail transit bridges [4].

In this paper, the large-scale finite element analysis software Abaqus is used for analysis, and a finite element model is established to test the mechanical performance of QZ12500GD seismic spherical bearing under the design load, and to see whether it meets the requirements of practical engineering use.

1 Basic structure of bearing

In this paper, QZ12500GD seismic spherical bearing is analyzed as shown in Figure 1.



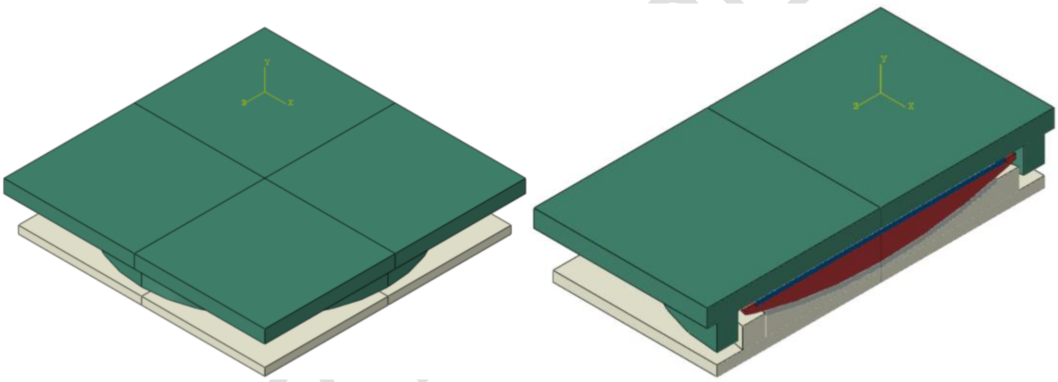
a Schematic diagram of fixed support structure b Bearing size (mm)

Figure 1 QZ12500GD Seismic Spherical Bearing

2 bearing finite element model

2.1 finite element model

Three-dimensional geometric models of upper and lower bearing plates, spherical cap liner plate and tetrafluoride plate are established in Abaqus finite element software and assembled. The dimensions of each component are established according to Figure 1b, and the assembled model is shown in Figure 2. The overall division unit types of bearings are basically C3D8R, eight-node linear hexahedral unit, reduced integral and hourglass control, with a total of 30,791 nodes and 17,214 units.



a Global model b Section model

Fig. 2 Finite Element Model of Seismic Spherical Bearing

2.2 Material parameters

The upper and lower bearing plates of the bearing are made of Q235, the spherical cap liner is made of titanium alloy, and the PTFE plate is made of polyethylene. See Table 1 for specific parameters of each material.

Table 1 The features of bearing material

Material	Elastic modulus /×10^9Pa	Poisson's ratio	Density /(kg • m^{-3})
Q235	200	0.3	7850
titanium alloy	96	0.36	4620
polyethylene	1.1	0.42	950

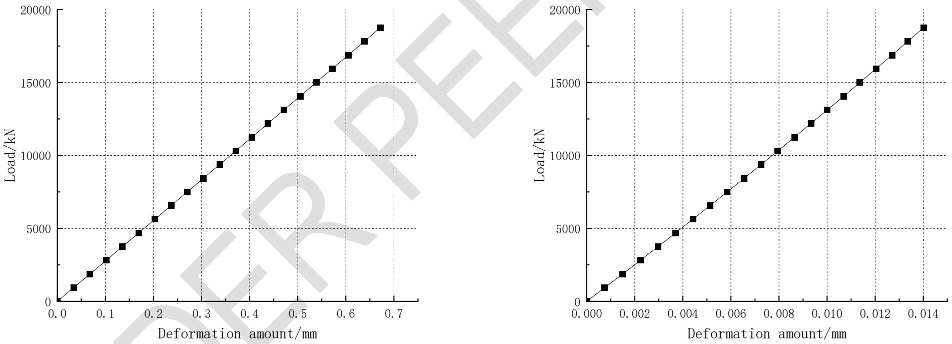
3 finite element analysis of bearing

In order to analyze and test the stress state of each component of the bearing and ensure the reliability of the bearing, the finite element analysis is carried out on the main load-bearing components of the bearing, focusing on the stress magnitude and distribution of its spherical lower seat plate, PTFE slide plate and spherical cap liner [5]. The main contents include: static analysis, bearing modal analysis and dynamic analysis.

3.1 Static analysis

3.1.1 Vertical pressure simulation

In order to simulate the test conditions of the bearing, a surface load is applied to the upper surface of the bearing, and the load is applied to 1.5 times the design bearing capacity (18750kN) step by step. The center point of the top surface of the upper bearing plate is selected for vertical displacement analysis, and the radial deformation of the bushing base of the lower bearing plate is analyzed, and the load-deformation curve can be obtained as shown in Figure 3. There is a linear relationship between load and deformation [6]. When the maximum load is 18750kN, the maximum vertical deformation is 0.67mm, and the compression deformation is not more than 1% of the total height of the bearing. The maximum radial deformation of the lower bearing plate bushing base is 0.014mm, which is not more than 0.5% of the radial length, and meets the design requirements.



a Load vertical deformation relation curve   b Load radial deformation relation curve

Fig. 3 Load-deformation relation curve

The result of extracting Abaqus partial stress is shown in Figure 4. As can be seen from the figure, under the action of 1.5 times the design load, the maximum Mises stress is 106.2MPa, and all materials are in elastic state, which also reflects the correctness of the linear change of load-deformation curve and meets the design requirements.

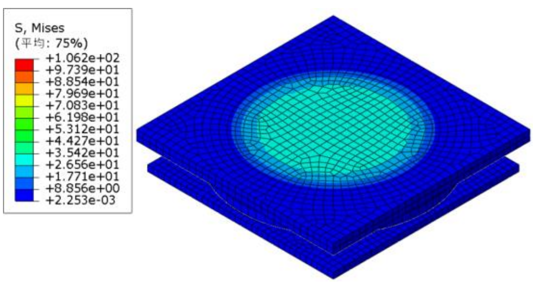
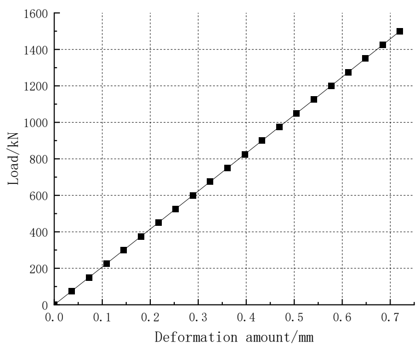


Fig. 4 Mises stress cloud

3.1.2 Horizontal Thrust Simulation

In order to simulate the actual stress of the test, it is necessary to apply a load of 12500kN on the upper surface of the bearing, and apply it to the side of the upper bearing plate step by step to 1.2 times the horizontal design bearing capacity (1500kN), and select the radial deformation of the spherical cap liner to analyze [7]. There is a linear relationship between load and deformation. When the maximum load is 1500kN, the maximum deformation in the horizontal direction is 0.72 mm. The results of extracting partial stress and displacement of Abaqus are shown in Figures 5 and 6.



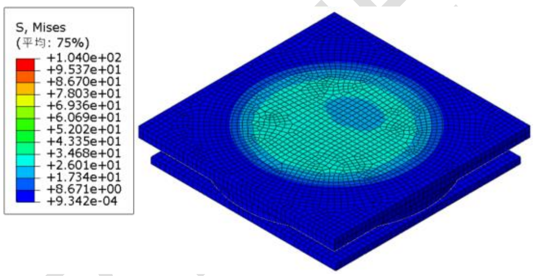


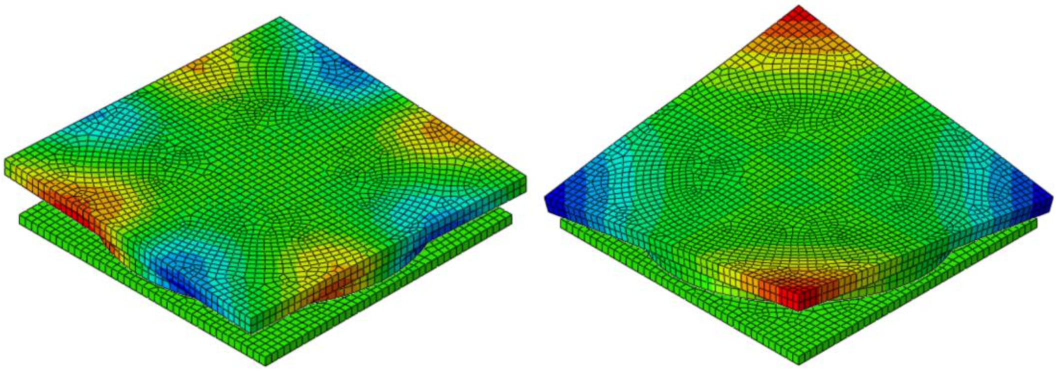
Fig. 5 Load radial deformation relation curve Fig. 6 Mises stress cloud

As can be seen from the figure, under the action of 1.2 times the design load, the maximum Mises stress is 104MPa, and all materials are in elastic state, which also reflects the correctness of the linear change of load-deformation curve and meets the design requirements.

3.2 Modal analysis

The purpose of modal analysis and calculation is to find out the natural frequency and main vibration mode of the structure. For an actual structure, there are many kinds of vibration frequencies and modes of each order. Because the high-order modes generally have little dynamic response to the structure, the lowest 3 to 5 modes are usually considered [8].

Give each component the material parameters of its design material selection, and then carry out modal analysis on the established model. We calculated and analyzed the first 20 modes of the model, among which the 4th and 5th modes are more useful. The first order frequency is 267.8Hz and its vibration direction is vertical; The second-order frequency is 276.1Hz, and its vibration direction is the up-and-down movement direction. This vibration can cause the bearing to rotate. See Figure 7 for the first-order and second-order modes. The mass damping of the structure with larger vibration frequency above the third order will weaken it.



a 1 mode diagram b 2 mode diagram

Fig. 7 Vibration Mode Diagram of Bearing

3.3 Dynamic analysis

In order to simulate the situation that the bearing encounters an earthquake in normal use, a surface load of 12500kN is applied to the upper surface of the bearing, and seismic waves are input at the bottom of the bearing (see Figure 8) for analysis, and the distribution law of stress and strain of the bearing in the seismic load is obtained [9].

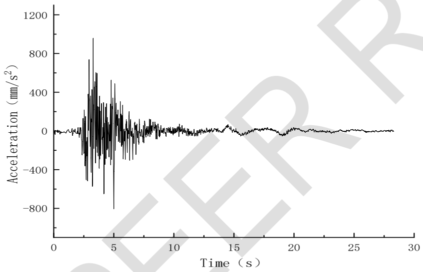
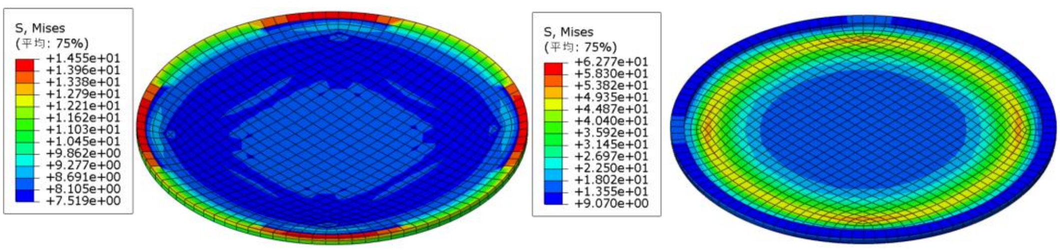


Fig. 8 Selected seismic waves

To analyze and study the dynamic response of the structure under the earthquake load, we mainly care about the strain of the main components of the bearing changing with time on the applied load spectrum [10]. Among them, the curved PTFE plate and the spherical cap bushing are the main parts of the bearing, and we analyze the stress and strain as follows.

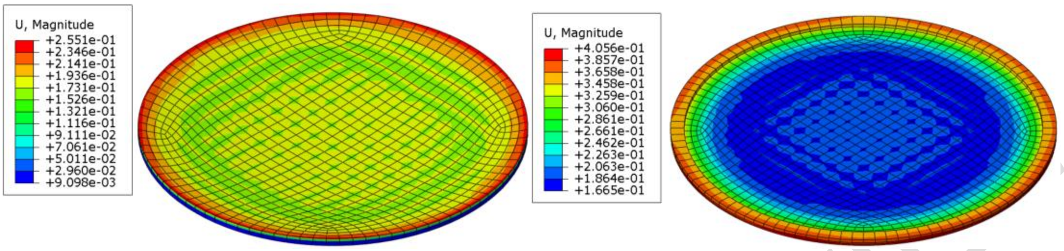
Fig. 9 is the stress nephogram of the structural model at the peak. It can be seen from the figure that the stress distribution of the curved PTFE plate is uniform in the central part of the model, and its compressive stress distribution is between 7.5 MPa and 9.2 MPa; The maximum local stress at the edge is 14.5MPa, and the stress is less than the fiber stress of PTFE 45MPa, which meets the design requirements. The stress distribution in the center of spherical cap bushing is between 9 MPa and 22 MPa; The local maximum stress is about 62.8MPa, and the stress is less than 375MPa, which meets the design requirements.



a Stress nephogram of curved PTFE plate b Stress nephogram of spherical cap bushing

Fig. 9 Model stress nephogram at the peak of earthquake acceleration

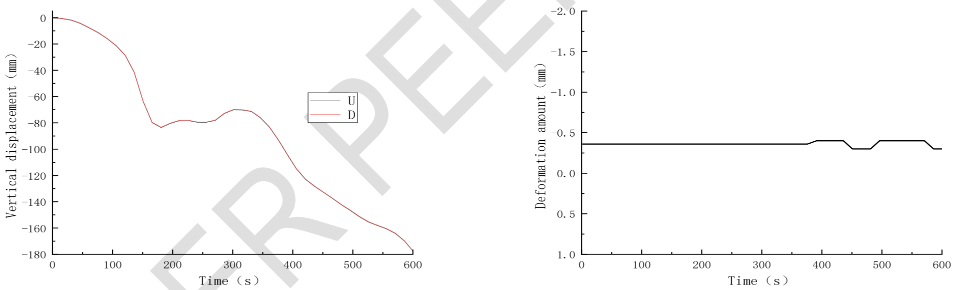
Fig. 10 is the strain nephogram of the structural model at the peak. It can be seen from the figure that the strain of the curved PTFE plate changes uniformly in the horizontal direction, and the maximum vertical compressive strain is generated at the center of the structure, with the maximum compressive strain of 0.17mm and the maximum circumferential strain of the model of 0.26mm, and the deformation meets the design requirements. The maximum vertical compressive strain of spherical cap bushing is 0.23mm, and the maximum circumferential strain is 0.41mm, and the deformation meets the design requirements.



a Strain nephogram of curved PTFE plate   b Strain nephogram of spherical cap bushing

Fig. 10 Model strain nephogram at the peak of earthquake acceleration

Select two points in the center of the upper and lower surfaces of the bearing, study the variation law of the strain of the bearing with time under the dynamic load (0~600s) at the peak section and draw its strain-time curve as shown in Figure 11. Curve d in the figure is the vertical displacement-time curve of the center point of the lower surface of the support; U curve is the vertical displacement-time curve of the center point of the upper surface of the bearing [11].



a DU curve displacement diagram   b DU curve difference diagram

Fig. 11 diagram of strain versus time

The figure shows the vertical strain of the upper and lower surfaces of the bearing changing with time under dynamic load. It can be seen from the figure that due to the unequal strain of the upper and lower surfaces of the bearing, the bearing produces compression, and the maximum compression is 0.4mm, which is 0.2% of the thickness of the bearing, and the compression deformation meets the design requirements [12].

4 Conclusion

In this paper, QZ12500GD seismic spherical bearing is analyzed. The static calculation, bearing modal analysis and dynamic analysis are carried out by Abaqus large-scale finite element analysis software, and the deformation and stress of the bearing are studied. It can be seen that the stress and strain of the main components meet the design requirements and the stress distribution of the main bearing components is uniform according to the relevant industry standards, which ensures the safety and reliability of the bearing.

References

[1] Shen Yinlan, Fan Zhong, Zhang Peiji. Design and analysis of spherical bearings in building structures [J]. Steel Structure, 2011,26(06):6-11.

[2]Lou Feng, Pan Wenzhi, Shi Kairong, et al. Experimental study on mechanical properties of spherical bearing of large steel structure [J]. Construction Technology, 2017,46(18):35-38+102.

[3] Song Lixun. Research and development of new seismic spherical bearings for rail transit [J]. Highway, 2018,63(04):76-79.

[4]Liu Nanyan. Research on engineering application technology of a new spherical seismic bearing [J]. Steel Structure, 2016,31(12):74-78.

[5]Ma Yongbin, Zhao Bin, Gu Bingdong. Optimal design of spherical seismic bearings [J]. journal of gansu sciences, 2015,27(06):5-8.

[6]Du Xuefeng. Finite Element Analysis of New Seismic Spherical Bearings [C]. Tsinghua University, Southeast University, China Institute of Architectural Design, China State Construction Engineering Corporation, China Zhongjian Design Group Co., Ltd. Proceedings of the 5th National Steel Structure Engineering Technology Exchange Conference. Hangzhou Jingwei Architectural Design Co., Ltd. Lishui Branch, 2014:3.

[7]Chen Jieliang, Xie Xiaobo, Tan Yongchao, et al. Finite element analysis of large-tonnage seismic spherical bearings [J]. Highway Traffic Technology, 2005,(08):98-101.

[8] Shao Bing, Song Lixun. Seismic analysis of spherical bearings of rail transit bridges [J]. china municipal engineering, 2017,(02):99-101+128-129.

[9] Wang Lin. Study on mechanical properties of spherical bearings with spatial grid structure [J]. Fujian Architecture, 2024,(10):34-37.

[10]ZhiJiang Chen, Huang Lishun, Yu Qixiao. Design and application of hyperboloid spherical cast steel bearing [J]. Construction Technology, 2010,39(06):124-126.

[11] Wang Lihu, Wang Shouren, Wang Peiwen, et al. Preparation and Tribological Properties of Spherical Bearing Plane Sliding Plate for Go/CF/PTFE High-speed Railway Bridge [J]. Shandong Science, 2017,30(03):46-51+58.

[12] Yang Guang, Wang Yuanqing, Zhang Tianshen, et al. Finite element analysis of influencing factors of horizontal deflection of spherical bearings of a curved overpass beam bridge in Beijing [J]. Sichuan Building Science Research, 2014,40(01):104-108.

13. Nabhan A, Nouby M, Samy AM, Mousa MO. Contact Stressdistribution of Deep Groove Ball Bearing Using Abaqus. Journal of the Egyptian Society of Tribology. 2015 Jan 1;12(1):49-61.

14. Yang Y, Zhang Y, Ju J. Study on the mechanical properties of a type of spherical bearing. Journal of Theoretical and Applied Mechanics. 2021;59.

15. Yadi FU, Hangyu WA, Yanqing FU. Study on the Fatigue of a Spherical Bearing Under Different Loading Conditions Considering Rotation Effect. Mechanics. 2022 Jun 21;28(3):190-7.