Dynamic Characteristic Analysis of Vibration Screen Based on Joint Face Analysis

Zhiheng NIE 1,2,3,*
1. Tiandi (Tangshan) Minging Technology Co.Ltd; 2. CCTEG Tangshan Research Institute Co.Ltd; 3. Hebei Coal Preparation & Technology Research Center.
Tangshan, China
* Corresponding author: niezhiheng@cctegts.cn

Abstract—Vibrating screens play an important role in industrial production. While the cross beam fracture is the main reason for the shutdown of the screen. In order to solve the problem of fatigue fracture of the beam of the large-scale vibrating screen, the 3.6x7.3 m banana vibrating screen was taken as the research object, and the joint surface analysis was introduced at the joint surface of each part of the beam to simulate the stiffness and stress concentration of the joint surface, so that the nonlinear problems will be transformed into linear problems, and the stress and deformation of the connection area are correctly simulated, thereby this project has given an accurate basis for further optimization and optimizing the structure.

1. INTRODUCTION
Screen designs are very variable, while Fig. 1 illustrates the typical construction of large screens. Screens are mounted exciters with rotating eccentric masses. Vibrating screens are widely used in mining, coal, metallurgy and petroleum industries. With the development of industrial automation and the improvement of production efficiency, the large-scale and standardization of vibrating screen equipment has become the trend. The reliability of the vibrating screen has become the key point of the user need. Large vibrating screens experience high-cycle fatigue cycles during the entire service period, as well as random shocks caused by changes in excitation force and feed volume over time, which are far more bad than small-scale vibrating screens. Practical experiences shows that the width of the beam increases while the strain of the beam will correspondingly increase.
Practical experiences also show that when the vibrating screen has been used for about three years, the beams of the vibrating screen will generally be overhauled and replaced. However, some large-scale vibrating screens are in use due to the pursuit of high amplitude on the spot or the harsh working conditions on the site, and the screen box vibration exceeds the limit. The beams will fail in a year or even less, which seriously affects the normal production arrangement on site, causes the entire washing system to shutdown, seriously affects normal production, and brings huge economic losses to the user.

2. Operational Principle and Analysis on the Damaged Part

2.1 Basic Structure and Operational Principle
The beam of the vibrating screen is used to support the screen plate, so that the material can be screened. The cross-section of the beam has various forms such as round tube, square tube, I-beam, etc. There are several pallets in the middle, and six rivets are used between the pallet and the beam. For riveting, the angle steel supporting the sieve plate and the pallet are also riveted with rivets, and the whole is assembled and sprayed with polyurea. The structure was shown in Figure 2.

![Figure 2 Schematic diagram of vibrating screen beam structure](image)

2.2 Analysis on the Damaged Part
The cyclic stresses of the screen are almost entirely due to the rotating eccentric masses, typically around 5g zero to peak. The stress range with and without material on the deck is similar. The loading therefore approximates to a constant amplitude cyclic stress. Although there are start-up and shutdown transients, the associated stresses are typically lower than during normal operation. The structural figure of the fracture position was shown in Figure 2. The fracture position is generally in the middle of the beam.

![Figure 3 Structural Figure of vibrating screen beam structure](image)
3. PRESENT ANALYSIS OF THE FRACTURE OF THE VIBRATING SCREEN

At present, there are a lot of analysis and research on the fracture of the vibrating screen. Most of them use the finite element method to analyze. In the present finite element analysis of the vibrating screen, in order to simplify the model, domestic researchers consider that the assembly as a consolidated rigid body that bonds all the connected components together, and does not consider the influence of the connection of the bolt or the connection of the guide rail on the assembly. The harmonic response analysis is completed by the modal superposition method. It can determine the change of the structure after it bears a changing load. The vibrating screen moves linearly under the alternating load. From the harmonic response, the dynamic stress and dynamic displacement of each position of the vibrating screen under the working frequency excitation force can be obtained. According to the obtained beam stress cloud diagram, it can provide a basis for optimizing the structure of the part with excessive stress.

In actual work condition, the working frequency of the sieve machine is 14Hz, the exciting force is 650,000N, which performs linear motion. Import the 3.6 x 7.3 m banana screen three-dimensional model into the finite element software, complete the basic settings, set the modal analysis to 15 orders, and the analysis frequency to 0–100Hz , The interval is 20 times, and the damping is set to 0.01 according to the experimental data. The modal superposition method is used for harmonic response analysis. Obtain the stress cloud diagram of the beam. According to the stress cloud diagram in Figure 2, it is shown that the maximum stress under the Rigid body condition is 8.2MPa, which is much smaller than the yield limit of this material which is 24.5MPa, and according to on-site feedback, the fracture position of the beam is generally the support in the middle of the beam. At the bolt connection between the plate and the beam, the analyzed maximum stress position is different from the broken position of the beam, which cannot give a good basis for optimization.

4. NONLINEAR FINITE ELEMENT ANALYSIS ON THE SCREEN

4.1 Introduction to Joint Analysis

The same mechanical structure exhibits different dynamic characteristics under different stresses. In a complex assembly, many parts are connected to each other through joint surfaces. Data shows that the total stiffness of the whole structure is 60% to 80% of the total damping comes from the joint surface, and the various parts are riveted by the bolt joint surface. The existing analysis does not consider that the pallet is connected by the pull rivet bolt, nor does it consider the distribution of the pull rivet bolt. It is impossible to correctly simulate the stress and deformation of the connection area, and these conditions are the conditions to be considered in the establishment of the model.

Therefore, it is necessary to import the analysis of the joint surface and simulate and optimize the joint surface stiffness. The modal calculation of the joint surface analysis is based on the ordinary modal calculation and further considers the influence of the stress stiffening matrix. In other fields such as aviation Engines, machine tools and other fields have taken into account the influence of the force on
the combined face, and improve the reliability of the machine by optimizing the connection structure. In the field of vibration machinery, there is currently no literature report in the domestic related research.

The modal calculation formula of the finite element method is:

$$\{M\}\{x\} + [K]\{x\} = 0 \quad (1)$$

$M$ is the mass matrix; $K$ is the stiffness matrix:

$$\{x\} = [M^{-1}]\{\delta\}\cos\omega t \quad (2)$$

and the following equation can be obtained:

$$[[K + S] - \omega^2 [M]]\{\delta\} = 0 \quad (3)$$

When performing joint surface analysis, the stress stiffening matrix will be considered, and the joint surface modal algorithm equation is obtained:

$$[[K + S] - \omega^2 [M]]\{\delta\} = 0 \quad (4)$$

4.2 Application of Boundary Conditions

Due to the excessive number of bolts in the overall structure. This article quotes a new simplified method used in aero-engine vibration analysis, which improves work efficiency and ensures accuracy. When prestress is applied, the limit torque of the riveting bolt is 246Nm. From the calculation formula of bolt tightening force, it can be obtained The tightening force is 77.5kN, after setting the contact, meshing, applying load, setting the load step setting, then the application of boundary conditions is completed.

![Application of boundary conditions](image)

4.3 Solution

The stress cloud chart of the finite element analysis was shown in Figure 6.

![Stress cloud chart of the beam](image)

Shown from Figure 6, the stress at beam was extremely small; its maximum stress appeared at the screen beam as $\sigma_{\text{max}} = 23.079\text{MPa}$ in red colour, which is close to the yield limit of this material which is 24.5MPa, and will fail after thousands hours in operation, equivalent to $4 \times 10^8 - 8 \times 10^8$ cycles.

There also exist stress concentration. Stress concentration refers to the phenomenon of significant increase of stress in local range due to external factors or its own factors. The presence of bolt holes will reduce the structural strength of the beam.
The maximum stress appeared at the screen beam is near the bolt hole in the middle of the beam, which is consistent with the actual fracture position of the beam. The solution can accurately simulate the stress and deformation of the connection area. The beam transmits the exciting force to the screen plates and the material. Since the load carried by the beam is transmitted through the connection part, which will cause the fatigue damage. The structure should be optimized.

5. STRUCTURE OPTIMIZATION

According to the analysis results, To reduce the stress concentration, the rivets between the beam and the pallet were changed from six to four. The results are shown in the figure below. It can be found that the position has not changed, but the stress distortion near the bolt hole is eliminated.

![Stress cloud chart of the beam after optimization](image)

It showed that, its maximum stress appeared at the screen beam as $\sigma_{\text{max}} = 14.805 \text{MPa}$, which is not as close to the yield limit of this material which is 24.5MPa as the result of six rivets, less than the allowable stress amount for the Therefore, it accorded with the standard.

![Schematic diagram of vibrating screen beam structure after optimized](image)

According to the analysis results, the structure is optimized as shown in the Figure 6. It has been sent to the site as an accessory for replacement, and after replacement, it has been in safe operation for two and a half years, equivalent to almost $10^9$ cycles, and the expected effect has been achieved.

And the design of the new machine has been changed and optimized into the new structure.

6. CONCLUSIONS

Firstly, guided by the current finite element simulation method of large vibrating screen, the author presented the beam’s stress cloud chart of the finite element analysis which is 8.2MPa, and is much smaller than the yield limit of this material which is 24.5MPa, cannot give a good basis for optimization because in practical applications, the beam of the banana vibrating screen has been structurally damaged and the fracture position does not match the actual situation.

Secondly, In order to fit the actual situation and accurate the optimization, the boundary conditions of the model need to be modified. The maximum stress appeared at the screen beam as $\sigma_{\text{max}} = 23.079 \text{MPa}$, is close to the yield limit of this material which is 24.5MPa, and the position
consistent with the actual fracture position of the beam. The solution can accurately simulate the stress and deformation of the connection area.

Finally, to reduce the stress concentration, the rivets between the beam and the pallet were changed from six to four. the stress distortion of the beam near the bolt hole is eliminated. Its maximum stress appeared at the screen beam as $\sigma_{\text{max}} = 14.805 \text{MPa}$, which is not as close to the yield limit of this material which is 24.5MPa as the result of six rivets, less than the allowable stress amount for the Therefore, it accorded with the standard. The stress of this structure is favorable.

ACKNOWLEDGMENT
Fund projects: Special project of science and technology innovation and entrepreneurship fund of Tian Di Science & Technology Co.Ltd.(2020-TD-QN001).

REFERENCES
[1] Cheng H Z. (2012) Present situation and development trend of coal preparation technology in China. J. Coal Preparation Technology, (02): 79-83.
[2] Wang Z S. (2007) Characteristic analysis of vibrating screen based on finite element method. J. Coal mine machinery, 28(9): 68-69.
[3] He X M, LIU C S, Zhang C Y.(2008) Dynamic characteristics of a large-scale vibrating screen with a statically indeterminate mesh beam excitation structure. J. Journal of China Coal Society, 33(9): 1040-1044.
[4] Li D, Liu Y, Hu H K, Yuan W Q, Zhang M T, Meng G Y.(2015) Study on the modal analysis of vibrating screen based on ANSYS Workbench. J. Coal Mine Machinery, 36(8): 280-282.
[5] Lu G.(2016) Analysis of dynamic characteristics of large-scale high-frequency vibrating screen. J. Mining Machinery, 44(6): 58-61.
[6] Tian Jianyong.(2014) Calculation analysis and experimental investigation of bolt contact stiffness.D.Dalian University of Technology.
[7] Li L M. (2005) ANSYS Practical course of finite element analysis. Tsinghua University Press, Beijing.
[8] Ma B. (2014) Analysis of bolt connection of aero-engine thin-walled casing. D. Dalian University of Technology.
[9] Zhao J W, Wang H Y. (2001) Structural finite element analysis. Science Press, Beijing.
[10] Mohamed Sajeer, Arka Mitra, Arunasis Chakraborty. (2020) Spinning finite element analysis of longitudinally stiffened horizontal axis wind turbine blade for fatigue life enhancement. J. Mechanical Systems and Signal Processing, 145.
[11] Cheng Y F. (2015) Structural optimization design and modal analysis of linear vibrating screen. D. Taiyuan University of Technology.
[12] Zhang Z C. (2019) Screen box structure optimization considering fatigue life. D. Yanshan University.