Finite element analysis of belt conveyor bracket

Ming-Yuan Zhang¹ᵃ, Hai-Dong Zhu²ᵇ*, Wei Zhang³ᶜ
¹²³College of Mechanical and Electrical Engineering, Guilin University of Electronic Technology, Gui Lin 541004, China
ᵃemail: 1801302059@mails.guet.edu.cn
ᵇ*corresponding author email: zhuhaidong@guet.edu.cn
ᶜemail: 1801302061@mails.guet.edu.cn

Abstract. As an important part of the belt conveyor, the triangle bracket supports the roller and carries its load. Its stability and service life directly affect the function of the conveyor, so that the conveyor system can work stably and efficiently. Due to the characteristics of the belt conveyor, such as long transportation distance and large conveying volume, there are many problems in the actual working condition of the triangle. For example, the triangle bracket bears too much load, resulting in deformation and damage of the bracket structure. Working time is too long, the yield limit of the support is reduced, and the structure of the support is seriously failed, which leads to the shutdown of the conveying system and huge economic loss of the enterprise. There are many problems in the actual production of triangle bracket, so it is necessary to analyze and study triangle bracket to solve these problems. This project takes the triangle support of a belt conveyor as an example, establishes a three-dimensional solid model of the support, conducts finite element analysis on the triangle support model with ABAQUS software, analyzes the stress-strain displacement of the triangle support under the load, and optimizes the triangle support according to the finite element analysis results.

1. Introduction
With the continuous progress and development of industrial revolution and science and technology, as the main conveying tool of material transport, belt conveyor is developing towards the goal of long distance, fast belt speed and large conveying volume. Due to the characteristics of the belt conveyor, such as long transportation distance and large conveying volume, there are many problems in the actual working condition of the triangle. For example, the triangle bracket bears too much load, resulting in deformation and damage of the bracket structure. Working time is too long, the yield limit of the support is reduced, and the structure of the support is seriously failed, which leads to the shutdown of the conveying system and huge economic loss of the enterprise. The finite element analysis method in contemporary design scheme is an advanced method to realize innovative design, which greatly improves the design level, reduces the design time and reduces the expenditure of design. For this, this article will use the contemporary design of finite element analysis technology, using the ABAQUS finite element analysis software to the triangle bracket are in the process of working stress and strain and displacement is analyzed, according to the analysis of the results to guide the design of triangular bracket, providing theoretical basis for the triangle bracket design, complete the important technology in the design of belt conveyor triangle bracket.
2. Outline and stress analysis of belt conveyor triangle bracket

2.1. Overview of belt Conveyor triangle bracket

Belt conveyor is the machine equipment to transport goods, widely used in mineral transport, logistics transport, grain transport, machine transmission and other aspects. The belt conveyor is mainly composed of end unloading, driving drum, cleaning device, guide drum, unloading trolley, conveyor belt, upper and lower idlers, feeding hopper, tensioning drum, tensioning device, etc. Depending on its transport and delivery capacity, belt conveyor can be divided into large conveyor machines such as industrial and mining belt conveyor delivery machine, small conveyor such as applied in electronic products, light industry, medicine and other industries. Belt conveyor has high transport capacity, long distance transport, simple composition and easy maintenance, easy to operate and operate. Belt conveyor running speed, high stability, low noise, and can be horizontal transmission and different levels of transmission, applied in many complex working conditions.

Triangle bracket is an important part of the belt conveyor, which is mainly installed at the end of the belt conveyor as a belt. The unloading and transporting device of the type conveyor is mainly composed of vertical column, diagonal brace, bottom beam, reversing roller, driving roller and conveyor belt. The triangular bracket is a welded part, which is mainly made of carbon structural steel (Q235-A).

2.2. Stress analysis of triangle bracket of belt conveyor

In this paper, the triangle bracket of belt conveyor is taken as an example to get the structure diagram of force calculation, the calculation formula of force component and the force size of the column. Figure 1 is the stress analysis diagram of the triangular bracket structure.

The circumferential driving force required on the driving drum is the sum of all resistances, that is $F_u$. Such as formula 1.

$$F_u = C/L g [q_{RO} + (2q_B + q_G)] + F_{S1} + F_{S2}$$  \hspace{1cm} (1)

The coefficient $C$ is 1.5. Is the simulated friction coefficient $f$, which is selected according to the working conditions and the manufacturing and installation levels. The value is 0.023. Is the length of the conveyor $L$ (center distance of the head and tail roller), and the value is 146.505m; Is the acceleration of gravity, $g = 9.81$ m/s$^2$; Is the mass of the rotating part of the bearing branch idler per meter long, its value is 18.45kg/m; Is the mass of the conveyor belt per meter $q_{RO}$, and its value is 17.66kg/m; Is the quality of conveying material per meter long $q_G$, its value is 111.11kg; Is the special main resistance $F_{S1}$, namely the forward roll friction resistance and the guide groove friction.
resistance, the value of which is 2523N; Additional resistance $F_{s2}$, that is, the resistance of sweeper, unloader and turn-back branch conveyor belt, is 4680N.

3. **Finite element analysis of triangular bracket of belt conveyor**

3.1. *The establishment of finite element triangular bracket model and its material properties*

According to the actual use of the triangle support of belt conveyor and the parameter requirements of the structure size, the three-dimensional modeling of the triangle support structure was carried out. The two-dimensional plane size diagram was established in SolidWorks software, and then the three-dimensional model was obtained by drawing, mirror image, cutting and other commands and imported into ABAQUS software. On the premise of not affecting the analysis results, in order to facilitate the subsequent meshing and the subsequent modification and processing of the model, when establishing the three-dimensional solid model of the triangle bracket, it is necessary to reasonably simplify the structure of the bracket, such as ignoring the rounded corners and chamfered corners on the bracket to create a physical model in line with the reality.

The triangular bracket of the belt conveyor is endowed with material properties, which is welded by carbon structural steel (Q235-A). According to the properties of metallic materials, the elastic modulus of the triangular upsilon was defined as $E=210$GPa, allowable stress $[\sigma]=130$MPa, and poisson's ratio upsilon $\nu=0.28$.

3.2. *Grid division, boundary condition processing and load application*

In this paper, the three-point method is used to define the cutting plane for proper segmentation. After determining the type of hexahedral elements, the bracket model was seeded. The global seed option was opened in the grid interface, and 5.5 was input in approximate global size. The rest were default, divided into 2,308 hexahedral elements and 4,183 nodes.

The bottom end of the triangle bracket is fixed on the ground with anchor bolts, so the degrees of freedom of the upper, lower, left, right, front and back of the triangle bracket are fixed. It is known that during the normal operation of the belt conveyor, the column of the bracket bears the maximum load, and the horizontal component force of the column is calculated as $F_{1a}=53823$N, and the load is applied to the column in the ABAQUS analysis software.

3.3. *Analysis of calculation results of triangle bracket of belt conveyor*

In the ABAQUS/ Standard solver, the finite element analysis of the triangular bracket is carried out, and the resulting nephogram of stress, strain and displacement is obtained. See Figure 2.
The maximum stress of the support structure exists at the bottom of the support column, and its maximum stress is 109.6MPa, less than the allowable stress value of the support $\sigma = 130\text{MPa}$. The safety factor is 0.843, which meets the strength requirement, but the stress is almost close to the allowable stress. In the actual situation, the structure of the bracket is unstable, which affects the normal work of the triangle bracket. At the same time, the stress on the diagonal brace of the triangular bracket is relatively uniform, which is about 60MPa, and the safety factor is 0.4, meeting the strength requirements. The maximum strain of the scaffold structure exists below the scaffold column, and the maximum strain is $5.438 \times 10^{-11}$, and the allowable strain is $\delta = 6.19 \times 10^{-4}$. The maximum strain is less than the allowable strain and meets the stiffness requirement. The maximum displacement of the triangle bracket appears at the position where the driving drum is installed above the triangle bracket, and the maximum displacement is 1.713mm. However, the maximum displacement result is relatively large, and the deformation will affect the normal operation of the support under the actual working condition.

3.4 **Structural optimization of supports**

The 3d Solid model was created in the SolidWorks software, and the width of the column was increased by 10mm during the creation process, so that the cross-sectional area of the column was increased by 1000mm$^2$. Then the 3d Solid model was imported into the ABAQUS software for finite element analysis, and the stress-strain displacement results of the scaffold structure were obtained. The optimized result analysis diagram is shown in Figure 3.
Figure 3. Stress, strain, and displacement neutrograms

(a) Stress nephogram of support structure

(b) Strain cloud map of the support structure

(c) Support structure displacement cloud map
4. Conclusion
In this paper, the triangle bracket of belt conveyor is taken as the research object, and the finite element analysis software ABAQUS is used to analyze the operation of the triangle bracket under normal working conditions, and the stress-strain displacement results of the triangle bracket are obtained. Based on the triangle support the stress finite element analysis is made on stress and strain displacement nephogram, can see clearly support the biggest stress and strain in the bottom of the column, the maximal displacement appears above the triangle bracket installation position of the driving drum, maximum stress is 109.6 MPa, maximum strain was 5.438 e-11, the maximal displacement of 1.713mm. In this paper, the structure of the scaffold is optimized. After optimization, the maximum stress of the scaffold is reduced by 37.32MPa, the maximum strain is reduced by 1.734e-11, and the maximum displacement is reduced by 0.354mm. The analysis results show that the maximum stress-strain displacement of the triangular scaffold is significantly reduced, and the performance of the scaffold is significantly improved, so as to achieve the purpose of optimization.

Acknowledgments
This research was supported by the Natural Science Foundation of Guangxi Province (No.2017GXNSFAA198122).

References
[1] Zheng X. (2007). Research on THE CAD/CAE Technology of The Complete Machine and Head Bracket of the Belt Conveyor. (The Little Book of Resistance against Globalization, Zhengzhou University).
[2] Li, Y. (2002). Dynamic design and theoretical calculation of belt conveyor. Hoisting and transportation machinery (03), 8-12.
[3] Xue H, Hu W, & Shang X. (2004). Research on dynamic Characteristics of belt Conveyor based on virtual prototyping technology. Lifting transport machinery, 000(010), 5-8.
[4] Xi P. (2005). Dynamic Simulation of belt Conveyor power failure process. Hoisting transport Machinery (8), 37-39.
[5] Huang G. (2004). Basic finite element method and ANSYS application. Machinery industry press.