Geosciences and Engineering, Vol. 11, No.1 (2023), pp. 110–116.
https://doi.org/10.33030/geosciences.2023.01.009

FINITE ELEMENT ANALYSIS OF A STEEL BRIDGE FRAME FOR BELT CONVEYORS

SÁNDOR SZIRBIK[1]*, ZOLTÁN VIRÁG[2]
[1]**Institute of Applied Mechanics, University of Miskolc; sandor.szirbik@uni.miskolc.hu*
[2]*Institute of Mining and Energy, University of Miskolc; zoltan.virag@uni.miskolc.hu*
[1]https://orcid.gov/0000-0002-7757-3750
[2]https://orcid.gov/0000-0003-2259-8878

Abstract: Due to lignite production in open-pit mining, the need to transport overburdened rock also increased, which can be handled cost-effectively by recycling existing equipment like a bridge frame of conveyor belt system. This paper briefly outlines the 3D finite element stress analysis of a bridge structure, which is reused to assist the path of the conveyor belt in the work area. In this paper, the steel bridge, which consists of various structural steel beams, is modeled as a 3D frame using beam elements. Our investigation of the bridge frame refers to the operation of the conveyor belt during transport at full capacity. Considering that the loads applied are derived from only the masses, in this article, the effects of transverse vibrations of the belt are neglected on the structure. The results of the simulations were in favor of reusing the bridge frame, as it has sufficient reserves up to the proportional stress limit.

Keywords: *conveyor belt, bridge frame, finite element analysis, static safety factor*

1. INTRODUCTION

A new belt conveyor was installed within a frame, which is wider than the previously installed unit, thus the total weight of the belt conveyor increased compared to the previous operating load of the structure. Another substantial difference is that before, the unit had to do a new function that would convey overburden rock instead of lignite. The bulk density of the overburden's material is significantly higher than the lignite's, this means that, an extra load relative to the previously designed operating situation of the bridge. A stress assessment for this bridge was presented by Ladányi [1], which contains pencil and paper techniques for a 2D truss bridge. However, the simulation by finite element analysis with beam elements with more complexity shows the behavior of the bridge under loads and provides more information. In fact, the focus of an assessment of the robustness of this type of structure is based on a 3D geometric model for the frame that is verified with a finite element analysis for changed loads. To avoid overloading, the stress and displacement distributions in the finite element analysis show the response of the 3D structure at higher load capacities.

Finite element simulations have been very popular method for design in recent years. They have numerous advantages, for example, they provide reliable, fast, and high-accuracy results in engineering simulations in various industry fields. In this paper, a conventional element type, the linear beam element is applied. The deformations and more accurate stress distributions can be analyzed using FEA with beam elements. Since pin-jointed truss elements are such one-dimensional bars or rods that are assumed to deform by axial stretching, the use of truss elements does not model how the structure deforms under bending.

The frame structures are widely used in mining equipment because of their lightweight and simple maintenance which are very important in open pit mines. The vibrational properties of these structures can also be investigated by numerical and experimental modal analysis with dynamic loads. For instance, mechanical vibrations of the boom of a bucket wheel excavator during operation are analyzed via a modal analysis which is useful for vibration control to identify the natural frequencies [2]. Frequency response analysis is essential, mainly, in the case of the structural elements of the mining machine to know the frequencies at which the structure does not remain stable but may behave erratically so that there are investigations in different kinds of industrial fields, e.g., mining excavators [3].

2. Model geometry of the frame

The geometry of the 23 m length, 2.3 m height, and 3 m width bridge frame is illustrated in *Figure 1*. The structural design is asymmetrical due to the arrangement of the inclined members shown in the figure, which results from the previous rebuilds of the structure and which we accepted as a given for cost-effectiveness. Damages from the overloading of the bridge structure, etc. must be continuously monitored, which is why a sidewalk is also placed on the bridge on one side along the conveyor belt. Thus, the use of geometric symmetry to reduce the size of the problem is not available.

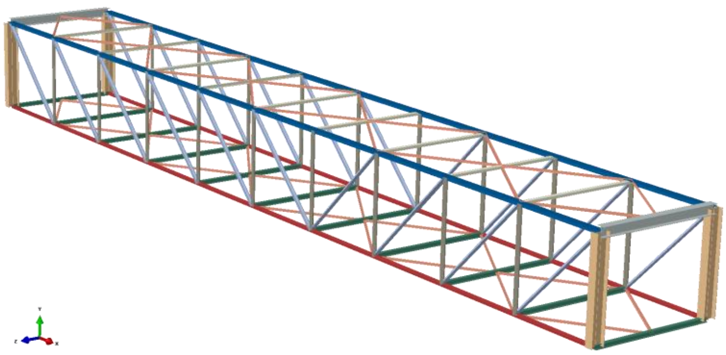

Figure 1
3D model of the bridge frame for a belt conveyor

The cross-section of the upper struts and the lower and upper bridge braces is a non-conventional T-shaped cross-section ($b = 110$ mm, $h = 75$ mm, $t_f = 7$ mm, $t_w = 14$ mm). The lower struts and the frame of the gates at two ends are steel structural I-beams with dimensions of 100, 200, and 315 mm in height. Lateral bracings are steel structural beams and have an L-shaped cross-section ($b = 50$ mm, $h = 50$ mm, $t = 6$ mm). The complex profile of the columns of the bridge is modeled as a generalized profile, replaced by a rectangular cross-section ($b = 127.08$ mm, $h = 23.43$ mm). The moments of inertia and area of that rectangle closely approximate the real data of the columns. It is necessary to produce the data of this rectangular because using the program is only effective for stress distributions in the case of built-in cross-sections like T, L, U, etc. profiles and based geometries like circle and rectangular.

The standard steel material is assumed to be isotropic elastic, for which Young's modulus of $E = 2.1 \times 10^5$ MPa, Poisson's ratio of $\nu = 0.25$, and density of $\rho = 7.85 \times 10^{-9}$ t/mm^3 are prescribed.

3. FE MODEL DESCRIPTIONS

The frame is supported at four points of the lower corners *(Figure 2)*. Three types of 3D supports are used in the FE model of the bridge frame. One is the pinned support describes zero displacement in three directions. The other is a hinge support that provides two degrees of restraint, vertical and horizontal, and only rotational displacements and one-direction motion can occur. Finally, two roller supports are used, which provide only one degree of restraint, in the vertical direction, and horizontal and rotational displacements are also possible. Applying these types of prescribed displacement boundary conditions prevents the rigid body motions of the structure but ensures translations along the four supports.

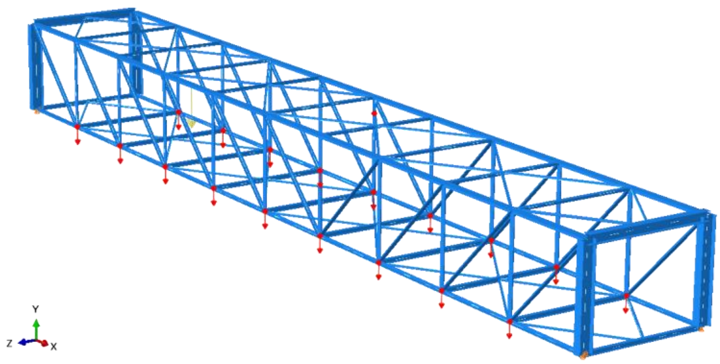

Figure 2
Apply loads and boundary conditions to the bridge frame

The moving belt guiding on conveyor idler rolls, the conveyor belt support frame underneath, and the transported materials to be moved are assumed to be 270 kN of the weight. For simplicity, that load is equally given at the eighteen nodes of the lower bridge braces, as shown in *Figure 2.*

The finite element method, which is a very popular numerical technique for design in recent years, is used to numerically solve differential equations arising in engineering problems and has received more and more attention in various industry fields. The main concept of the technique is that the geometry of the structure subdivides into non-overlapping small parts. These so-called finite elements are implemented by the construction of a mesh. The conventional element types [4] possess simple shaped geometry with well-defined stress displacement relationships. Thus, the sufficiently refined meshes need to ensure that the results from simulations are adequate [4]. Accordingly, the frame is meshed into finite elements, which are 2-node beam elements (Abaqus element type B31 [5]) with linear interpolation formulations in three-dimensional space with various profiles according to the part of the structure *(Figure 2)*, and the element number is 1936.

4. Results of finite element analyses

To determine the self-weight of the bridge frame, the gravity load associated with the given geometry is used in the finite element analysis. It is defined as exactly equal to the product of the self-mass and the gravitational acceleration, where $g = 10\ \mathrm{m/s^2}$, and the density is also known. This means that the self-weight is 62.17 kN. The deformation and strength properties of the structure were also investigated using FE analysis for stress/displacement analysis. The effect of the self-weight is illustrated in *Figure 3*, in which the Von Mises stress is depicted on the deformed geometry of the bridge frame.

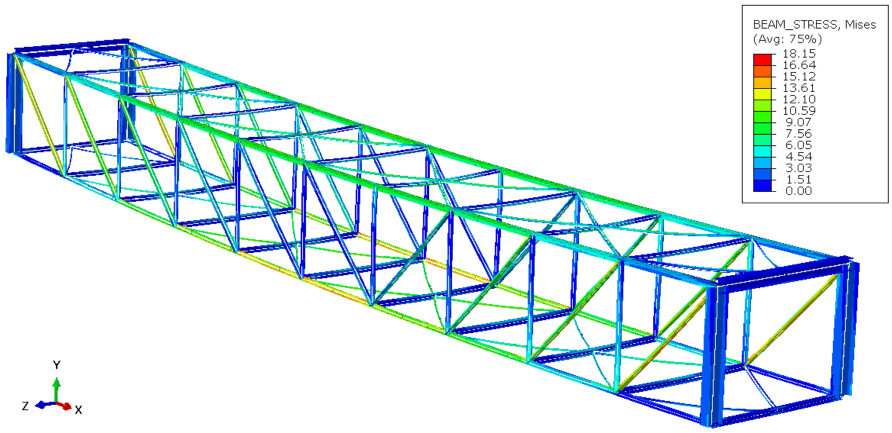

Figure 3

Von Mises stress in the beams from the self-weight of the bridge frame

Figure 3 shows that the maximum values of Von Mises stress can be found in the middle, on the lower bridge braces, and the first lateral bracings next to the gates. Their values are not more than 18.15 MPa. These members are the main load-bearing elements. The I-beam gates at the ends strengthen the structure, thus increasing the robustness.

The stress distribution resulting from the load of 270 kN, which is equally prescribed at the eighteen nodes of the lower bridge braces, is depicted in *Figure 4*. The I-beam gates at the end of the bridge remain the structural elements that provide reinforcement to the structure and have a significant effect on its load capacity. Von Mises stresses are increased in the main load-bearing elements, but their values are below 89.86 MPa. If the structure is made of grade S275 steel with a minimum yield strength of 275 MPa, the ratio of the yield stress and maximum value of Von Mises stress is equal to 3.06 as a static safety factor. For steel structures, the safety factor value of 3.06 is well above the acceptable lower limit.

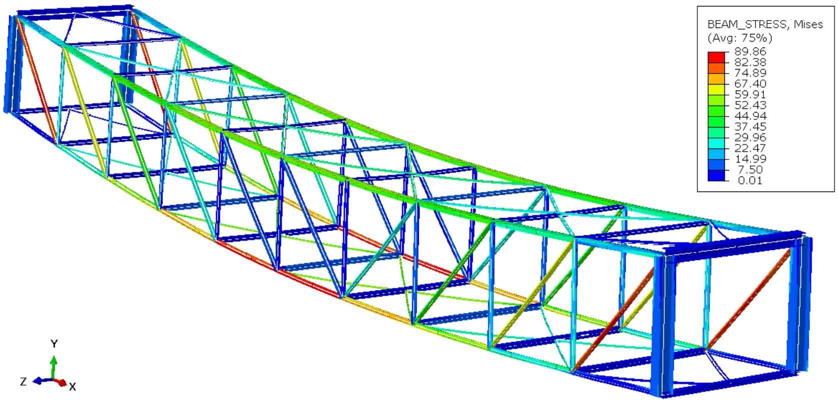

Figure 4
Von Mises stress in the beams from loads on the bridge frame

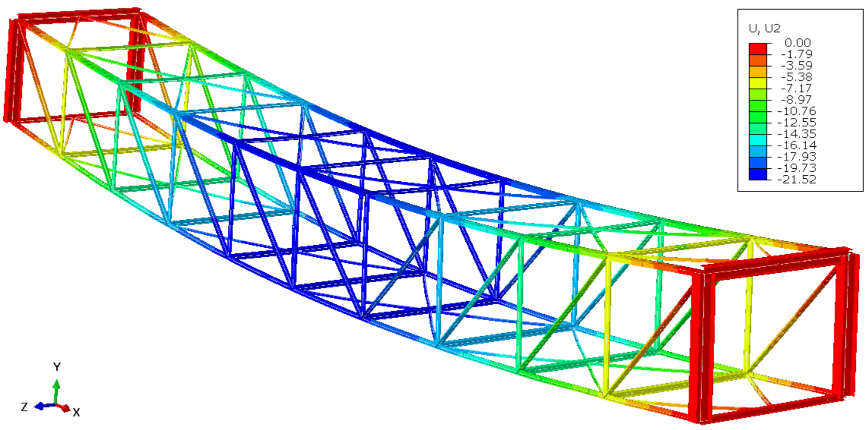

Figure 5
Vertical displacements on the bridge frame

Our goal is to know that the structure is properly designed and constructed considering loading during their planned lifespan to burden the new operating load of the structure. The FE analysis has provided the displacement and stress distributions so that the strength behavior can be further investigated, for instance, by buckling analysis which provides the factor by which the live load must be multiplied to reach the buckling load. In our FE analysis, the displacement distributions can be investigated, and the results are for the 270 kN load shown in *Figure 5*. The maximum value of the vertical displacement is 21.52 mm in the middle of the bridge, which means that the deflection ratio is 0.094%. The structure of the bridge frame remains rigid enough under the examined load conditions, which is advantageous in terms of the idler sets of the conveyor belts.

5. CONCLUSIONS

The goal of this study was the recycling of the bridge frame for the conveyor belt. This structure had been properly designed and built considering a load from the conveyor belt transporting lignite during its planned lifetime does not exceed the proportional stress limit. The question was whether the new conveyor belt could be incorporated into the bridge frame while the maximum value of the stress of the bridge remains below the proportional stress limit despite the new substantial modifications.

Our finite element analysis with beam elements provides more detailed information and calculates components of displacements and stresses to examine the behavior of the bridge frame under loads. We have stated that the robustness of the bridge frame for the conveyor belt is adequate because the values of the deflection ratio and the static safety factor also prove that for the new load conditions from the new conveyor belt and the transported materials to be moved. Furthermore, the I-beam gates at the ends also strengthen the structure, which greatly increases robustness.

The dynamic behavior of this structure can be further investigated by modal analysis and dynamic response analysis to obtain the forced response on the bridge frame structure caused by the non-linear vibrations from the transporting belt and idler sets.

REFERENCES

[1] Ladányi, G. (2017). Strength assessment of a truss bridge for belt conveyors. *Annals of the University of Petroşani, Mechanical Engineering*, 19, pp. 91–96.

[2] Popescu, F. D., Radu, S. M., Andras, A., Brînas, I., Marita M. O., Rebedea, N. I. (2020). The modal analysis, using simulation and modelling, of the boom of the ERC-1400 Bucket Wheel Excavator during operation. *Applied Mathematics, Mechanics, and Engineering*, 63 (4), pp. 353–362.

[3] Popescu, F. D., Radu, S. M., Kotwica, K., Andras, A., Kertesz, I. (2019). Simulation of the Time Response of the ERC 1400-30/7 Bucket Wheel Excavator's Boom during the Excavation Process, *Sustainability*, 11 (16), p. 4357.

[4] Bathe, K. J. (1996). Finite Element Procedures, Prentice-Hall Inc.: Englewood Cliffs, NJ, USA.

[5] Abaqus 6.13 Online Documentation; Dassault Systems, (2015). Available online: http://130.149.89.49:2080/v6.13/index.html