MODAL ANALYSIS OF A BRIDGE FRAME FOR BELT CONVEYORS

SÁNDOR SZIRBIK [1], ZOLTÁN VIRÁG[2]

Abstract: This paper is devoted to the modal analysis of a 3D bridge frame subjected to the weight of the materials to be moved during full-capacity transport, where the moving belt rests on suspended hanging roller sets from beams installed mounted in the middle of bridge columns, as an alternative instead of the underlying conveyor belt support frame. The scope of our investigation is limited to the vibration of the frame bridge when the loaded conveyor belt is substituted in the FE model with concentrated forces on the columns. The structure of the bridge frame is reused to assist the path of conveyor belts in the work area. The bridge frame, which consists of various structural steel beams, is modeled as a 3D frame using beam elements with various profiles. The purpose of finite element analysis is to provide a starting point for studies of the transient forced vibrations with the results of modal analysis.

Keywords: conveyor belt, suspended hanging rollers, bridge frame, FEA, modal analysis

1. INTRODUCTION

Various beam structures are widely used in mining equipment because of their lightweight and simple maintenance which are particularly 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 [1]. Frequency response analysis is essential 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

[1] *Associate Prof, Eng. Ph.D., University of Miskolc, Hungary, sandor.szirbik@uni-miskolc.hu*
[2] *Associate Prof, Eng. Ph.D., University of Miskolc, Hungary, zoltan.virag@uni-miskolc.hu*

erratically so that there are investigations in different kinds of industrial fields e.g., mining excavators.

A frame structure similar to the boom of the excavator is also used in conveyor belt systems, where various bridging is solved with it. This bridging can take place both horizontally and vertically between levels. The structure of the bridge frame is used to assist the path of conveyor belts in an open pit mine work area. The waste belt conveyor is wider than the previously installed lignite unit, so the total weight of the belt conveyor increases compared to the previous operating load of the structure. The bulk density of the waste 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 [2], which contains pencil and paper techniques for a 2D truss bridge. However, the simulation by finite element analysis with linear beam elements with more complexity shows the behavior of the bridge under loads and provides more information [3]. The assessment of the robustness of this type of structure is based on a 3D geometric model of the frame. Natural frequencies of the bridge frame are verified with a modal analysis in the case of suspended hanging roller sets and changed loads. Drawing the conclusion from finite element analysis, the modal analysis shows the response of the 3D structure at higher load capacities.

Finite element simulations have numerous advantages, for example, they provide reliable, fast, and high-accuracy results in engineering simulations in various industry fields [4, 6]. In this paper, a conventional element type, namely the linear beam elements, is used. The deformations of beams 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.

2. MODEL GEOMETRY OF THE FRAME

This section is devoted to the description of the bridge frame structure. The geometry of the 23 m long, 2.3 m high, and 3 m wide structure is illustrated in Figure 1.

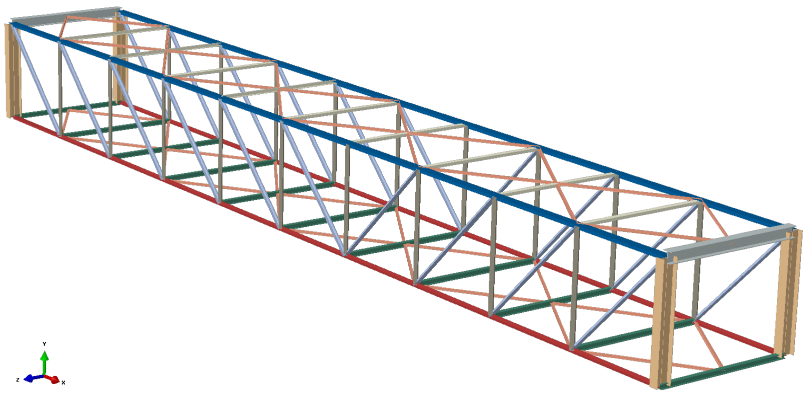

Fig. 1. The 3D model of the bridge frame for a belt conveyor

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. During the simulations, the reinforcement on the longitudinal beams is neglected so that the support structure with its original cross-sections is considered at the full length of 23 m. It is also necessary due to 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. This construction is similar to that found on mining machinery such as mining excavators, spreaders, etc. visible. There is no remained space for the walking pad inside the structure, next to the new wider conveyor belt. Thus, the use of geometric symmetry is not available to reduce the size of the problem.

The cross-section of the upper struts and the lower and upper longitudinal beams 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). Axes 1 and 2, as shown in Figure 2, help to orient the cross-sections on the 3D geometric model.

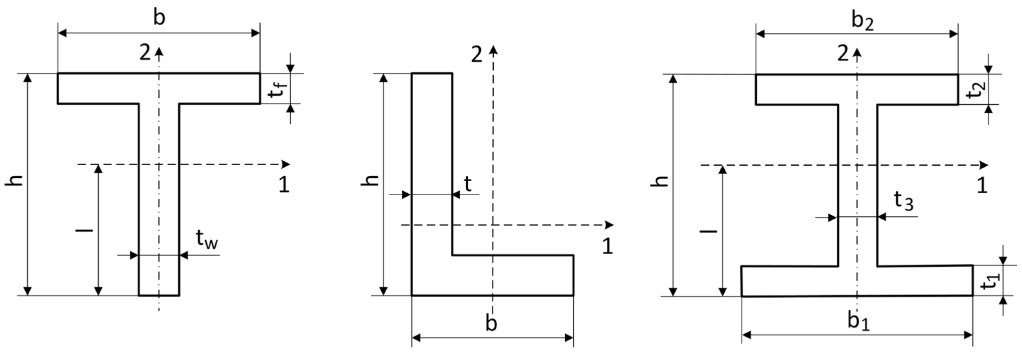

Fig. 2. Cross-sections of the beams

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 [4] 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.

3. FE MODEL DESCRIPTIONS

The frame is supported at four points of the lower corners. Three types of 3D supports are used in the FE model of the bridge frame. One is the pinned support, which describes zero displacements 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.

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 $\upsilon = 0.25$ and density of $\rho = 7.85 \times 10^{-9}$ t/mm^3 are prescribed.

The total load of 27×10^4 N is calculated from the weight of the steel structure elements of the suspended hanging roller sets, the weight of the belt, and the weight of the transported material. This load is imposed in the vertical direction, uniformly distributed, at the eighteen midpoints of the columns, where suspended hanging roller sets on beams are mounted. Due to suspended hanging rollers, where the angle of the wing rolls is 30°, the perpendicular concentrated force components are prescribed as 25.981×10^3 N at the same points, as shown in Figure 3 with red vectors. The walking pad is modeled with concentrated force components applied at the nine lower endpoints of the columns on one side by 1.5×10^3 N.

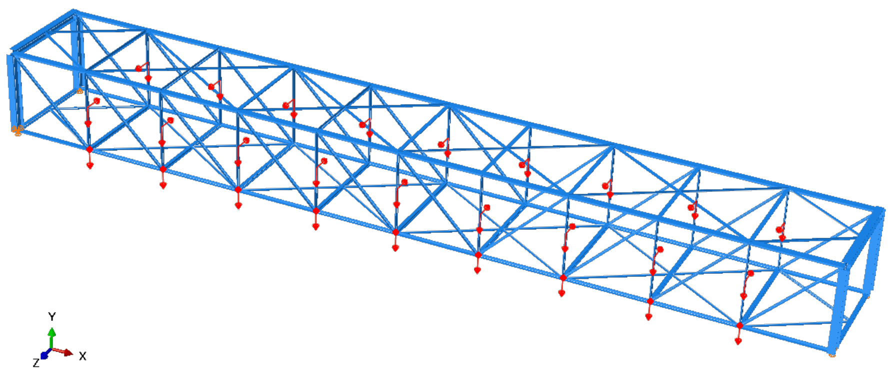

Fig. 3. Apply loads and boundary conditions to the bridge frame

Two analysis steps are used for the natural frequencies belonging to the 3D bridge frame under loading. The constant load resulting from full-capacity transport is taken into account in both steps of the FE analysis. In the first step, a static analysis procedure with geometric nonlinearity is performed, so that the structural deformation determined at the end of the first general analysis step can be included in the second step, which is eigenvalue extraction with linear perturbation, as seen 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. The maximum value of the vertical displacement is 23.15 mm in the middle of the bridge, which means that the deflection ratio is 0.01%.

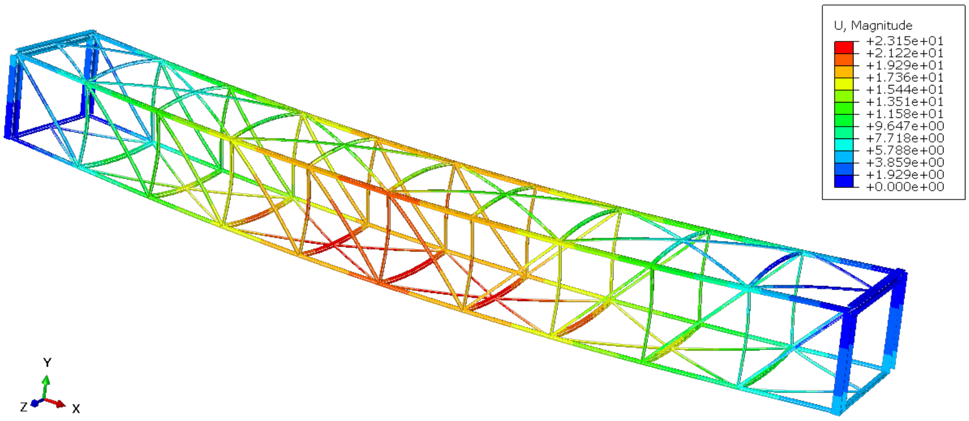

Fig. 4. Magnitude of displacements on the bridge frame under static loading conditions in step 1

The bridge frame under full-capacity transport is analyzed via a modal analysis to identify the natural frequencies that are useful for vibration control and are essential to frequency response analysis. To investigate the natural frequencies, a linear perturbation analysis is performed in the commercial software Abaqus, where the frame is meshed into finite elements, which are 2-node beam elements (Abaqus element type B31 [5]) with linear interpolation formulations. The natural frequencies are presented in Table 1. The other two cases in Table 1 are reported to compare them. In loading case A, the conveyor belt is not installed in the bridge frame. Loading case B is that when the moving belt rests on conveyor idler rolls, the underlying conveyor belt support frame [2]. Column C in Table 1 contains the first six natural frequencies when the moving belt rests on suspended hanging roller sets on two beams installed.

Table 1. **Comparison of natural frequencies [Hz] of the first six mode shapes according to loading cases A, B, and C**

Mode sequence number	Loading case		
	A	B	C
1.	7.727	7.701	7.847
2.	9.285	9.284	9.408
3.	10.095	10.149	13.789
4.	14.480	14.476	14.549
5.	14.738	14.548	15.461
6.	16.004	15.361	16.610

By comparing these frequencies with the simulation results without load (Loading case A), the average changes in the numerical values are not very significant, and the corresponding mode shapes are very similar to the results in A, as we expected. The different types of conveyor belt installation affect the magnitude and the line of action of the forces transferred to the structure so the load states on the beams are also modified, which change the natural frequencies as seen in the 3rd and 4th columns of Table 1.

4. CONCLUSIONS

The goal is to recycle the bridge frame, which undergoes significant modifications due to the replacement of the conveyor belt. This structure had been properly designed and built considering a load from the conveyor belt trans-porting lignite. A new belt conveyor is used, which is wider than the previously installed unit, so the total weight of the belt conveyor increases. There are some opportunities to install the new conveyor belt in the existing bridge frame when the moving belt rests on the underlying conveyor belt support frame or suspended hanging roller sets from beams installed mounted in the middle of bridge columns. In this article, the purpose was to investigate the latter installation and compare it with the usual underlying conveyor support frame in terms of natural frequencies. The different installations of the conveyor belt system on the bridge frame cause different load conditions. The forces transferred to the beams of the 3D structure play a crucial role in the structural integrity and the magnitude of natural frequencies.

Our modal analysis provided more information and calculated components of displacements and the magnitude of natural frequencies 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 natural frequencies also prove that the two load conditions from the new conveyor belt installations are not dangerous. 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 dynamic response analysis to obtain the forced response on the bridge frame structure caused by the non-linear vibrations from the transporting belt and roller sets.

REFERENCES

[1]. **I. Mitran, F.D. Popescu, M.S. Nan, S.S. Soba,** *Possibilities for Increasing the Use of Machineries Using Computer Assisted Statistical Methods,* WSEAS Transactions on Mathematics, Issue 2, Volume 8, February 2009.

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

[3]. **Szirbik S., Virág Z.** *Finite element analysis of a steel bridge frame for belt conveyors*, Geosciences and Engineering: a Publication of the University of Miskolc, 11(1), pp. 114-119. (2023)

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

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

[6]. **Popescu F. D., Radu S. M., Kotwica K., Andraş A., Kertesz Brînaş I., Dinescu S.** *Vibration analysis of a bucket wheel excavator boom using rayleigh's damping model,* New Trends in Production Engineering, 2 (1), pp. 233-241., 2019.