K. SEMRÁD, D. MARASOVÁ, Ľ. AMBRIŠKO, Ľ. CABAN, M. SPODNIAK

ISSN 0543-5846
METABK 59(3) 385-388 (2020)
UDC – UDK 656.96:629.1-8-621.231:518.1:167.7:531.01=111

NUMERICAL AND EXPERIMENTAL ANALYSIS OF INNOVATIVE SUPPORT SYSTEM OF BELT CONVEYOR

Received – Primljeno: 2019-12-11
Accepted – Prihvaćeno: 2020-02-25
Preliminary Note – Prethodno priopćenje

The article processes an issue of the strength of a support system of a belt conveyor applying the numerical and experimental analysis approach. The support system of a belt conveyor is during the operation exposed to high impact forces due to the impact of the foreign objects. The methodology of the numerical analysis is based on the finite element modelling and according to the experimental analysis the boundary conditions are estimated. The results obtained using the numerical analyses are identical to the results from the experimental analysis. Maximal damage of the support system of a belt conveyor is estimated and according to the proposed methodology verified by the comparison of the experimental and numerical results, the design and optimization of the support system for the practical operation can be performed.

Key words: transport, conveyor belt, numerical analysis, experimental analysis, finite element method (FEM)

INTRODUCTION

Tracked traffic with a closed conveyor belt is of a global importance, from the point of view of the transport of materials over the long distances of large transport capacity. The main problem in so complex construction of the conveyor belt is the operational reliability. When transporting materials, there is a destruction of the conveyor belt (the impact of large-sized heavy-weight material, sidelines from the transport route) and particularly for this reason, it is necessary to analyze, test and acquire new knowledge about the reliability of the conveyor belt operation. An integral part of the transfer point is the support system. The methodology based on the experiment results and numerical simulations involving the possibility of the impact of the support systems on the belt conveyor operation, damage and even destruction was therefore created and is presented in the article.

Impact of the material on the conveyor belt is an ongoing subject of scientific and research activities, in order to increase the lifetime of the conveyor belt construction. In order to estimate material and mechanical properties large number of experiments and simulations needs to be carried out [1].

EXPERIMENTS

Belt conveyor support system test

The usage of impact hammer elements (impactor) means better resilience of impact energy, elimination and reduction of dustiness compared to the rollers. The advantage of the impactors is that they offer strong support and protection of the belt in the impact area. In order to verify resistance to the impact it is important to carry out the experimental analysis. Two types of support bed elements are analyzed during the experiment. The different material characteristics of the various support systems can play a key role. In our case the comparison of support system beds based on the steel and aluminum are analyzed. Using the hoist a hammer weighing 105 kg was lifted to the selected impact height. From a given height the hammer is dropped freely on the test specimen. The impact force is recorded by strain gauges. The impact time as well as the reflection of the hammer is estimated. The speed of the hammer is assessed according to the formula for the kinetic and potential energy [2]:

$$m \cdot g \cdot h = \frac{1}{2} \cdot m \cdot v^2 \Rightarrow v = \sqrt{2 \cdot g \cdot h}$$

where m is the weight of the hammer, g is the gravitational acceleration, h is the height and v is the speed of the hammer.

The value of the maximal force during the testing is assumed according to the maximal bending of the tested specimen. The results from the tests can be seen in the Figure 1, where the maximal deformation was 36 mm.

Numerical simulations

In order to estimate the deformation for the dynamic analysis it is essential to carry out the static analysis and to calculate the static deformation. The 3D geometry model is created in the Creo-Parametric (Figure 2) and according to the geometry the finite element model is

K. Semrád (e-mail: karol.semrad@tuke.sk), M. Spodniak, Faculty of Aeronautics, D. Marasová, Ľ. Ambriško, Ľ. Caban, Faculty of Mining, Ecology, Process Control and Geotechnologies, Technical University of Kosice, Slovakia

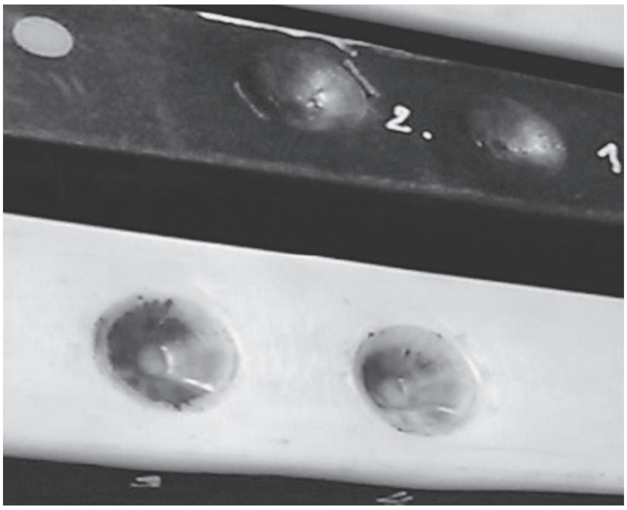

Figure 1 Tests results – max. deformation

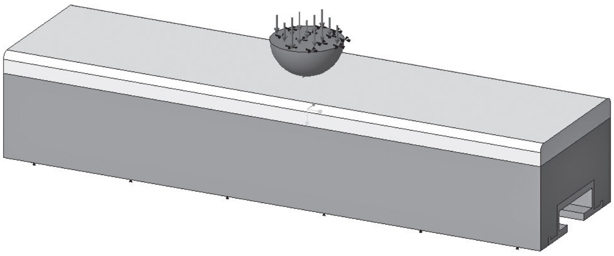

Figure 2 3D assembly CAD model

prepared. The material model is obtained from the tests and creates the basis for the properties definition for the purposes of the static analysis [3].

Static analysis

Based on the 3D model the finite element model is created and the static analysis is performed in the Creo-Simulate. For the boundary conditions definition the same parts of the assembly as in the real test are used. The impact hammer is placed in the middle of the impact rod where the force is applied with the magnitude F = 1 030,05 N in the Z direction of the global coordinate system. The estimated force is equal to the weight of 105 kg testing hammer.

The results of a static analysis are presented in the Figure 3. In this Figure the distribution of the deformation is clearly shown. The deformation is an essential input for the dynamic simulations because according to the static analysis results the dynamic impact factor can be calculated. The maximal static deformation has a value of y_{st} = 0,747 mm.

Simulation of test conditions

The maximal deformation is calculated for the static conditions as discussed in the previous chapter and according to this value it is possible to calculate the dynamic impact factor. Once there is a contact between the hammer and the specimen during the test, there is a

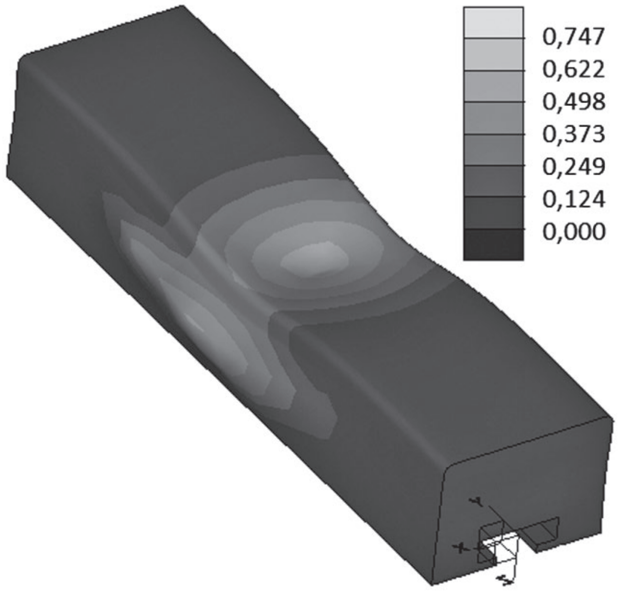

Figure 3 Static displacement / mm

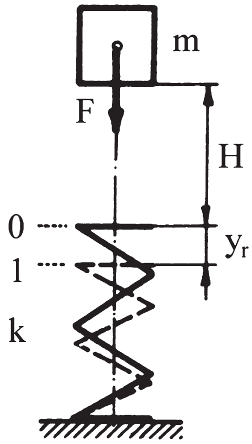

Figure 4 Elastic assembly

significant load change in the short time interval. The stresses reached during the experiment are therefore several times higher than the static stress in the assembly. In the described situation there is a change of kinetic energy into the potential energy of the elastic assembly (Figure 4) [4]:

$$F(H + y_r) = \frac{1}{2} \cdot k \cdot y_r^2 \quad \text{where} \quad \frac{F}{k} = y_{st}$$

which represents the static deformation y_{st} that is caused by the body ductility during the static loading of the assembly F and k is the stiffness of the flexible system. H is the height the hammer is dropped freely on the test specimen and y_r is the dynamic deformation. The final equation for the dynamic impact factor is:

$$\beta_r = \frac{y_r}{y_{st}} = 1 + \sqrt{1 + \frac{2H}{y_{st}} \cdot \frac{1}{1 + \alpha \cdot \frac{Q_1}{Q_2}}}$$

where α is the coefficient of the body mass reduction, which the impact absorbs in the place of the contact, Q_1

is the gravity of the body absorbing the impact and Q_2 is the gravity of the body creating the impact. Then:

$$F_r = F_{st} \cdot \beta_r,\ \sigma_r = \sigma_{st} \cdot \beta_r$$

where β_r carries the information about the ratio of the dynamic deformation y_r and static deformation y_{st}. When the y_{st} = 0,747 mm from the static FEM analysis is applied, the dynamic impact factor has a value β_r = 48,116 for the impact of the hammer from H = 2 000 mm. Once the dynamic impact factor and the static deformation are known, it is possible to calculate the deformation during the dynamic test y_r. Also it is possible to calculate the impact force F_r and the impact stress σ_r according to the dynamic impact factor. Applying the dynamic impact factor, the deformation calculated during the dynamic test is y_r = 35,92 mm while the impact hammer is fallen down on the conveyor rod from the height of 2 000 mm. The estimated force is 49 583 N. During the dynamic experiment (Figure 1) the measured deformation is 36 mm and the stress obtained during the test is 55,28 MPa.

Dynamic simulation

The simulation of the experimental measurement is in this case carried out using the dynamic coefficient. The assembly of the whole device used for the testing is created in the 3D software Creo-Parametric but in order to perform the dynamic analysis it is essential to simplify the model. For the purpose of the dynamic analysis the assembly with the impact road and the impact hammer is created (Figure 2). The diameter of the impact hammer is 50 mm. In the numerical analysis the impact hammer with the steel structure is studied. When the dynamic analysis is solved it is better to place the impact hammer near to the impactor hammer (Figure 2). In the case of the high speed impact analyses that are solved in the explicit dynamics it is optimal to place the impact hammer on the impact hammer surface. Such a placement reduces the solving time significantly and increases the accuracy of the solution. This type of a placement also ensures better definition of the contacts between the particular parts of the assembly. The contact needs to be defined in a way which simulates the real conditions as the friction and other nonlinear effects. One of the boundary conditions in the proposed analysis is the speed of the impact hammer defined just before the impact (Table 1). The impact speed v is estimated according to the previous formula [2].

The boundary conditions correspond to the experimental tests. The degrees of freedom of impact hammer were considered only in the Z direction, in which the translation of the impact hammer is performed.

Table 1 **Boundary conditions**

Force / kN	Height / mm	Speed / ms^{-1}
49,56	2 000	6,26468

Material properties are acquired from the material tests. For the dynamic analysis it is essential to define Young´s modulus, density and Poisson ratio for every material of the assembly. The impact hammer is placed 0,5 mm above the contact area between the impact hammer head and the support system of the belt conveyor. For the contact areas friction and other nonlinear properties are defined. The applied force in the Z direction is equal to 49,56 kN. The lower part of the support system is fixed as can be seen in the Figure 2 [5].

RESULTS AND DISCUSSION

The dynamic analysis was solved in the ABAQUS software and the results are presented in the form of comparison of the experimental tests with the simulation results. The experimental results are presented in the Figure 1, where the deformation with the magnitude of 36 mm caused by the impactor can be clearly seen. The simulation results are presented in the Figure 5, where the maximal deformation is approximately 34 mm.

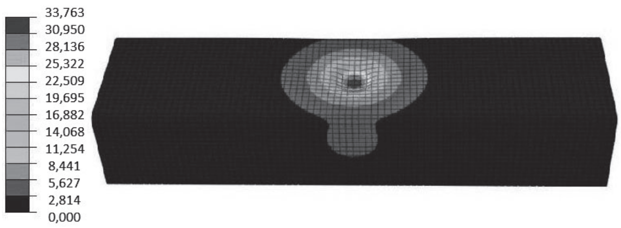

Figure 5 Dynamic displacement / mm

The estimated relative error 6,322 % could be caused by the uncertainty of the measurements and by the idealization of the material characteristics during the calculations. Results of experimental tests and calculations are summarized in the Table 2.

Table 2 **Experimental test and analysis results**

	Test	Calc.	FEM
Weight / kg	105	105	105
Height / mm	2 000	2 000	2 000
Force / kN	46,56	49,56	49,56
Def. / mm	36	35,92	33,763

CONCLUSION

The main purpose of the proposed research is to experimentally and numerically asses the belt conveyor support resilience to the mechanical impacts during the operation. Conveyor belts are used in various industry sectors and since they are one of the most expensive parts of the system, it is important to know their mechanical properties. According to the presented results it will be possible to design a convenient belt with required properties. In the article the experimental results and finite element analysis are presented and compared. The difference in the maximal deformation was only

6,322 %. The obtained results are highly valuable not only for research but also for the practical applications, because according to the presented methodology it will be possible to design new conveyor belts.

Acknowledgements

This work was supported by the APVV-18-0248, VEGA 1/0429/18 and the VA No. 019/2019/1.1.3/OP-VaI/DP with the ITMS code 313011T557 projects.

REFERENCES

[1] Marasová, D., Cehlár, M., Ambriško, Ľ., Taraba, V., Stařičná, N., Innovations in Monitoring Conveyor Belts with Implemented RFID Technology, 4th International Innovative Mining Symposium, Bristol, Édition Diffusion Presse Sciences (2019), 1-7.

[2] Šmelko, M., Praslička, D., Draganová, K., Lipovský, P., Kán, V., Bajús, J., Wireless strain gauge for composite materials, UPB Scientific Bulletin, Series D, Mechanical Engineering, 78 (2016) 1, 59-66.

[3] L. Novotný, H. F. G. Abreu, J. Zajac, M. Béreš, Imperfection generation in finite element models of welding, Science and Technology of Welding and Joining, 23 (2017) 2, 148-157.

[4] Főző, L., Čerňan, J., Cúttová, M., Ratkovská, K., Adamčík jr., F., Effect of TBC on heat transfer into nickel based refractory alloy, SAMI 2017, Danvers, IEEE (2017), 435-438.

[5] Jakel, R., Analysis of Hyperelastic Materials with MECHANICA - Theory and Application Examples, Presentation for the 2nd SAXSIM, Technische Universität Chemnitz (2010).

Note: The English Language translation was done by Katarína Draganová, PhD., ING-PAED IGIP, Technical University of Kosice, Slovakia