Methodology of the Impact Process Applying the Finite Element Method

Ľuboš Caban, Daniela Marasová, Ľubomír Ambriško

Technical University Košice, Faculty BERG, Institute of Logistics and Transport, Letná 9, Košice, Slovakia

Abstract – The economy of material conveyance by pipe conveyors is largely affected by high costs of a conveyor belt and the support system of a conveyor. Even though the operational properties of a conveyor belt are taken into consideration, destruction of the top cover layer and punctures of the whole belt occur rather frequently. The most frequent cause of such damage to a conveyor belt is the impact load at critical sections of the transportation process. Such critical sections, where approximately 60 % of all the damage occurs, include chutes and filling stations. These sections are exposed to the elastic and plastic impact at the moment when the material of a certain weight falls down from a certain height (the maximum allowable impact height is 2.0-2.5 m). The magnitude and the direction of the impact force depend on the structural design of the chute and on the type of the support system. The present article deals with the application of the Finite Element Method (FEM) within the study of the dynamic effects developed at the impact of a load onto the impact bar.

Keywords – Mathematical modelling, FEM, impact bars, breakdown resistance.

1. Introduction

The strategy applied within the management of certain processes is based on the available information on such processes, such as the process inputs and outputs. In the case of a mining process, the conveyance technology belongs to the key inputs [1]. The selection of an optimal conveyance technology in the mining process, while applying the multi-criteria assessment, is described in the paper by authors [2]. In practice, the technology that is most frequently used for the coal conveyance is the belt conveyance system [3]. When assessing the quality of the belt conveyance, several aspects should be considered, including the dynamic load at the conveyor belt filling station. A conveyor section exposed to the load is particularly the conveyor belt and the impact bed. A conveyor belt and the support system may only be used in the operation if they meet certain criteria for mechanical and physical properties. The structure of a conveyor belt facilitates elimination of the impact load produced by the fall of the transported material. Depending on the conditions of the conveyor belt deployment, different requirements for mechanical and physical properties apply [4].

It is assumed that the application of the FEM, based on the numerical simulation, will in this case facilitate an accurate analysis of the strain distribution over the load area. This method will facilitate identification of the area where the strain is concentrated or where the load has negative effects. This method may be regarded as an important extension and supplementation of the analytical method of deformation identification [5].

Impact beds are designed to provide better conveyance and capacity conditions and their purpose is to minimise the conveyor belt damage [6]. A novel type of the impact bed structure consists of impact bars of various damping degrees. It is a composite that may be subjected to strain and deformation analyses of mechanical properties.

Mathematical modelling facilitates the analysis of the processes running at the material's impact point and of the resulting strain and deformation [6], [7]. The data obtained by the simulation facilitate a

reliable comparison of the results, for example, of the quality control which can be performed better if numerical methods are applied, in particular the Finite Element Method. It is advisable to use the numerical analysis through the Finite Element Method (FEM); it is the most frequently used method of the computer-assisted calculation and the strength analysis [8].

An impact bed with impact bars is a rather novel component and the designing method constantly represents the subject of the developmental efforts [9]. An important advantage of this solution is a shorter period of material stabilisation due to the absence of elements which might cause rotation of individual pieces of the material [10]. This is also related to the conveyor belt wear caused by the transported material. Impact beds are manufactured according to the specific requirements for a belt conveyor structure [11], [12].

Due to the damage to conveyor belts by punctures as well as the high costs of conveyor belts, more and more experts are now studying the issues regarding support systems in belt conveyance systems [13], [14]. To a large extent, such damage is a result of inappropriate support bed which significantly shortens the belt's service life. Papers [15], [16], deal with the theoretical aspects of the dynamics of an impact bed with impact bars and present the results of operational experiments.

2. Analysis of inputs for the dynamic effects modelling

The verification of the results obtained through experimental measurements may be carried out by comparing them to the results of a simulated calculation while using a simplified model and applying the Finite Element Method (FEM) [17]. In practice, there are several state-of-the-art software products to be used for the FEM (ABAQUS, ANSYS, MARC, PATRAN and NASTRAN); they are based on the latest knowledge of the computation problems in the field of mechanics. These software products are rather expensive, that is why in Slovakia the limited-scope application programs are used in practice [18]. The software that was primarily selected to be used for the model verification and for obtaining the output parameters was the ANSYS WORK BENCH.

In order to create a mathematical model of an impact bar, it is necessary to know the properties determined by its structure. An impact bar consists of several layers with different material properties; it is a composite material. Rubber, a hyperelastic material, is one of the materials used in the manufacture of impact beds and impact bars. Behaviour of rubber is very nonlinear and it cannot

be easily characterised; that is why the mathematical modelling of rubber parts is a complicated process. An impact bar also consists of the reinforcing elements which affect its strength at certain points; the identification of the properties of the whole unit is therefore more complex. In the mathematical modelling it is necessary to know the basic characteristics, such as a flexibility module, the Poisson ratio, and the material density [19].

2.1 Explicit dynamics in the ANSYS software

The simulation of rapid processes running in the structures is carried out using the software products like ANSYS EXPLICIT, ANSYS AUTODYN and ANSYS LS-DYNA. They are based on the explicit method of solving motion equations. These software products are primarily used in crash tests, drop tests, simulations of forming, pressing, and cutting materials, simulations of penetration of gun bullets, simulation of the explosion effects on the surrounding structures, etc.

ANSYS EXPLICIT

- An universal explicit simulation software, fully integrated in the ANSYS WORK BENCH environment.
- The working environment is identical to the ANSYS MECHANICAL software.
- Typical applications include drop tests, simulation of impacts of impactors, simulation of forming and simulation of damaging the materials at high loading rates.

ANSYS AUTODYN

- An advanced explicit simulation software for modelling the nonlinear dynamics of solid substances, fluids, gases and their interactions, fully integrated in the ANSYS platform.
- Typical applications include simulated resistance tests of armours in armoured vehicles during explosions of mines, grenades and flying bombs.
- Other applications include simulations of behaviour of vessels and containers filled with fluids at falls and impacts.
- The software contains a rich library of materials.

ANSYS LS-DYNA

- A complex computation module, fully integrated in the ANSYS APDL and partially (only the most frequently used tools) integrated in the ANSYS

WORKBENCH platform. This facilitates the use, for example, of the ANSYS DESIGN MODELER for the preparation of geometry, the ANSYS MESHING for the discretisation of geometric models, and the ANSYS PARAMETERS for defining the optimisation analyses. The results may be visualised in the ANSYS MECHANICAL graphical environment or in the LS-PRE POST software which is a part of every installation.

- The settings of the analyses may be adjusted using the ANSYS WORKBENCH graphical environment or using the text commands.
- Typical applications include forming, pressing and cutting of materials, simulation of the impact of impactors, simulation of impacts and falls of various structures, simulation of damaging the materials at high loading rates.
- Suitable for special and research applications [20].

3. Preparation of the model geometry

An impact bar specimen, made according to the manufacturer's specifications, was subjected to the tests for the purpose of the dynamic analysis. In the preparation of the model geometry, it had to be simplified due to a prolonged duration of the calculation of explicit dynamics in the ANSYS software [21], [22]. The analysis was created using the EXPLICIT DYNAMICS module (Figure 1.). The higher the number of elements in the set was and the more accurate their shapes were, the more time was required for the calculation. It is important to preserve the key elements in the set which are of high explanatory power.
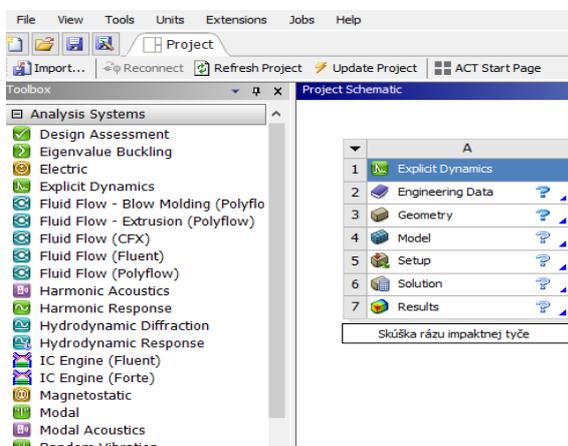

Figure 1. Explicit Dynamics module

For the purpose of simulating the impact of the material with the mass of 105 kg onto an innovative impact bed, an impactor weighing 105 kg was created in the DESIGN MODELER environment (Figure 2.). The impactor head was spherical. It simulated the shape of the transported material (sharpness of edges). In the set with a spherical head, the distance between the impact bar and the load head was 500 mm. The impact bar simulated the support point of a conveyor belt. In practical operations, there are various types of bar structures (various strengths, dimensions and structural designs). For this particular case, a standard impact bar was simulated, free of the reinforcement. The support system simulated various types of impact beds (steel rollers, impact sets, etc). In the simulation, the impact bar was firmly attached to the supporting frame.
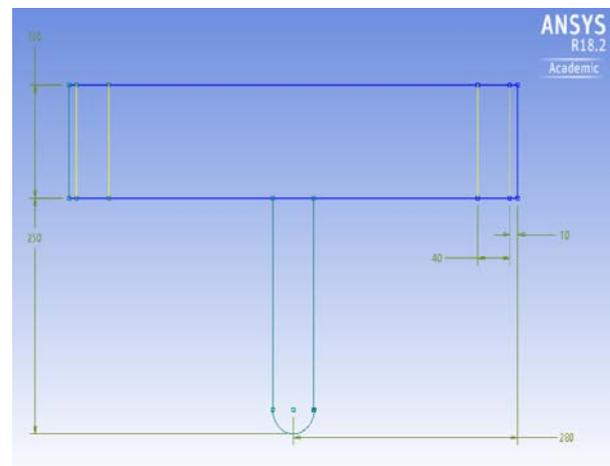

Figure 2. Dimensions of the load with a spherical head

3.1 Analysis creation in the ANSYS environment

The construction of the defined elements of the set was followed by the creation of the analysis. The first step consisted in defining the material characteristics in the ENGINEERING DATA section. The first defined material was STRUCTURAL STEEL; this type of material was assigned to the impactor (105 kg) and the guide bars (Figure 3.).
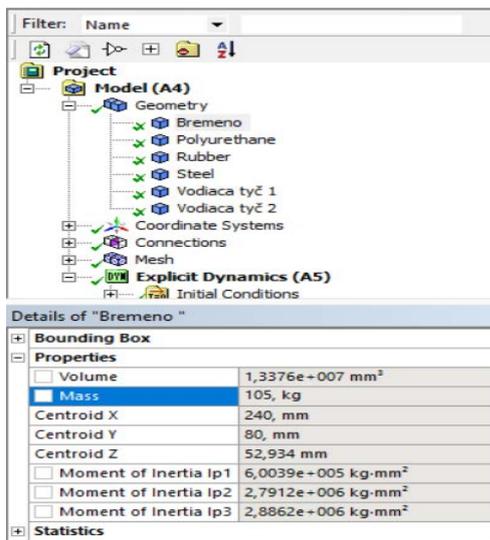
Figure 3. Image of the load's mass depending on the steel density

The lower reinforcement section of the impact bar was also assigned the steel, with the adjusted Poisson ratio, density, and elasticity module (Figure 4.).

Rubber was chosen from among EXPLICIT MATERIALS within the ENGINEERING DATA SOURCES options by selecting RUBBER. This type of material is present in the central part of the impact bar and it has the damping effects.

Figure 4. 1Example of defining the material characteristics – Structural Steel

The last defined material was polyurethane which is present in the upper part of the impact bar. It ensures high abrasion wear resistance and strength. Out of the library of materials, we chose POLYURETHANE and defined the properties of the material.

Subsequently, it was possible to access the MECHANICAL module in the project scheme.
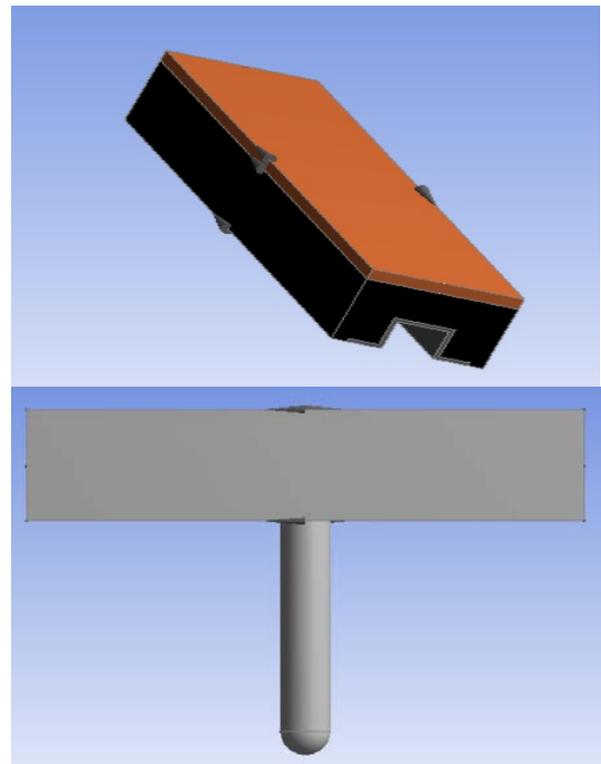

Figure 5. Elements of the 3D dynamic analysis – the impact bar and the drop hammer

After selecting the GEOMETRY item in the project scheme, the material was changed for those set elements for which it was required. For all the elements, the standard material, STRUCTURAL STEEL, was selected. In the MATERIAL section, in the ASSIGNMENT option, a list of defined materials was then displayed in the library. The top layer of the impact bar is made of POLYURETHANE. By selecting this material it is assigned to the given part. For the central part of the impact bar, RUBBER 1 was chosen. For other elements, STRUCTURAL STEEL was defined as the material (Figure 6.).
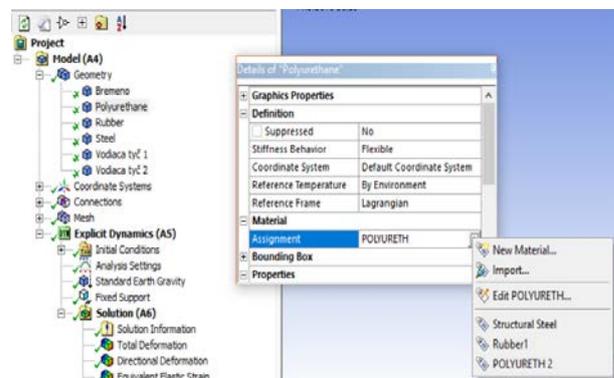

Figure 6. Assigning materials to the set elements

After choosing the CONTACTS item, it was necessary to remove all contacts between the elements, excluding the contacts between the impact bar layers, because it is a composite material (Figure 7.). A connection between the upper part, POLYURETHANE, and the damping central part, RUBBER 1, had to be preserved. The other connection was between the central part, RUBBER, and the lower STEEL reinforcement. Both types are defined as BOUNDED FIXED.
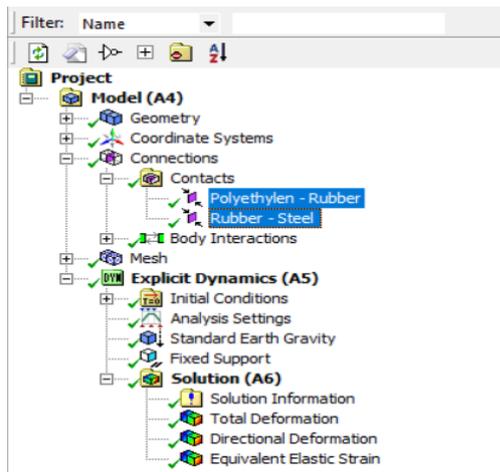

Figure 7. 2 Contacts between the elements

Subsequently, it was possible to define the mesh of finite elements of the model. Through right-clicking on the MESH item in the SIZING option we defined the MESH for individual set elements. The sizes of the meshes for individual elements are listed in Table 1.

Table 1. Defining the meshes for individual analysis elements

ELEMENT	MESH	MESH SIZE
IMPACT BAR	BODY SIZING	2 mm
GUIDE BARS	BODY SIZING	40 mm
LOAD HEAD	FACE SIZING	10 mm
LOAD BODY	BODY SIZING	40 mm

The mesh was generated by right-clicking on the MESH item, the GENERATE MESH option. The generating was applied to a rather high value in order to verify the functionality of the given analysis while considering a limited performance of the computer hardware.
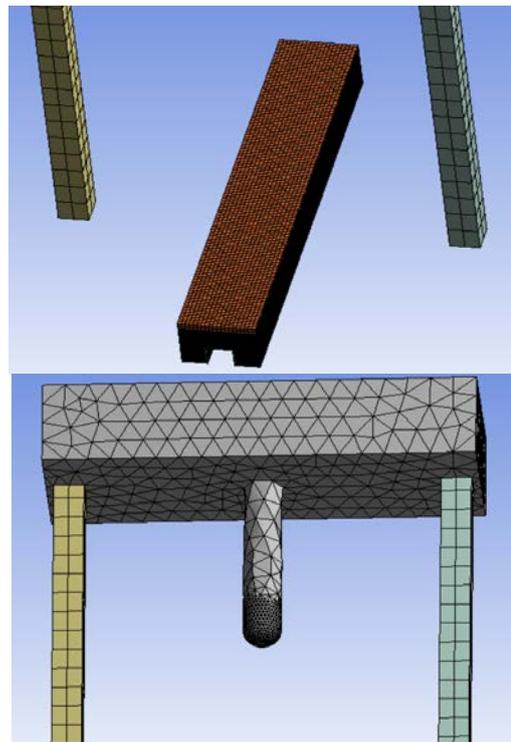

Figure 8. Application of the MESH

In the ANALYSIS SETTINGS item it was necessary to define the analysis duration, END TIME, for the timeline of analysing the impact of the load with a spherical impactor onto the impact bar. Through the MAXIMUM ENERGY ERROR option we determined the value of 1. If a much higher value was determined, the analysis would continue even after puncturing the entire structure of the impact bar; this, however, was not necessary.

The boundary conditions of the model were defined through the EXPLICIT DYNAMICS item (Figure 9.) and gravitation through the STANDARD EARTH GRAVITY item. The gravitation direction was defined by selecting a direction. The gravitation direction may also be defined in a dialogue box – a negative direction of the Z axis, according to the original coordinate system. Other directions, Y and X, had zero gravitational acceleration.
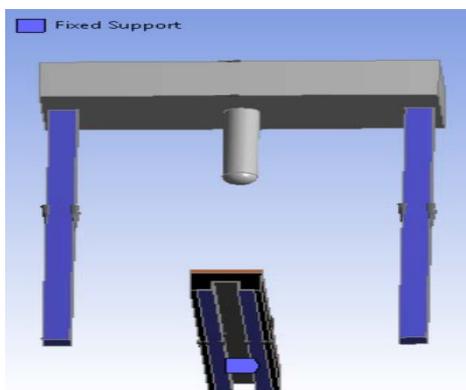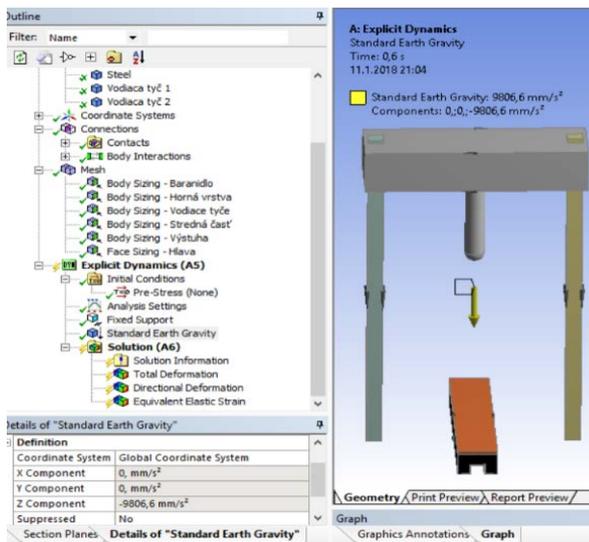
Figure 9. 3 Defining the boundary conditions

The next task was to fix the impact bar. Through the EXPLICIT DYNAMICS item we entered the fixation of the set elements in the model as FIXED SUPPORT. The purpose was to limit the degree of freedom in certain directions. For the reinforcing part of the impact bar and guide bars of the impactor all the degrees of freedom in the X, Y, and Z directions were disabled – a fixed placement.

Once the boundary conditions were defined, we could proceed with the dynamic analysis. Through the INSERT item, the SOLUTION option, we chose the solution of the tasks – TOTAL DEFORMATION, EQUIVALENT STRAIN and EQUIVALENT ELASTIC STRAIN. The following step enabled the initiation of the solution of the given task through the SOLVE option. During the calculation, it was possible to identify the estimated duration of the analysis calculation. The analysis duration depends on the complexity of the given task and on the performance of the computer.

By choosing the OUTPUT CONTROLS option it was possible to select the method of recording the analysis results. The preset method was the storage of results in 19 points – EQUALLY SPACED POINTS. By increasing the number of result points they may be condensed in order to capture the

puncture or the bounce of the load away from the impact bar and ensure the continuity of the analysis results [21].

Due to a long duration of the analysis calculation by means of explicit dynamics, the model geometry had to be simplified. By using a larger mesh for individual set elements we could also reduce the duration of the calculation; this, however, could result in decreased accuracy of the analysis. The main problem within the creation of an analysis was the lack of information on accurate material characteristics. We could use only a limited amount of necessary material properties. For the comparison purposes, it would be more appropriate to use other up-to-date computer software products that use the FEM analysis (ABAQUS or PATRAN). To make it more accurate, it is necessary to have detailed information on the properties of individual components. Following the proper defining, also other effects on the analysis could be supplemented, such as abrasion etc.

4. Conclusion

The purpose of the proposed methodology for the simulation of the material's impact onto the impact bar was to describe the procedure of the creation of the FEM in the available ANSYS WORKBENCH software.

One of the potential methods of obtaining more accurate knowledge of these processes is to apply mathematical modelling while using available software products. Mathematical modelling may be used to analyse the processes running at the location of the material's impact, as well as the produced strain and deformation.

The analysis and assessment of the measurements of the dynamic load of the special support system using the available software products, aimed at obtaining relevant results, is very important for the development of the discussed equipment.

It is therefore very important to innovate such equipment and subject it to testing, as well as pursue further development and structural improvements thereof.

References

[1] Markulik, Š., Cehlár, M., & Kozel, R. (2018). Process approach in the mining conditions. *Acta Montanistica Slovaca, 23*(1) 46-52.

[2] Grujić, M., Malindžak, D., & Marasova, D. (2011). Possibilities for reducing the negative impact of the number of conveyors in a coal transportation system. *Tehnički vjesnik*, *18*(3), 453-458.

[3] Andrejiova, M., Grincova, A., Marasova, D., & Grendel, P. (2015). Multicriterial assessment of the raw material transport. *Acta Montanistica Slovaca*, *20*(1), 26-32.

[4] Apalak, Z. G., Apalak, M. K., & Davies, R. (1996). Analysis and design of tee joints with double support. *International journal of adhesion and adhesives*, *16*(3), 187-214.

[5]. Godzimirski, J., Kozakiewicz, J., Łunarski, J., & Zielecki, W. (1997). Konstrukcyjne połączenia klejowe elementów metalowych w budowie maszyn. *Oficyna Wydawnicza Politechniki Rzeszowskiej, Rzeszów*.

[6]. DVB-AF s.r.o.: Hradec nad Moravicí. Retrieved from: https://www.flexco.cz/pdf-soubory [accessed: 02 October 2018].

[7]. Bindzár, P., & Grinčová, A. (2007). Príspevok k modelovaniu prvkov s hyperelastickými vlastnosťami. In: Doprava a logistika. č. mimoriadne, 4 s.

[8]. Marasová, D., Ambriško, Ľ., Andrejiová, M., & Grinčová, A. (2017). Examination of the process of damaging the top covering layer of a conveyor belt applying the FEM. *Measurement*, *112*, 47-52.
[9]. Quek, M. Y. (2004). Analysis of residual stresses in a single fibre–matrix composite. *International journal of adhesion and adhesives*, *24*(5), 379-388.

[10]. Klimecký, O., Veverková, H., Bailotti, K., & MŰLLER, J. (1988). Manipulace s materiálem: doprava v lomech. *Vydání. Ostrava. VŠB v Ostravě.*, s. 320.

[11]. Boháč, L. (2010). Přesypová stolice B 1200 mm. DVB-AF s.r.o., Opava, 18 s

[12]. Gondek, H., & Neruda, J. (2014). Wyniki oceny żywotności stacji przesypowej z prętami udarowymi na wysięgniku załadowczym kołowej koparki kubełkowej, Mezinárodní konference Komeko, Institut technikigórniczej Komag, Gliwice, 417 – 425.

[13]. Gondek, H., Neruda, J., & Plchová, A. (2012). Nové řešení přesypových stanic pásových dopravníku v hlubinných dolech Mezinárodní konference KOMAG, Rytro, Institut Techniki Gorniczej.

[14].Wadham-Gagnon, M. (2008). *Hyperelastic Modelling of Rubber Behaviour in Finite Element Software*. ProQuest.

[15]. Gondek, H., Neruda, J., & Pokorný, J. (2014). The dynamics of impacts tools the loading boom bucket wheel excavators. In *Applied Mechanics and Materials* (Vol. 683, pp. 213-218). Trans Tech Publications.

[16]. Marasová, D., Ambriško, Ľ., Grinčová, & Andrejová, A. (2015). Testovanie rázovej odolnosti progresívnych tlmiacich komponentov. Vedecko výskumná správa, FBERG, Košice.

[17]. Ambriško, Ľ, Marasova, D, & Grendel, P. (2016). Determination the effect of factors affecting the tensile strength of fabric conveyor belts. *Eksploatacja i Niezawodnosc – Maintenance and Reliability, 18*(1), 110–116.

[18]. Neruda, J. (2016). Měžení dynamiky dopadové stolice s impaktními tyčemi na pásovém dopravníku rypadla KU 300. Doktorská dizerační práca. VŠB Ostrava.

[19]. Neruda, J., (2015). Výsledky posouzení životnost přesypové stanice s impaktními tyčemi na nakládacím výložníku kolesového rypadla. Presentace doktorandů katedry 340, VŠB-TU Ostrava, 2015.

[20]. SVS FEM s.r.o.: Produkty firmy ANSYS Inc. Retrieved from: https://www.svsfem.cz/content/produkty-firmy-ansys-inc [accessed: 07 December 2018].

[21]. TechSoft Engineering s.r.o. Explicitní dynamika. Retrieved from: http://www.techsoft-eng.cz/ansys-2/strukturalni-analyzy/explicitni-dynamika/ [accessed: 10 February 2019].

[22]. ANSYS, Inc.: Explicit dynamics. Retrieved from: http://www.solidcad.co.za/WebsiteAssets/pdf/brochures/ansys-explicit-dynamics-brochure-14.0.pdf [accessed: 05 February 2019].