Simulation of the impact bar top layer tensile test applying the Finite Elements Method

D Marasová¹, L Ambriško¹, L Caban¹ and K Semrád²

¹ Technical University of Košice, Faculty of Mining, Ecology, Process Control and Geotechnologies, Letná 9, 042 00 Košice, Slovak Republic
² Technical University of Košice, Faculty of Aeronautics, Rampová 7, 041 21 Košice, Slovak Republic

E-mail: daniela.marasova@tuke.sk

Abstract. Impact bars represent the main structural components of impact idlers which are used to absorb the impact of the transported material at chutes in belt conveyance systems. An impact bar is a composite comprising three basic structural parts. The top part consists of a polyethylene or polyurethane layer; the middle part is flexible rubber absorbing the dynamic impacts of the falling material; and the lower part is formed from the steel carrying profile. Mathematical modelling of the belt conveyance elements is one of the methods of obtaining relevant results regarding their dynamic load when being impacted by the falling material. The tensile test simulation, performed applying the Finite Element Method, was performed using the Abaqus 6.14 software. On the basis of the true laboratory experimental test, the tension diagram was created and subsequently used as the input for the mathematical model. For the purpose of examination of the plastic behaviour in the Abaqus environment, the logarithmic strain was used and the plastic load zone was demarcated. The elastic zone of relative deformation was determined by the modulus of elasticity. The present article contains the graphical interpretation of the separation of reversible and permanent relative deformations, model geometry, defined material characteristics and boundary conditions, as well as analysis results. The outcome is a high degree of concordance of the results of the laboratory experimental test and the results of the FEM analysis in terms of relative elongation at the breaking point and stress at the breaking point of the tested specimen of the impact bar top layer. The max. difference in the plastic deformation doesn’t exceed 10 %.

1. Introduction

Transport of materials of various types represents a very important part of technological processes across all industries. The efforts aimed at reducing the environmental impact of the transport are greatly facilitated by the use of enclosed belt conveyors forming a pipe (pipe conveyors) primarily intended for the transport of bulk, granular, or piece materials [1]. However, the operation of such conveyors is associated with a big problem – damage to rubber conveyor belts. The elimination of the damaging process is therefore subjected to the continuous research aimed at providing reliable operation thereof [2, 3]. Conveyor belts are mainly damaged as a result of the impact of the falling material. Chutes represent the most critical point of the transportation route where intensive conveyor belt wear and punctures are often observed [4]. These issues have been previously addressed not only in theory, but also through experiments [5, 6]. The impact idlers at chutes are formed of idlers with narrower spacing or garland idlers which possess only a limited ability to absorb the dynamic impact. The long-term
research and development contributes to the improved structure of impact idlers with impact bars and significantly better ability to absorb the energy of the falling material [7, 8]. Impact idlers are dimensioned with the aim to improve the transport and capacity parameters and hence minimise the damage to conveyor belts.

The problems associated with mathematical modelling of stress and strain of rubber composites were addressed within the research projects performed by several authors [9–12]. The modelling of rubber composites in terms of rubber behaviour has not been concluded yet because rubber is a specific material. The properties of rubber are diversified and depend on its composition and contents of special additives.

The purpose of the present article is to describe the simulation of the tensile test applying the Finite Element Method and the comparison of the mathematical model outcomes with the results of the experimental tensile test performed with the top layer of the impact bar. The geometric model of the impact bar is shown in figure 1. The key material that absorbs the dynamic impact is rubber.

![Figure 1. Geometry of the impact bar model.](image)

The strength and hardness of an impact bar is ensured by its reinforcing part. This model was designed with firm attachments between the top, middle, and carrying parts of the reinforcement.

2. Material and methods
The comparison of the results obtained by experimental measurements can be carried out using a simplified mathematical model, applying the Finite Element Method (FEM). The Finite Element Method is based on the division of the test specimen (blade) of the tested layer into the finite number of pieces, i.e., elements. The elements used in numerical simulations differ in their suitability for a particular simulation process. The software primarily selected for the model verification and obtaining the output values was the Abaqus.

The basic principle of the Finite Element Method is the simplification of the examined problem or task, i.e., segmentation of the specimen, or a set of specimens, into smaller elements. Such elements are then used to describe the physical behaviour of the given task (flow of fluids, stress-strength analyses). Eventual reassembly of the elements facilitates obtaining the resulting set of equations describing the behaviour of the tested system. The next step of the procedure is the application of initial and boundary conditions. The system defined as described above is subsequently used to obtain initially unknown parameters. Once the values of deformation displacement are determined, it is possible to identify the forces, stress, or other required parameters [10]. Each element is assigned certain material and physical properties. A specimen should therefore consist of simple elements in order to obtain the simplest possible mathematical description. From the physical point of view, each geometrical point of the system is assigned a value of the given parameter describing the distribution of such parameter around the relevant point. FEM is a numerical method used to solve partial differential or integral equations. Partial equations are solved using implicit or explicit algorithms [13]. Algorithms are selected depending on the type of the solved task (fast dynamic processes, impact load, static load, multi-physical tasks). Depending on the solved task, it is necessary to select the simulation software suitable for the task solution and verification.
2.1. FEM input data

The simulation of the tensile test applying the Finite Element Method was carried out in the Abaqus 6.14 environment. The results of the true tensile test were used to create a tension diagram which was then used in the mathematical model. The tensile properties of the top layer of the impact bar – the correlation between the stress (MPa) and the relative elongation expressed in (%) is shown in figure 2.

![Tension diagram of the impact bar top layer.](image)

The determination of the relative deformation requires knowing the physical dimensions of the specimens before the test (prior to deformation) and after the test (permanent deformation). It is the ratio of change in the length of the tested specimen to the original length of the tested specimen prior to the test. The external effects result in relative elongation of the specimen. The determination of relative deformation \( \varepsilon \) was calculated using the following formula:

\[
\varepsilon = \frac{\Delta l}{l_0} = \frac{l_u - l_0}{l_0}
\]

where \( l_0 \) is the specimen length prior to deformation (mm); \( l_u \) is the length after deformation (mm).

However, when the data on the plastic behaviour are entered in the Abaqus software, it is necessary to use the true or logarithmic strain. The description of the tension diagram through the logarithmic strain represents a correct description of the entire relative deformation zone. Such description takes into consideration additional material characteristics, e.g., strain rate and deformation hardening:

\[
\dot{\varepsilon} = \int_{l_0}^{l_u} \frac{dl}{l} = \ln (1 + \varepsilon)
\]

where \( \dot{\varepsilon} \) is the engineering or true strain; \( l_0 \) is the specimen length prior to deformation; \( l_u \) is the length after deformation; \( \varepsilon \) is the relative deformation.

After the tension diagram was described through the engineering relative strain, it was possible to identify basic material characteristics, such as the modulus of elasticity, and allocate the plastic load zone, i.e., the zone of permanent or irreversible deformation. The modulus of elasticity \( E \) was determined using the following formula:

\[
E = \frac{\sigma}{\varepsilon}
\]

where \( \sigma \) is the stress; \( \varepsilon \) is the relative deformation.
The modulus of elasticity may only be used to describe the elastic zone of relative deformation. The following Eq. (4) was used to separate the plastic from elastic deformation:

$$\varepsilon_{pl}^{RP0.2} = \left( \varepsilon^{tot} - \frac{\sigma}{E} - 0.002 \right) = 0$$

where $\varepsilon_{pl}^{RP0.2}$ is the relative deformation at conventional yield strength; $\varepsilon^{tot}$ is the total relative deformation; $\sigma$ is the stress; $E$ is the modulus of elasticity.

After all the values of stress and relative deformation were substituted to Eq. (4), it was possible to obtain positive and negative values of the relative deformation. The values lower than or equal to zero were assigned to the elastic zone and the values higher than zero were assigned to the plastic zone. The graphical interpretation of the separation of reversible and permanent relative deformations is presented in figure 3. Following such calculations, the material model may be used in the numerical analysis.

![Figure 3. Elastic and plastic deformation zones.](image)

2.2. Model geometry
The first step is to create a geometric description, or the geometry, of the tested specimen model (figure 4). The SolidWorks software was used to create a model of the specimen of the impact bar top layer while respecting the dimensions prescribed for the tensile test with the thickness of 15 mm. Specific points were demarcated on the model for the attachment of 55 mm long hydraulic jaws. Subsequently, these surfaces were separated and fixed contact points were defined, so that they behave as separate surfaces, for which the boundary conditions of the calculation were defined in the following steps of the procedure.

![Figure 4. Model of the tested specimen applying the FEM.](image)

2.3. Defining material characteristics
The Young’s modulus of elasticity was defined for the given geometry according to the above specified formulas. This was followed by defining the plasticity. The plastic load zone was defined using the user
plasticity. The calculated values were entered in the software in a table with prescribed relative deformation and relevant stress in true relative deformation.

2.4. Defining boundary conditions
On the left side of the model (figure 5), all degrees of freedom were removed so that the real experiment may be best described. On the right side, the displacement along the x-axis was defined. It is a stress inflicted on the specimen by the relevant force in the given direction. The calculation model was created without considering the thermodynamic effects or the local temperature elevation caused by plastic deformation. Defining the boundary conditions was followed by creating a mesh of finite elements and defining the type of elements suitable for the analysis. Abaqus contains a rich library of finite elements. Individual element types are described in the Results chapter hereof.

![Figure 5. Defining boundary conditions.](image)

3. Results
Mesh models – analysis results. When creating a mesh, it is important to select the optimal global size of finite elements and the number of individual nodes in order to facilitate the best possible description of the geometry of the tested specimen and hence ensure that the resulting design correspond the most to the experimental test. Also, the shapes of the model elements should be uniform as much as possible. The shape of the finite elements should be a hexahedron. Proper selection of the elements is crucial for the correctness of FEM calculations. The analysis was carried out with several element types – C3D8, C3D4 and C3D20R, depending on the performance of the given numerical analysis.

The C3D8 element was used in the first calculation. It is a standard three-dimensional continuous element with eight nodes and four integration points. The highest stresses (figure 6) that are formed at the given load were observed in the working zone. The stress was 0.801 MPa higher than the one observed in the real test.

![Figure 6. Tensile model – C3D8 element.](image)
The C3D4 element showed low resulting stress of 9.989 MPa in the working zone of the model and large local stress concentrators in the undesired load zone that affect the working zone. Subsequently, the calculation was made using the C3D4 element. It is three-dimensional finite element of the tetrahedron shape with four nodes and a single integration point.

This element is unsuitable for this type of nonlinear statics, as may be seen in figure 7, because the mesh of finite elements is very rigid. As may be noted, large local stress concentrators were formed. These stress concentrators were of considerable intensity and if a rupture of the component had been included in the predefined options, the rupture would have occurred at a different location than in the real test, which is unacceptable.

Subsequently, the analysis of the nonlinear static problem was carried out using the quadratic C3D20R element (figure 8). It is a three-dimensional square element with twenty nodes and a reduced number of integration points. This type of the finite element model is capable of providing a very good description of the nonlinear response of the material to the external load. The element of this type appears to be the optimal one because at the breaking point in the relative elongation and at the stress of 0.41 MPa it represents the difference of 0.53 %. The stress was evenly distributed in the longitudinal section of the model.

Figure 7. Tensile model – C3D4.

Figure 8. Tensile model – C3D20R.
4. Conclusion
The resulting graphs (figure 9) obtained from the tensile test (experiment) and from the analysis performed applying the Finite Element Method (FEM) are almost identical in the elastic zone. The yellow curve represents the real tensile test and the orange one represents the FEM calculation.

![Figure 9. Comparison of the analysis of the experiment results and the FEM results.](image)

The difference between the plastic deformation zone (experiment) and the plastic deformation (FEM) does not exceed 10%. The FEM model is more rigid than the real test. The outputs (table 1) are very well comparable and show a high degree of concordance.

|                  | Relative elongation at the breaking point (%) | Stress at the breaking point (MPa) |
|------------------|---------------------------------------------|----------------------------------|
| Real test        | 45.56                                       | 10.29                            |
| FEM calculation  | 46.09                                       | 10.70                            |
| Difference       | 0.53                                        | 0.41                             |

The simulation of the tensile test of the impact bar top layer performed applying the Finite Element Methods proved that the mathematic modelling may be used when developing novel structures of composites (impact bars). The outcome of the present article is the design of the tensile model of the top layer of the impact bar and the verification of the results obtained through the experimental research. The created tensile model will facilitate deeper investigation of the top layer behaviour when simulating the real conditions. The results obtained by the modelling are important especially for impact bar manufacturers. Tensile models will facilitate the selection of correct structure of the impact bar top layer when designing and developing novel structures with better ability to absorb the impact energy.

5. References
[1] Grujic M, Malindzak D and Marasova D 2011 Possibilities for reducing the negative impact of the number of conveyors in a coal transportation system Tehnicki Vjesnik-Technical Gazette 18 453–8
[2] Andrejiova M, Grincova A and Marasova D 2018 Failure analysis of rubber composites under dynamic impact loading by logistic regression Engineering Failure Analysis 84 311–9
[3] Grincova A, Berezny S and Marasova D 2009 Regression model creation based on experimental tests of conveyor belts against belt rips resistance Acta Montanistica Slovaca 14 113–20
[4] Honus S, Bocko P, Bouda T, Ristović I and Vulić M 2017 The effect of the number of conveyor belt carrying idlers on the failure of an impact place: A failure analysis Engineering Failure Analysis 77 93–101

[5] Czuba W and Furmanik K 2013 Analysis of a grain motion in the transfer area of the belt conveyor Eksplatacja i Niezawodnosc – Maintenance and Reliability 15 390–6

[6] Gondek H, Neruda J and Pokorny J 2014 The dynamics of impacts tools the loading boom bucket wheel excavators Applied Mechanics and Materials 683 213–8

[7] Gondek H, Neruda J and Pokorny J 2014 The dynamics of impacts tools with impact bars on transfer set place, the loading boom bucket wheel excavators Applied Mechanics and Materials 683 196–201

[8] Ambrisko L, Frydrysek K and Jelisavac Erdeljan D 2017 Experimental research of a new generation of support systems for the transport of mineral raw materials Acta Montanistica Slovaca 22 377–85

[9] Bindzar P 2006 Conveyor belt modelling by finite element method (in Slovak) Transport & Logistics 6 1–5

[10] Diani J, Brieu M and Gilarmini P 2006 Observation and modelling of the anisotropic viscohyperelastic behaviour of a rubberlike material International Journal of Solids and Structures 43 3044–56

[11] Mazurkiewicz D 2009 Problems of numerical simulation of stress and strain in the area of the adhesive-bonded joint of a conveyor belt Archives of Civil and Mechanical Engineering 9 75–91

[12] Marasova D, Ambrisko L, Andrejiova M and Grincova A 2017 Examination of the process of damaging the top covering layer of a conveyor belt applying the FEM Measurement 112 47–52

[13] Manual Abaqus. Available from: http://abaqus.software.polimi.it/v6.14/index.html

Acknowledgements
This work was supported by the Slovak Research and Development Agency under the contract No. APVV-18-0248 and with the support of projects VEGA 1/0577/17 and VEGA 1/0429/18.