DETERMINATION USING FEA OF THE STATIC STRESS OF A MINE HOIST DRUM AFTER SAFETY BRAKING

FLORIN DUMITRU POPESCU[1], ANDREI ANDRAŞ[2], ILDIKO BRÎNAŞ[3]

Abstract: Mine winding systems are a complex set of machines, devices, and mechanisms, which work together both to transport useful mineral substances or tailings from the mine to the surface or to other horizons, as well as to transport personnel, materials, and equipment, from surface to underground. The mine winding systems consists of the actual mine hoisting machine, the head-frame, cables (ropes), conveyances, auxiliary equipment, control and safety devices. The braking system can operate as service brake or safety brake. The service brake acts on the rope wheel to achieve the required tachogram, while the safety braking occurs independently of the operator's will, in the following situations: the lack of power supply, the decrease in the pressure of the hydraulic fluid, over winding of the conveyances, exceeding the maximum permitted extraction speed. A functional condition imposed on the brakes of a mine windings system is to ensure the immobilization of the conveyances after a safety braking. This paper proposes a study of the stresses and deformations to which the drive wheel drum of a single-rope friction mine hoist is subjected to, when it must keep the conveyances immobilized following a safety braking. To solve the mentioned problem, a virtual model of the drum of such an installation was created in SOLIDWORKS.

Key words: Mine hoist, emergency braking, force, rope, moment.

1. GENERAL ASPECTS REGARDING MINE HOISTS

The single-rope friction mine hoists are generally equipped with cages or skips and are intended for medium to long shafts. The admissible difference between the tensions of the two branches of the cables is limited to small values, by imposing the condition of non-slipping of the cable on the drive wheel friction material. Thus, the existence of the tail rope for balancing is mandatory (figures 1, 2 and 3). In figure 1, the directions of movement of the cables are highlighted.

[1] *Prof. Habil., Eng. Ph.D., University of Petroşani, fpopescu@gmail.com*
[2] *Assoc. Prof., Eng. Ph.D., University of Petroşani, andrei.andras@gmail.com*
[3] *Lecturer, Eng. Ph.D., University of Petroşani, kerteszildiko@ymail.com*

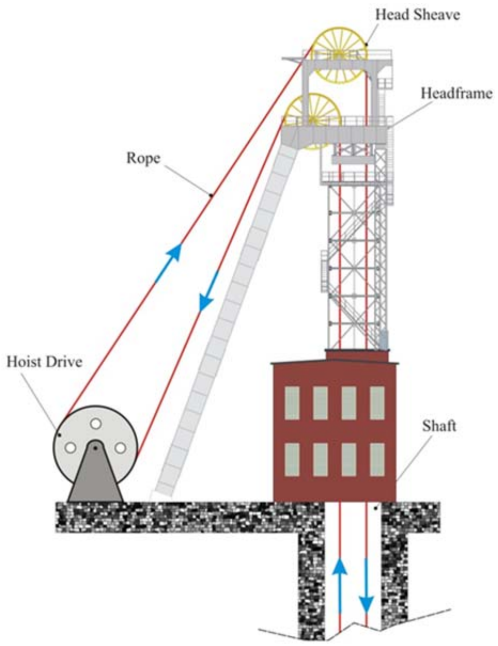

Fig.1 Multi-rope friction (Koepe) hoist with ground located drive wheel

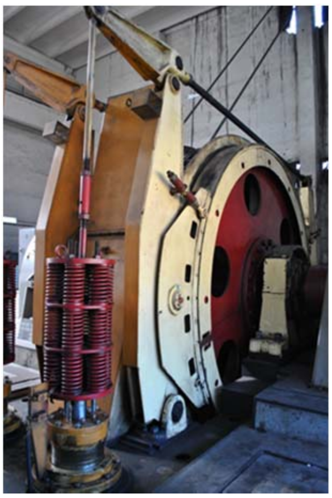

Fig.2 Drive wheel and drum braking system from a mine hoisting machine

Fig.3 The headframe of a mine winding machine

2. CONSTRUCTION OF THE MINE HOIST DRIVE WHEEL DRUM

To simulate the stresses to which the drive wheel friction drum of an mine hoist is subjected after a safety brake, a virtual assembly was built in SOLIDWORKS (figure 4), which is made up of the following component parts:

- the actual drum of the mine hoist;
- two supports for the drum;
- the rotation shaft of the drum;
- two brake discs.

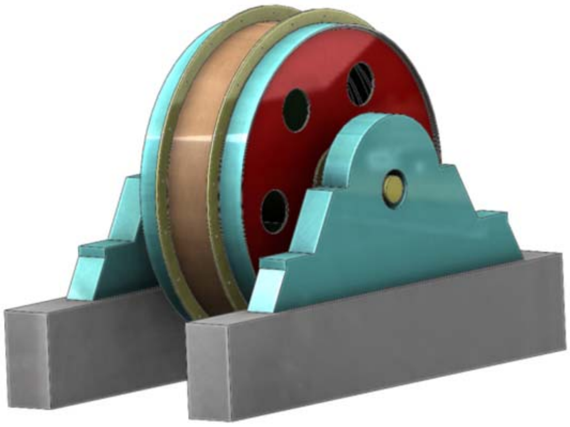

Fig.4 The virtual assembly of the drive wheel drums of the mine hoist

2.1 The actual drum of the mine hoist

The model and dimensions of the drum of are presented in figure 5. The material from which the drum model is built is AISI 304 Steel.

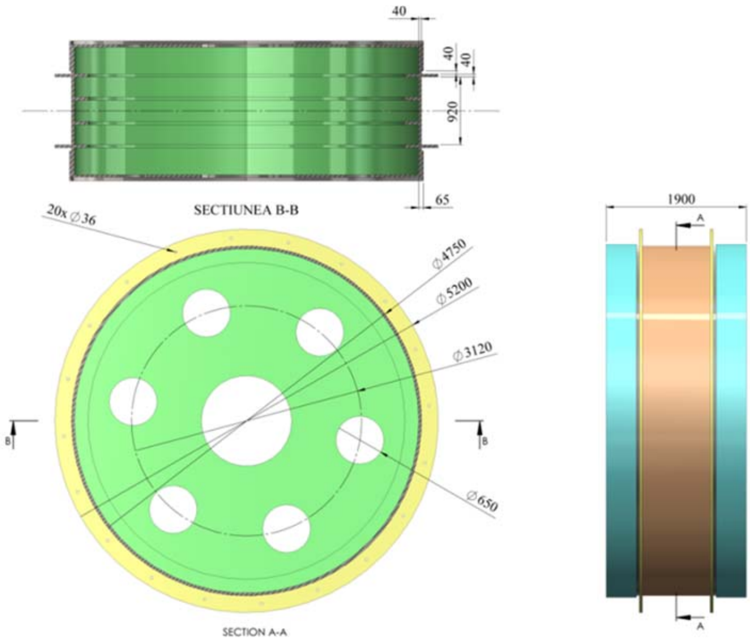

Fig.5. Model of the drum

2.2 The drum supports

Figure 6 shows the model and dimensions of the drum supports.

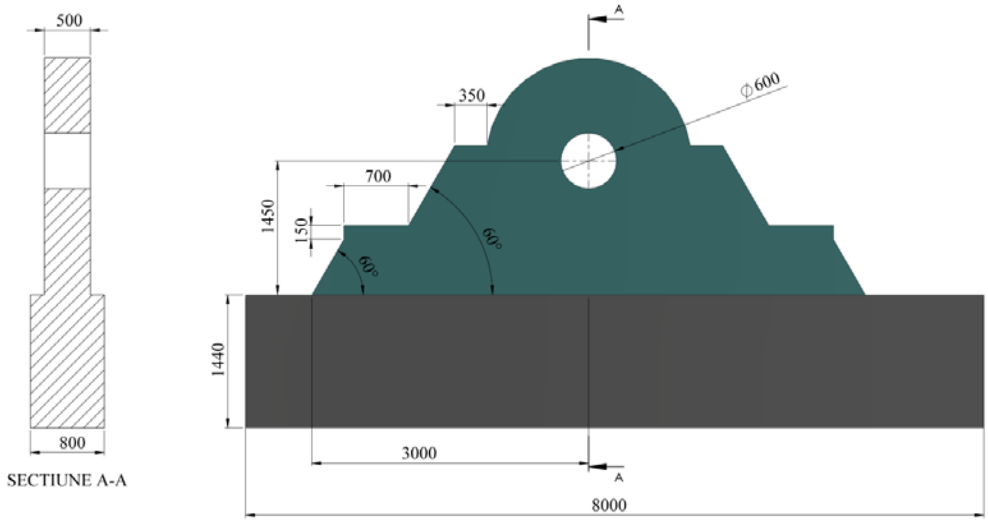

Fig.6 Model of the drum supports

2.3 The rotation shaft of the drum

Figure 7 shows the model highlights the dimensions of the rotation axis of the mine hoist drum.

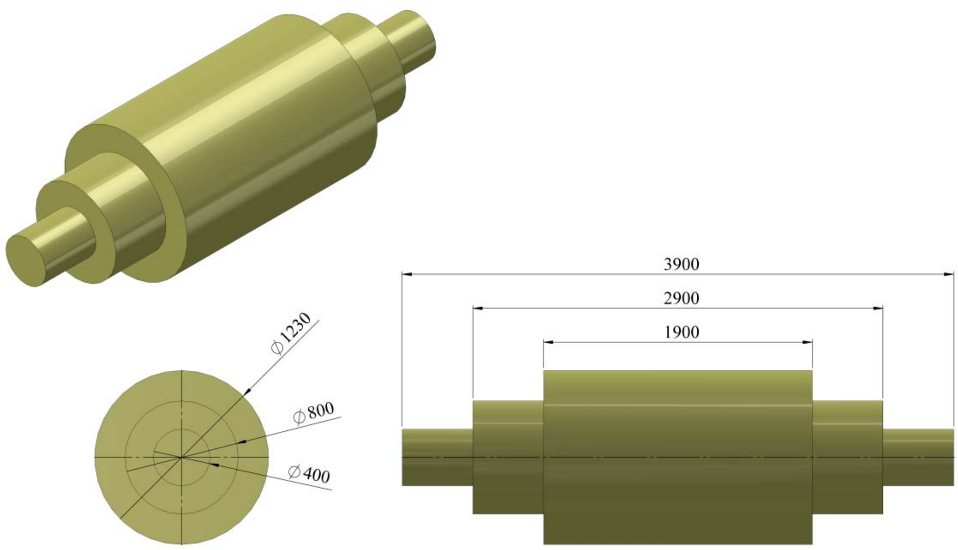

Fig.7 Model of the rotation shaft of the drum

2.4 Brake disc

Figure 8 shows the model and dimensions of the brake disc of the drum of the extraction machine.

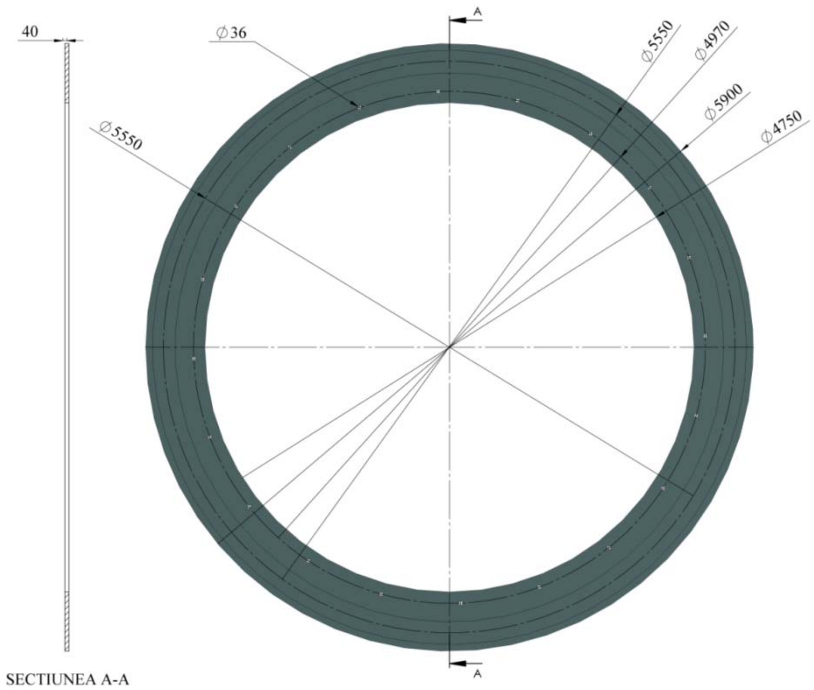

Fig.8 Model of the brake disc of the mine hoist drum.

2.5 Establishing the links between the parts of the virtual model

In order to create the virtual assembly of the mine hoist drum, certain links between its parts must be established. Thus, we used standard geometric connections of concentricity between the rotation shaft and the drum of the mine hoist on the one hand and between the rotation shaft and the mounting holes of the bearings on the two supports of the drum on the other hand. Coincidence links were also imposed between the extremities of the rotation shaft and the outer surfaces of the drum supports, as well as coincidence links of the frontal planes of the shaft, respectively the drum with the frontal plane of the assembly.

3. SIMULATION OF THE STATIC STRESS OF THE DRUM OF THE MINE HOIST WHEN THE PAYLOAD IS SUSPENDED IN THE SHAFT

The simulation of the stress acting on the drum in this case was done in SOLIDWORKS using the `Simulation` menu where a static analysis was performed.

3.1 Establishing the connections between the parts

The connection of the rotation shaft with the two supports of the drum is provided by two virtual bearings as seen in figure 9. In figure 10 we present the characteristics of connection by virtual bearings of the mentioned parts. The outer cylindrical surface of the rotation shaft and the inner cylindrical surface of the hole on the drum supports are specified as the placement entities of the two virtual bearings. We opted for self-aligning of the bearings, and their connection with the contact surfaces is rigid. At the same time, the initial compression (tightening) of the virtual bearings is zero.

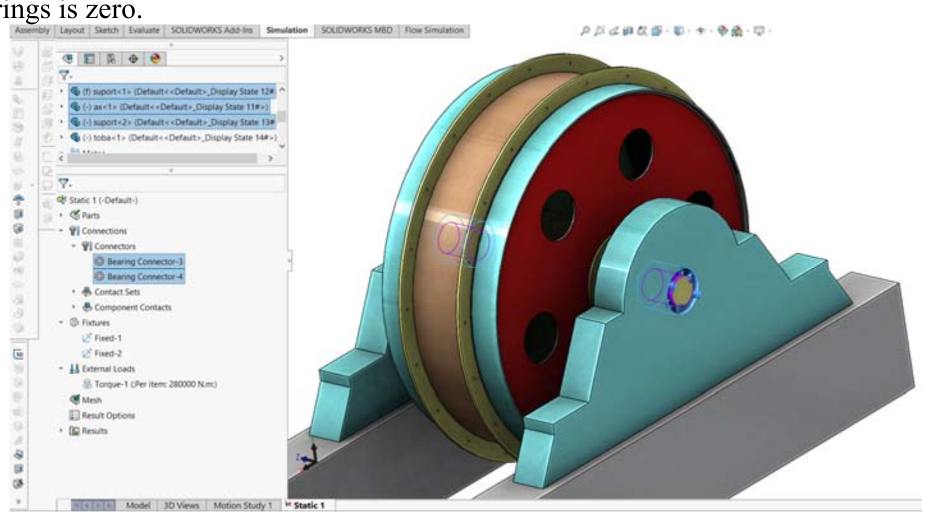

Fig.9 Connecting the rotation shaft with the drum supports by means of virtual bearings

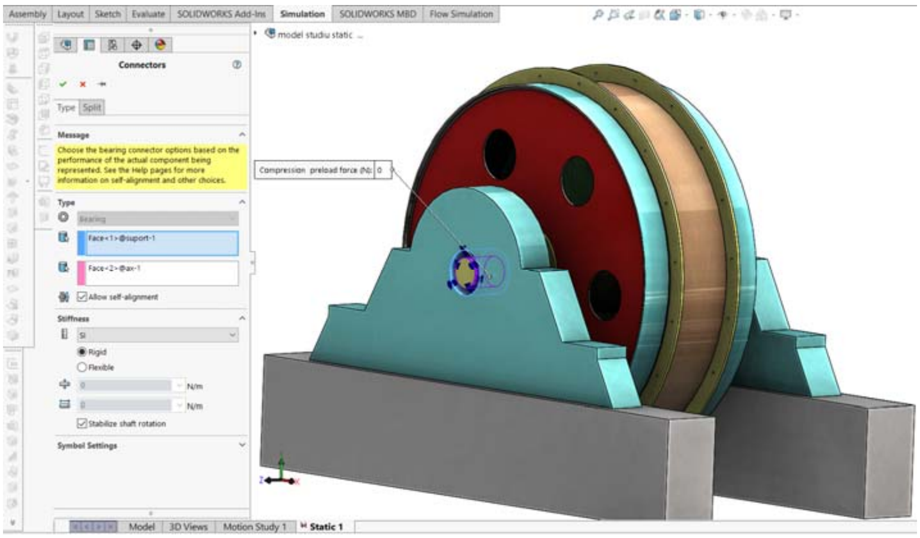

Fig.10 The characteristics of the connection mode through virtual bearings

3.2 Establishing the fixed parts of the assembly

Finding a solution for any static simulation using numeric methods requires setting up of fixed elements of the studied assembly. First, we considered fixed the two bottom surfaces of the drum supports (imposing fixed geometry), as seen in figure 11. Since the simulation is done following the safety braking which implies the immobilization of the drum of machine, we also considered external surface of the rotation shaft as fixed (figure 12).

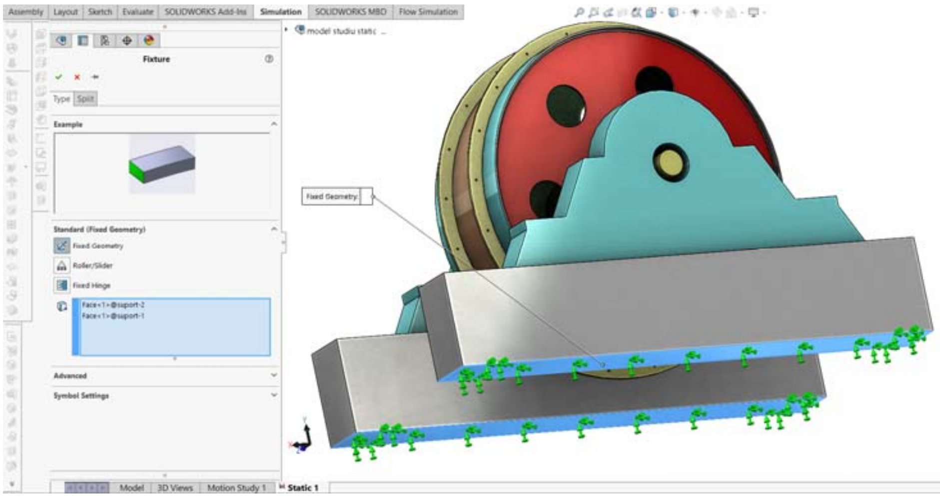

Fig.11 Setting the bottom surfaces of the drum supports as `Fixed Geometry`

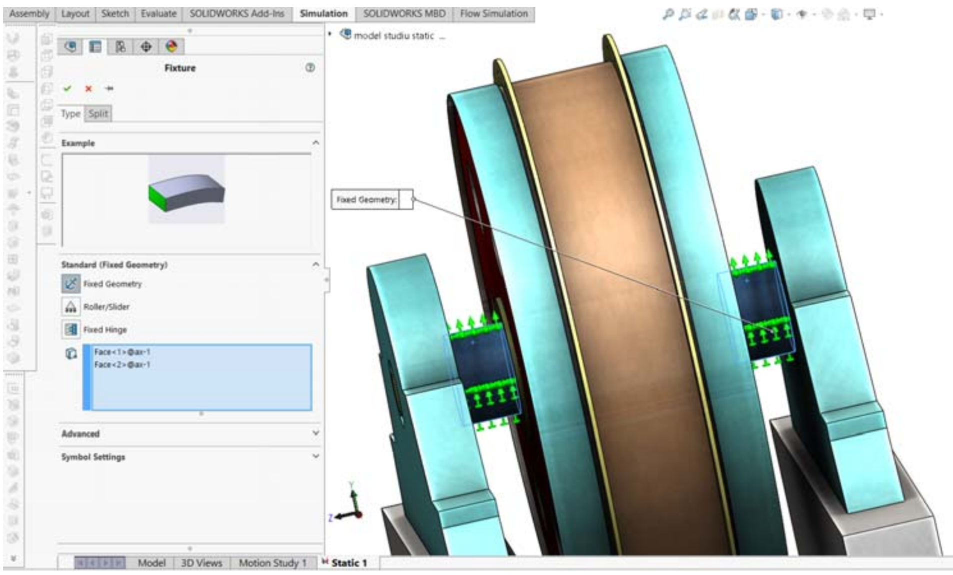

Fig.12 Setting the surface of the rotation shaft as `Fixed Geometry`

3.3 Establishing the load of the drum of the extraction machine

The static force at the periphery of the drive wheel of a multi-rope mine hoist with friction wheel:

$$F_{st} = [Q_u + (q - q_1) \cdot (H - 2x)]g \text{ , } [\mathrm{N}] \quad (1)$$

If we consider the hypothesis of a perfectly balanced mine hoist that assumes the equality of the specific mass of the extraction and balancing cables $(q = q_1)$, then equation (1) becomes:

$$F_{st} = Q_u \cdot g, \ [\mathrm{N}] \quad (2)$$

This static force determines a moment (figure 13) whose value can be calculated considering the manufacturer data of the mine hoist, in our case.

$$M_{Su} = F_{st} \cdot R = g \cdot M_{St} \cdot R = 9{,}81 \cdot 12.000 \cdot 2{,}375 \approx 280.000 \ \mathrm{N} \times \mathrm{m} \quad (3)$$

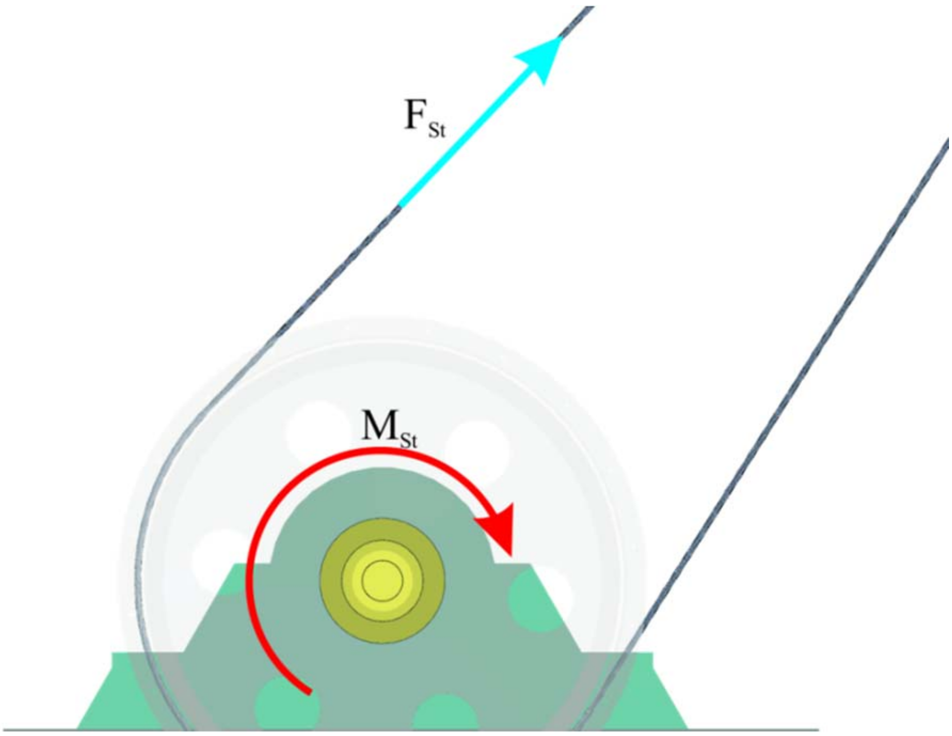

Fig.13 The moment acting on the drum of the mine hoist

We determined that the external load acting on the mine hoist drum assembly is a moment of torsion. Its application mode on the drive wheel model is presented in figure 14. Here we considered that the moment acts tangentially to the surface of the mine hoist drum driving the extraction cables, and we indicated the axis on which this moment is acting. We also specified that the indicated value of the moment refers only to the mentioned surface.

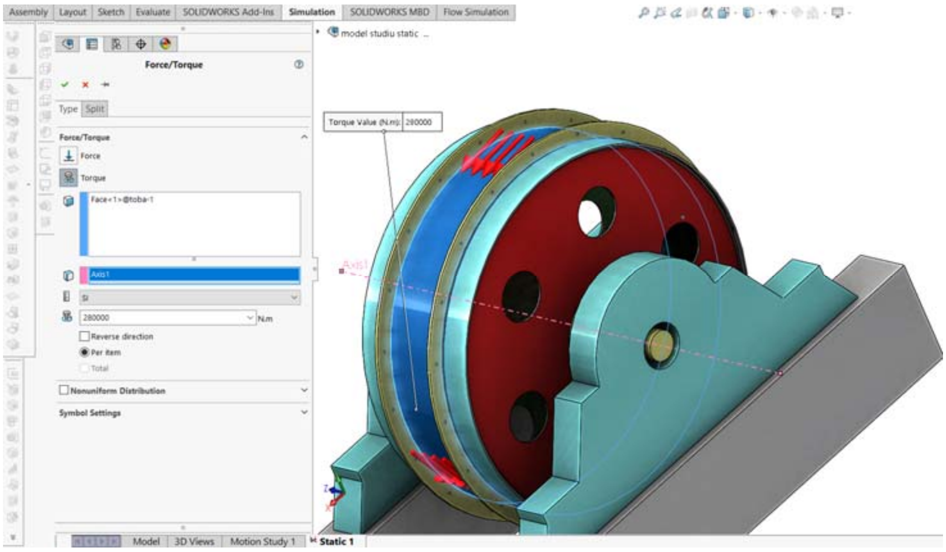

Fig.14 The application of the moment on the drum assembly model

3.4 Setup of the finite element mesh

In figure 15 we show the result of meshing, that shows the finite elements for the virtual assembly of the drum assembly.

Fig.15 Realization of the finite element

4. RESULTS OF THE SIMULATION

The simulation of the static stress of the drum of a mono-cable mine hoist subjected to the action of the payload suspended in the shaft resulted in the determination of the von Mises stress and the deformations, using the finite element method.

Thus, figure 16 shows the distribution of the von Mises stress on the whole assembly subjected to simulation.

Also in the figure we have highlighted the point on the drum where the von Mises stress is maximum.

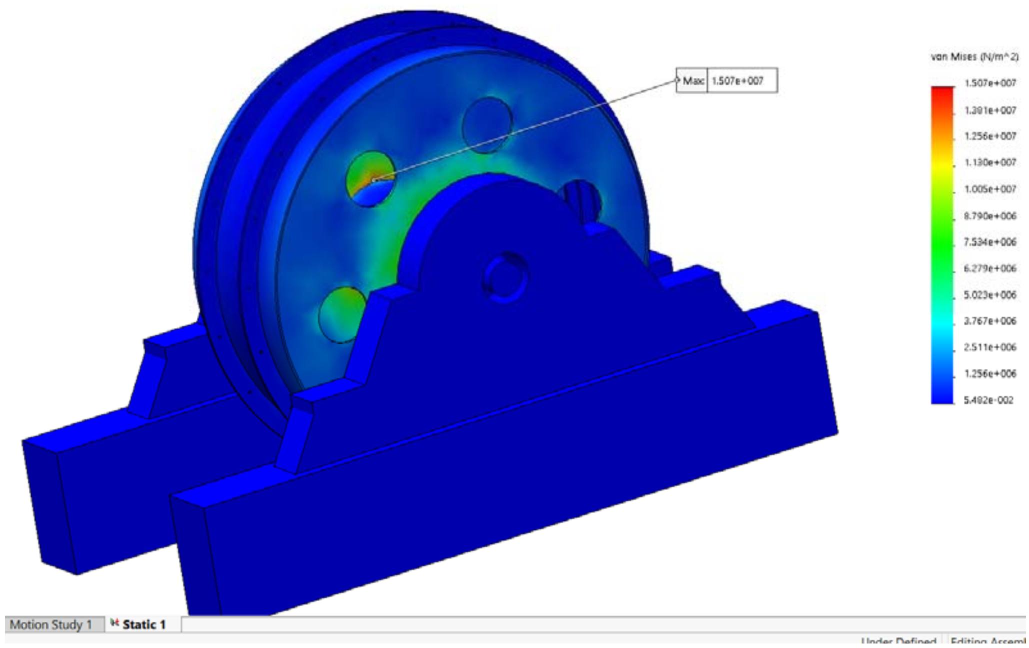

Fig.16 Distribution of von Mises stresses on the whole assembly subjected to simulation

To get a clearer picture of the von Mises stress distribution, we chose the `Section Clipping` option from the `Plot Tools` menu. We opted for a planar section.

The reference plane for sectioning is the side plane of the assembly (`Right Plane`). A clear picture of this option for representing the results is obtained by selecting `Show Section Plane`.

The result obtained in isometric view, highlighting the finite element, is shown in figure 17, and in figure 18 the same results are shown with a view from the side plane.

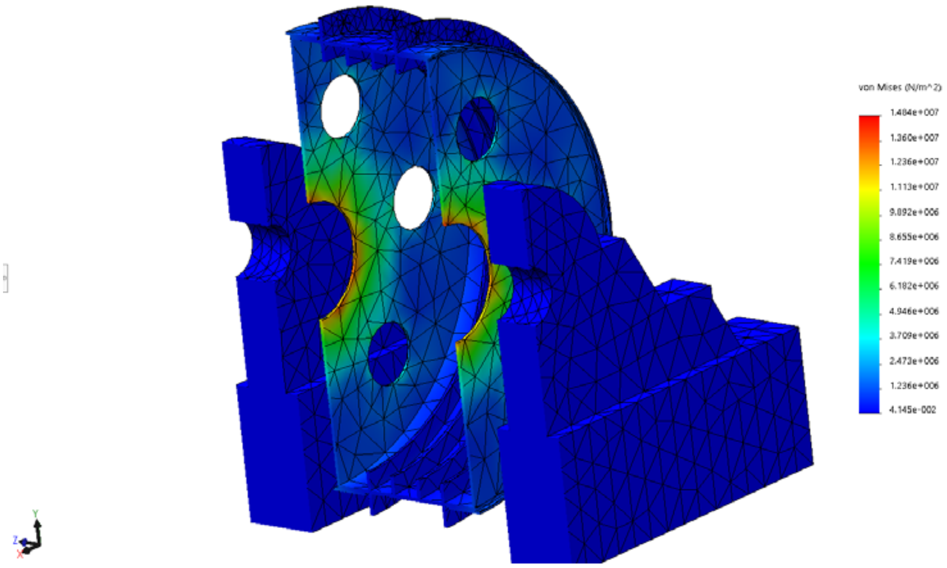

Fig.17 Distribution of von Mises stresses on the entire assembly subjected to simulation by using `Plot Tools` → `Section Clipping` in isometric view

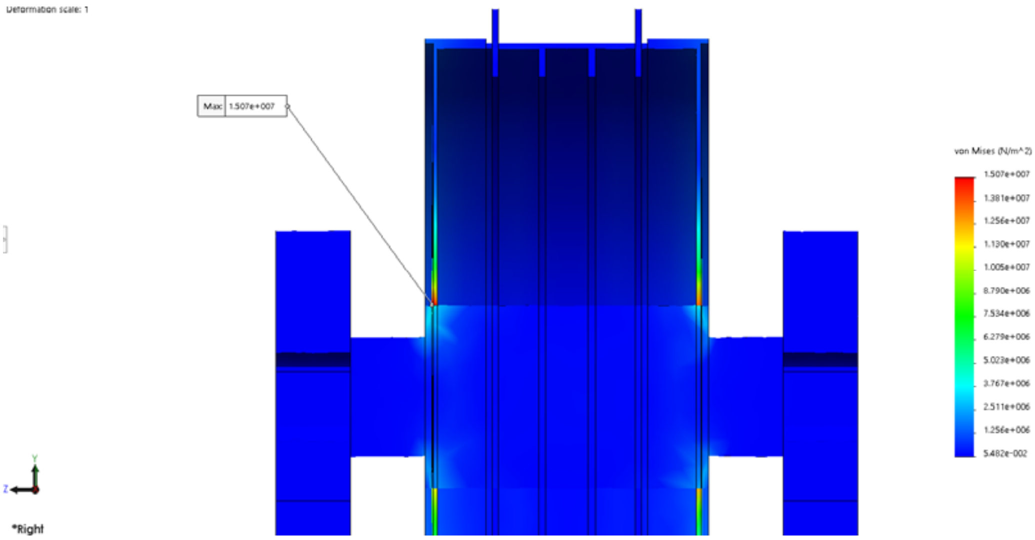

Fig.18 Distribution of von Mises stresses on the whole assembly subjected to simulation by using `Plot Tools` → `Section Clipping` with side view

Another way of viewing the same results is obtained through `Plot Tools` with the `Iso Clipping` option as can be seen in figure 19, and in figure 20 the same results are shown with the `Iso Clipping` view.

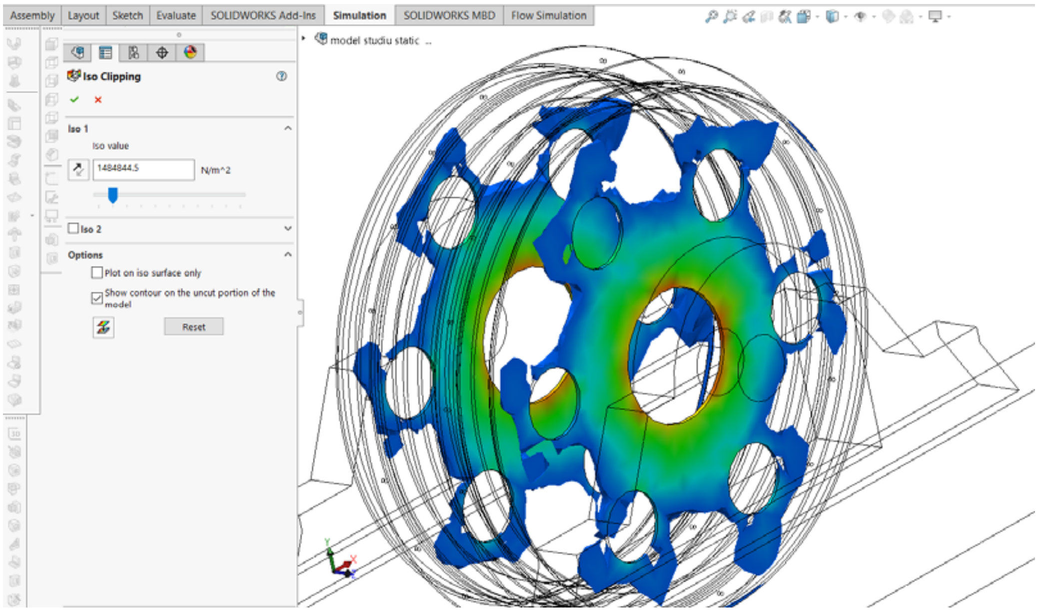

Fig.19 Highlighting the results by using `Plot Tools` → `Iso Clipping`

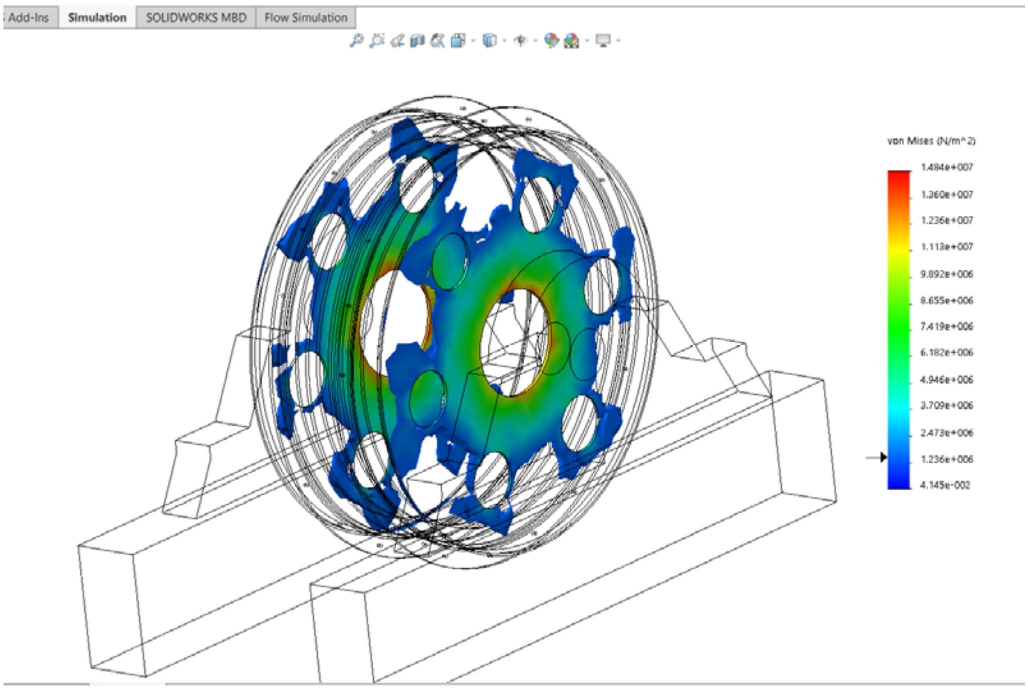

Fig.20 Distribution of von Mises stresses on the whole assembly subjected to simulation by using `Plot Tools` → `Iso Clipping`

Figures 21, 22 and 23 show the general deformations of the assembly in isometric view, using `Section Clipping` and `Iso Clipping`.

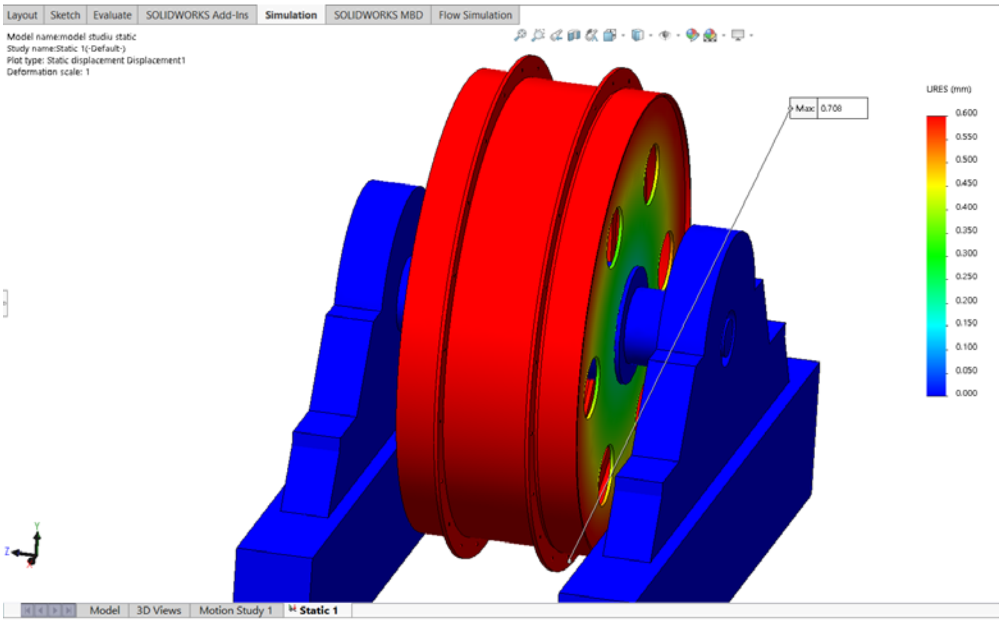

Fig.21 The distribution of deformations on the entire assembly subjected to simulation

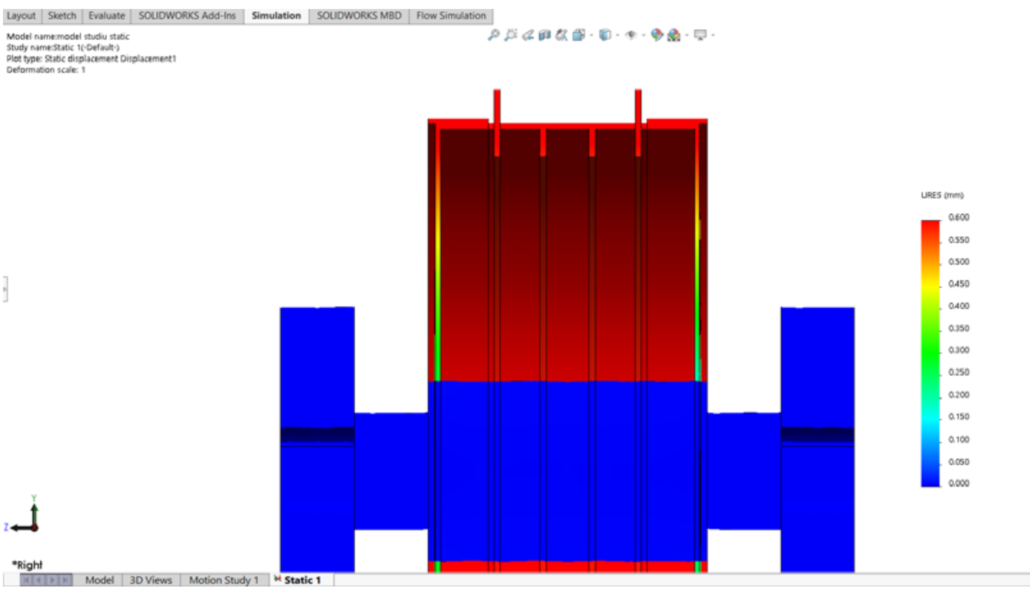

Fig.22 The distribution of deformations on the entire assembly subjected to simulation by using `Plot Tools` → `Section Clipping` with a side view

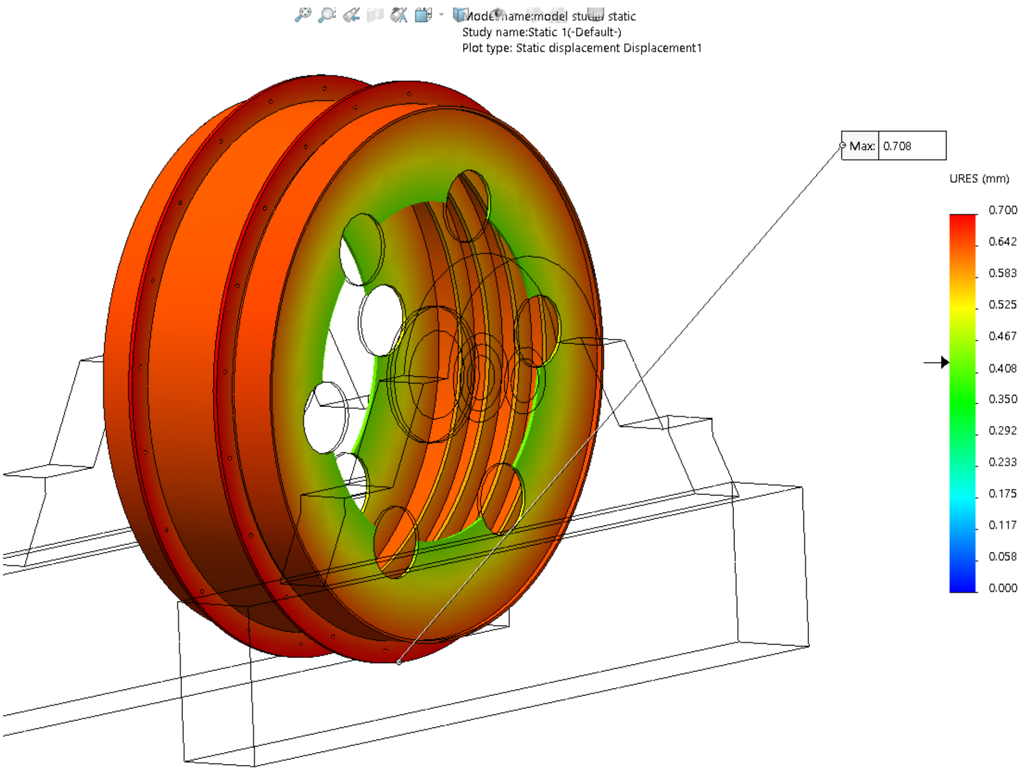

Fig.23 The distribution of deformations on the entire assembly subjected to simulation by using `Plot Tools` → `Section Clipping`

CONCLUSIONS

Finite Element Analysis (FEA) is a method of numerical analysis, being used to solve problems in various fields of engineering, such as machine design, acoustics, electromagnetism, solid mechanics, fluid dynamics and many others. From a mathematical point of view, finite element analysis is a numerical method used to solve problems characterized by a set of partial differential equations.

To be able to analyse the stresses and deformations to which the drum of a monocable mine hoisting machine with friction wheel is subjected when it has to keep the conveyances immobilized following a safety braking, a virtual model of the drum assembly was created in SOLIDWORKS.

Between the parts of the assembly a series of standard geometric connections were established, respectively coincidence and concentricity. At the same time, for the parts of the assembly the materials from which they are built and therefore implicitly their characteristics were imposed.

Using the `Simulation` menu of SOLIDWORKS it was opted for a static stress simulation for which the surfaces considered fixed were established, the connections by means of two bearings between the two supports with the rotation shaft, as well as the external stress, a moment acting on the contact surface of the drum and extraction cable were also defined and imposed.

The characteristics of the finite element mesh of this assembly was also setup and discretization took place.

The analysis carried out allowed the determination of the von Mises stress generated by the moment corresponding to the mass (payload) stopped and suspended in the shaft.

The calculated value does not exceed the yield stress of the material from which the drum is constructed.

The proposed model can be easily applied to other types of mining hoist and can be used in their design and verification.

REFERENCES

[1]. **Kertesz, I., Radu, S.M., Popescu, F. D., Andras, A.**, *Modelarea si simularea procesului de dislocare utilizând excavatoare cu rotor – format electronic*, Editura Universitas, Petroşani, 2020, ISBN: 978-973-741-639.3.

[2]. **Kurowski, P.,M.**, *Engineering Analysis with SOLIDWORKS® Simulation 2015*.

[3]. **Popescu, F. D., Radu, S.M., Andras, A., Kertesz, I.**, *Infografică, modelare şi simulare asistată de calculator – format electronic*, Editura Universitas, Petroşani, 2020, ISBN 978-973-741-715-2.

[4]. **Popescu, F.D.; Radu, S.M.** *Vertical Hoist Systems, New Trends Optimizations*; LAP LAMBERT Academic Publishing: Saarbrucken, Germany, 2013.

[5]. **Popescu, F.D., Radu, S.M. Brînaş, I.** *Simulation And Modelling Of A Statically Balanced Multi-Rope Hoisting Machinery,* Monografia Problemy Transportu I Przerobki W Gornictwie Modelowanie Procesow, pag. 211 – 221, Akademia Gorniczo-Hutnicza. Krakow 2017. ISBN 978-83-930353-7-3.

[6]. **Popescu, F.D., Dimirache, G.**, *Instalaţii de transport pe verticală –Calcule de verificare şi dimensionare*, Editura Focus, Petroşani, 2011, ISBN 978-973-677-182-8.

[7]. **Popescu, F.D.,** *Programarea calculatoarelor în limbaj C*, Editura Focus, Petroşani, 2010, ISBN 978-973-677-177-4.

[8]. **Popescu, F.D., Mitran, I.,** *Vertical Transport Installations Balance Survey*, WSEAS TRANSACTIONS on SYSTEMS and CONTROL, Issue 10, Volume 4, October 2009, http://www.wseas.us/e-library/transactions/control/2009/29-678.pdf, ISSN: 1991-8763, pp. 509...518.

[9]. **Popescu, F.D.**, *Dynamic and kinematics parameters optimisation of vertical transport installations*, Proceedings of the 12th International Conference on Artificial Intelligence, Knowledge Engineering and Data Bases (AIKED '13), Proceedings of the 12th International Conference on Software Engineering, Parallel and Distributed Systems (SEPADS '13) Cambridge, UK February 20-22, 2013, http://www.wseas.us/e-library/conferences/2013/CambridgeUK/AISE/AISE-37.pdf, ISSN: 1790-5109, ISBN: 978-1-61804-162-3.

[10]. Radu, S.M., Popescu, F.D., Andras, A., Kertesz, I., *Transport si instalaţii miniere*, Editura Universitas, Petroşani, 2018, ISBN 978-973-741-587-5.
[11]. Popescu, F.D. *Instalaţii de Transport pe Vertical*; Editura Focus: Petroșani, Romania, ISBN 978-973-677-182-8, 2010.
[12]. Popescu, F.D., Radu, S.M., Andraş, A., Brînaş, I., Budilică, D.I., Popescu, V. *Comparative Analysis of Mine Shaft Hoisting Systems' Brake Temperature Using Finite Element Analysis (FEA),* Materials, 2022, 15, 3363.
[13]. Popescu, F.D., Radu, S.M., Andraş, A., Brînaş, I. *Numerical Modeling of Mine Hoist Disc Brake Temperature for Safer Operation*. Sustainability, 2021, 13(5), 2874.