

Finite element analysis and geometry optimization of a flywheel used in a monorail conveyor

I Stanasel¹, F Blaga¹, T Buidos¹ and D Craciun²

¹University of Oradea-Romania, Department of Industrial Engineering, Universitatii, No. 1, 410087, Oradea, Romania

²University of Oradea-Romania, Department of Mechatronics, Universitatii, No. 1, 410087, Oradea, Romania

E-mail: stanasel@uoradea.ro

Abstract. In this work is presented the theoretical aspects for dimensioning calculations, design, and finite element analysis of the drive flywheel of the cable of a monorail conveyor. In order to obtain 3D model and study the behaviour of the drive wheel were used application for computer-aided design (CAD), applications for engineering calculations (CAE), and finite element analysis (FEM). After analyzing the obtained results, it was used a computer program for optimizing the geometry of a flywheel by modifying some geometrical parameters to obtain a minimum weight and the displacements and stress to be within allowable limits. The use of computer offers the possibility to automate engineering calculations ensuring the rapid processing of data and thereby reducing the time and cost of developing a product, in comparison with the traditional way.

1. Introduction

The paper presents sizing calculations, the 3D design and finite element analysis of the drive wheel of traction cable of a suspended conveyor. Carriage of products is provided by carts traveling on an airway. Drive carriage is realized by a flexible cable driven by a wheel controlled by an electric motor.

The drive wheel elements are subject to complex loads. The obstruction is required for traction as a result of the centrifugal forces, due to the force of the press on the wheel, and the arms are stretched due to the centrifugal forces, compression as a result of the pressure of the cable on the bending obstruction due to the moment of torsion by which the useful force is transmitted. The system is statically indefinite and the calculations are laborious. For this reason in practice is used a simplified calculation.

The scientific literature confirms that there are concerns of many researchers on different aspects of the flywheels. Thus, in the paper [2] is presented a comparative study on the distribution of stresses in the flywheels made from different materials, in order to satisfy the safety conditions in operation. Researchers [3] use finite element analysis to study the behavior of a flywheel and to reduce its weight in order to reduce costs.

In articles [3] and [5] is done an analysis regarding the maximum radial stress of flywheel manufactured by composite material. It is studied the distribution of stresses for a composite material under high speed conditions, the obtained results allowing the design and manufacture of high-performance impellers. In [5] and [7] is presented the optimization of a flywheel by using finite



element analysis to maximize momentum of inertia while reducing material consumption and manufacturing costs and ensuring increased flywheel reliability.

2. Theoretical aspects

Transmission of the movement to the conveyor transport trolleys is achieved by a transmission with a cable fixed on two wheels (Figure 1). In order to transmit the movement, the cable is mounted with an initial tension F_0 which produces a pushing force and which due to the friction between the cable and the wheel allows the transmission of a useful force expressed by [8], equations (1) and (2):

$$F_u = \frac{2M_t}{D_1} \quad (1)$$

$$F_u = \frac{P}{v} 10^3 \text{ [N]} \quad (2)$$

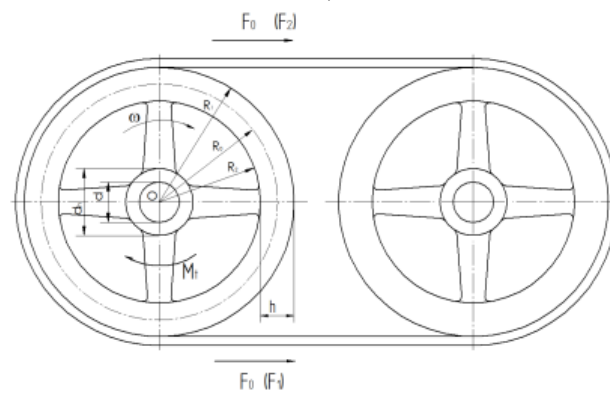


Figure 1. The cable transmission.

During operation, the forces in the transmission element branches change. Thus in the active branch the force increases from F_0 to F_1 , and in the passive branch the force decreases from F_0 to F_2 . The link between these forces is expressed by, (equation (3)):

$$2F_0 = F_1 + F_2 \quad (3)$$

From the equilibrium condition of the moments with respect to the axis of the drive wheel it results, (equation (4)):

$$F_u = F_1 - F_2 \quad (4)$$

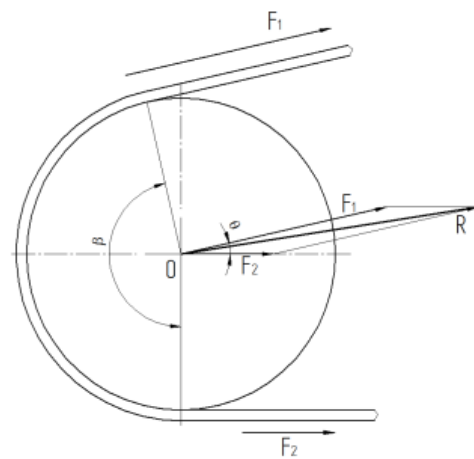
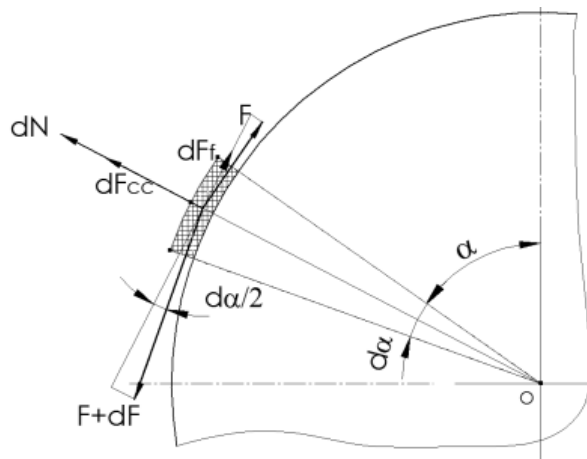


Figure 2. The forces which act on a cable element. **Figure 3.** The resultant force that presses the rim.

The transition from the force F_2 from the passive branch to the force F_1 in the active branch is gradually made by summing the frictional forces acting on each corner belt element. Knowing the F_u transmission force, you may have the problem of determining the pressing force on the wheel R . It is written the equilibrium on the radial and tangential direction of the forces acting on a cable element that describes the angle at the center $d\alpha$ (figure 2):

On radial direction, (equation (5)):

$$(F + dF) \sin \frac{d\alpha}{2} + F \sin \frac{d\alpha}{2} - dF_{cc} - dN = 0 \quad (5)$$

On tangential direction, (equation (6)):

$$(F + dF) \cos \frac{d\alpha}{2} - F \cos \frac{d\alpha}{2} - dF_f = 0 \quad (6)$$

The centrifugal force acting on the cable element can be expressed by the relationship, (equation (7) and (8)):

$$dF_{cc} = F_{cc} d\alpha \quad (7)$$

Or

$$dF_{cc} = \rho_c S_c v^2 d\alpha \quad (8)$$

Because angle $\frac{d\alpha}{2}$ has low values they might be done approximations $\sin \frac{d\alpha}{2} = \frac{d\alpha}{2}$ and $\cos \frac{d\alpha}{2} = 1$, but the term $dF \sin \frac{d\alpha}{2}$ can be neglected, being very small.

The friction force on the angle cable element $d\alpha$ is, (equation (9))

$$dF_f = \mu dN \quad (9)$$

After the calculations are obtained:

$$Fd\alpha - F_{cc}d\alpha - \frac{dN}{\mu} = 0 \quad (10)$$

$$dN = \frac{dF}{\mu} \quad (11)$$

By equations (10) and (11) results:

$$\frac{dF}{F - F_{cc}} = \mu d\alpha \quad (12)$$

The relation (12) is integrated between the limits F_2 și F_1 for the force and between the limits 0 and β for the angle α :

$$\int_{F_2}^{F_1} \frac{dF}{F - F_{cc}} = \int_0^\beta d\alpha \quad (13)$$

It is obtained:

$$\frac{F_1 - F_{cc}}{F_2 - F_{cc}} = e^{\mu\beta} \quad (14)$$

From relations (3), (4) and (14), after the calculations they result:

$$F_0 = F_{cc} + \frac{F_u}{2} \cdot \frac{e^{\mu\beta} + 1}{e^{\mu\beta} - 1} \quad (15)$$

$$F_1 = \frac{2e^{\mu\beta}(F_0 - F_{cc})}{e^{\mu\beta} + 1} + F_c \quad (16)$$

$$F_2 = \frac{2(F_0 - F_{cc})}{e^{\mu\beta} + 1} + F_c \quad (17)$$

$$F_u = 2(F_0 - F_{cc}) \frac{e^{\mu\beta} - 1}{e^{\mu\beta} + 1} \quad (18)$$

The resultant force R that presses on the wheel guard during operation can be easily determined from Figure 3:

$$R = \sqrt{(F_1 + F_2 \cos \theta)^2 + (F_2 \sin \theta)^2} \quad (19) \text{ Taking into account that:}$$

$$\theta = \pi - \beta \quad (20)$$

After solving the calculations, it results:

$$R = \sqrt{F_1^2 - 2F_1F_2 \cos \beta + F_2^2} \quad (21)$$

During rotation by angular velocity ω around the normal axis on the plane passing through the center of the wheel over the rim and spindle wheels, the centrifugal force acts. Considering the rim with rectangular section $S_r = b \cdot h$ and its mean radius R_0 , the value of the acting centrifuge force is given by the expression (22):

$$F_c = \rho \cdot b \cdot h \cdot R_0 \cdot \omega^2 \quad (22)$$

The centrifugal force acting on the arm with a constant section at the distance r from the centre of the wheel is equal to, (equation (23)):

$$F_{ca} = \rho \cdot S_a \cdot r \cdot \omega^2 \quad (23)$$

The flywheel can be built with rim or as disc shape. The empirical relationship that establishes this has the form, (equation (24)):

$$z = \left(\frac{1}{5} \dots \frac{1}{7} \right) \cdot \sqrt{D_1} \quad (24)$$

The value of z number indicates whether the wheel will be constructed with rim ($z > 3$) or disc-shaped ($z < 3$).

In the studied case it was considered a flywheel with the following characteristics: (Table 1):

Table 1. The main characteristics of the flywheel.

Parameters	Values
Outer diameter	1200 mm
Width of the rim	50 mm
Inner hub diameter	70 mm
Diameter of the hub flange	120 mm
Rim thickness	5 mm
Number of arms	4

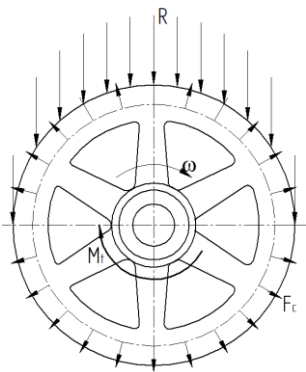


Figure 4. Loads of the flywheel.

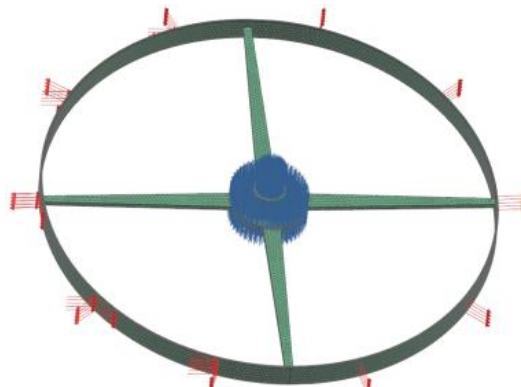


Figure 5. Loads and constraints of the discretized model.

After the calculus was obtained the following values for the loads (Figure 4).

- Pressing force: $R = 5000 \text{ N}$
- Torque: $M_t = 250000 \text{ Nmm}$
- Centrifugal force: $F_c = 210 \text{ N}$

3. FEM analysis

In order to improve the quality of the product, they simulate its behavior during operation using finite element analysis. The method allows to model and describes the approximate behavior of structures with complex geometry, telling the geometric form in finite elements and nodes [1]. The analysis involves the following steps: Preprocessing (input of the initial data, establishing the material properties, boundary conditions, tests, constraints, meshing the structure), processing (input data processing and equations solving) postprocessing (presentation of results). There are two types of analysis used in the industry: 2D and 3D. The 2D analyzes are simpler and allow a faster run, but the results are less accurate. The 3D analyzes give more accurate results, but calculations take longer time.

For the case study, the static analysis performed with the Simens NX application with the Nastran solver was used. The physical properties, the properties of the material from which the flywheel is made (Table 2) and the structure with CTERA10 elements of the size of 5 mm have been defined in the operations required for the preprocessing step. Discretization is one of the important stages of numerical analysis, the accuracy of the results depending on the mesh quality. Also, the attempts and constraints of the discretized model were applied. In Figure 5 are presented the meshed model with the constraints and trials applied.

Table 2. Material properties.

Property name	value
Density	7833.000 kg/m ³
Modulus of Elasticity	199947953.000 kPa
Poisson's Ratio	0.290
Yield Stress	262000.766 kPa
Ultimate Stress	358527.364 kPa

In the processing stage, the influence of each test taken separately was studied in the first phase in order to determine which is the load that produces the most pronounced effects on stresses and strains. At the same time, the cumulative effect of loads on deformation and stresses during operation was also studied. At the end of the processing stage, the application can provide data on the level of confidence of the model and recommends actions needed to be taken if the confidence level is low, (Figure 6).

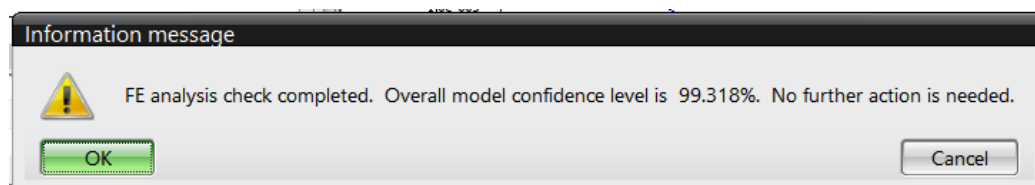


Figure 6. Confidence level.

In Figure 7 and Figure 8 are shown the displacements and stresses by applying loads. The analysis of the results shows that the wheel elements have large deformations and stresses that exceed the permissible limit of the material, which involves changing of the 3D wheel model by modifying the thickness of the rim and arms, changing the number of arms so that the displacements and tensions are minimal. In the conditions of series production, minimizing the material consumption is one important target.

Displacement - Nodal, Magnitude
Min : 0.000, Max : 10.862, Units = mm
Deformation : Displacement - Nodal Magnitude

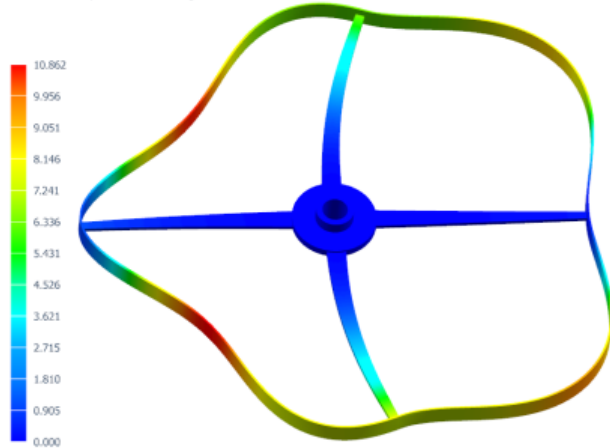


Figure 7. Displacements of the initial model.

Stress - Element-Nodal, Unaveraged, Von-Mises
Min : 0.03, Max : 603.91, Units = N/mm²(MPa)
Deformation : Displacement - Nodal Magnitude

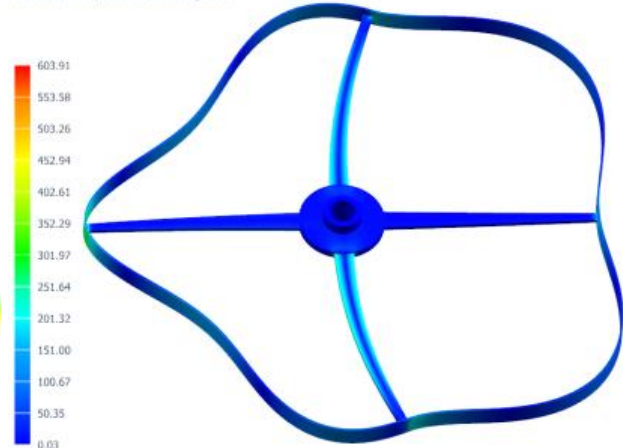


Figure 8. Stresses of the initial model.

4. Geometry optimization

The geometry optimization is a process that aims to modify specific geometric parameters of the 3D model to minimize wheel deformations and stresses during operation, at the same time with a decrease of the costs related to the quantity of material used.

For optimization, it is necessary to specify the objectives, and defining constraints and variables. Starting from finite element analysis results obtained previously, in the application Siemens NX are defined the geometric parameters of the 3D model that can be modified, as well as the limits in which they can take values and the number of iterations. The parameters which can be defined as variables can be sketches, feature dimensions, sections of the 3D model, material properties, mesh density, etc.

For this case study, the geometric parameters of the flywheel which were considered as variables within the iterations are:

- inclination angle of the flank of the arm—limits: 1.5 - 4
- rim thickness – limits: 2 – 25 mm
- arm thickness - – limits: 10 – 20 mm)
- number of arms – limits: 4 - 8

For constrained parameters the following values were imposed:

- displacements maxim 0.1 mm
- stress (max 300 MPa)

The objective consists of minimizing the weight of flywheel (max 800 N), while respecting the constraints and keeping within the limits of the variable parameters. For optimizing were used 20 iterations. During any iteration the value of each constrained element is compared with the imposed limits, and they are changed the geometrical parameters of the initial 3D. Each 3D model obtained as following the 20 iterations is discretized and subjected to the finite element analysis, determining the displacements and stresses.

After the completion of the calculation, the results are provided both for the geometry of the 3D model and those relating to the displacements and stresses. There were identified three models which get the imposed conditions regarding on displacements and stresses, as well as framing in the weight limit of the wheel. It was chosen the model that had the minimum weight, the mass of the flywheel being 53.686 kg.

In Figure 9- Figure 12 are presented in a comparative manner the variable geometrical parameters of the 3D model in the initial stage and those obtained after optimization.

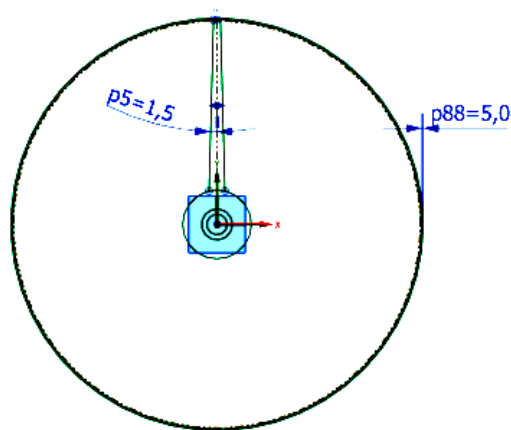


Figure 9. Geometrical parameters of the sketch of the initial model.

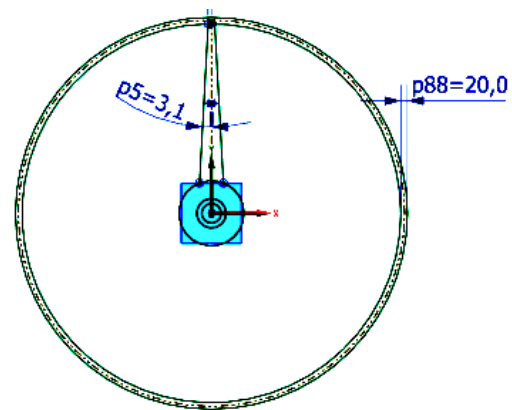


Figure 10. Geometrical parameters of the sketch of the optimized model.

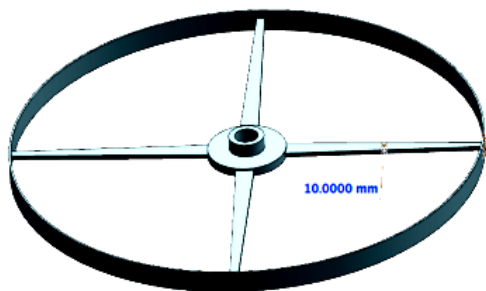


Figure 11. Numbers and thickness of the arms of the initial model.

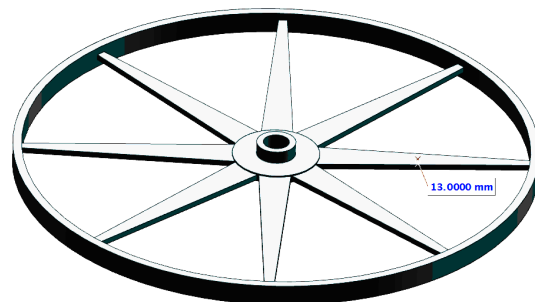


Figure 12. Numbers and thickness of the arms of the optimized model.

The results of the analysis for the optimized model of the flywheel are presented in Figure 13- Figure 20. They are presented the values of the displacements, and the stresses produced by each load considered separately, as well as the simultaneous application of the loads.

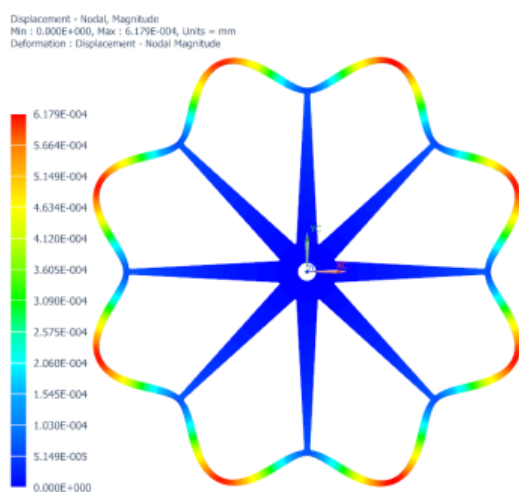


Figure 13. Displacements of optimized models considered only centrifugal force.

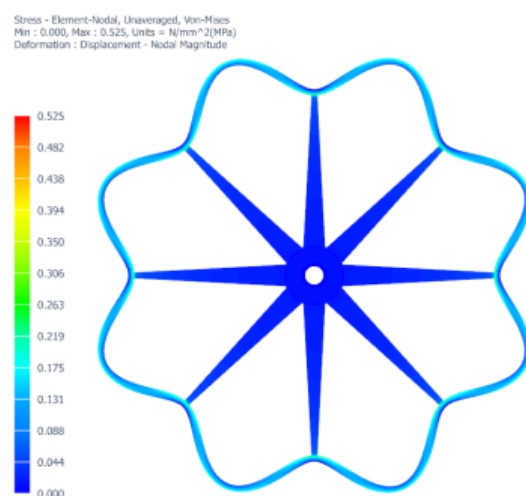


Figure 14. Stresses of optimized models considered only centrifugal force.

Displacement - Nodal, Magnitude
Min : 0.0000, Max : 0.0263, Units = mm
Deformation : Displacement - Nodal Magnitude

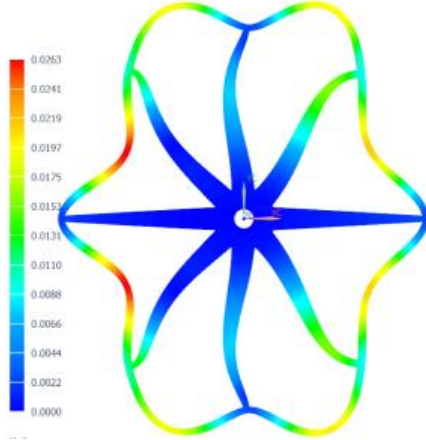


Figure 15. Displacements of optimized models considered only pressing force.

Stress - Element-Nodal, Unaveraged, Von-Mises
Min : 0.00, Max : 16.91, Units = N/mm²(MPa)
Deformation : Displacement - Nodal Magnitude

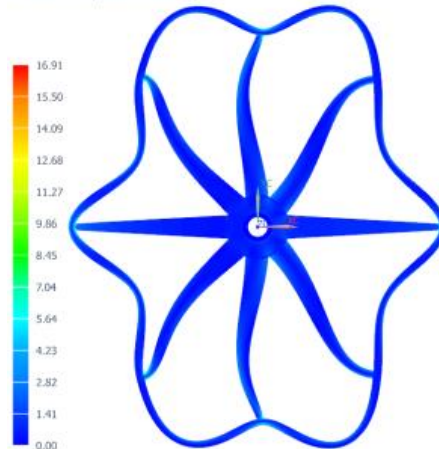


Figure 16. Stresses of optimized models considered only pressing force.

Displacement - Nodal, Magnitude
Min : 0.000E+000, Max : 9.506E-004, Units = mm
Deformation : Displacement - Nodal Magnitude

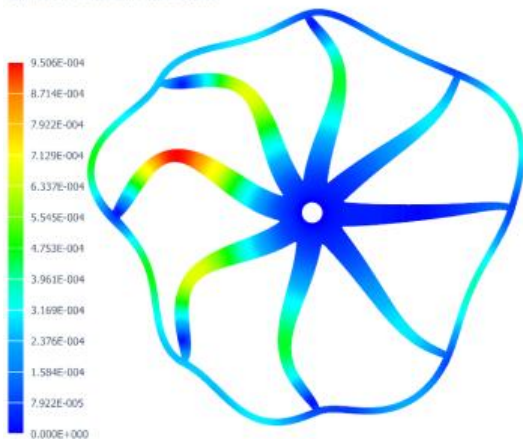


Figure 17. Displacements of optimized models considered only torque.

Stress - Element-Nodal, Unaveraged, Von-Mises
Min : 0.000, Max : 0.772, Units = N/mm²(MPa)
Deformation : Displacement - Nodal Magnitude

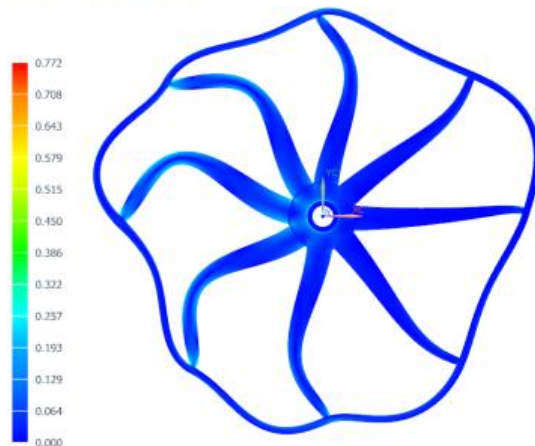


Figure 18. Stresses of optimized models considered only torque.

Displacement - Nodal, Magnitude
Min : 0.0000, Max : 0.0261, Units = mm
Deformation : Displacement - Nodal Magnitude

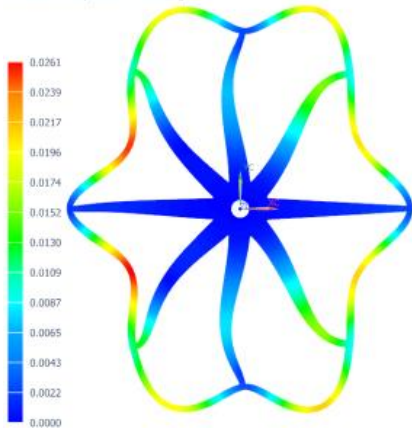


Figure 19. Displacements of optimized models considered all loads.

Stress - Element-Nodal, Unaveraged, Von-Mises
Min : 0.00, Max : 17.46, Units = N/mm²(MPa)
Deformation : Displacement - Nodal Magnitude

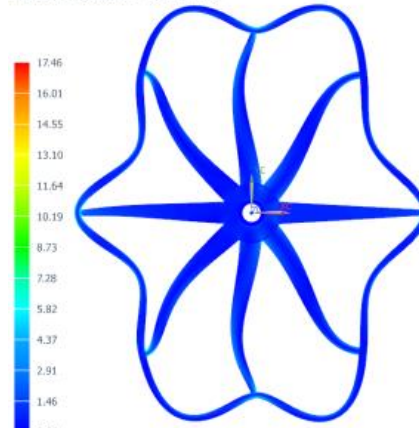


Figure 20. Stresses of optimized models considered all loads.

After the analysis of the influence of each load regarding on the displacements and stresses of the optimized flywheel, it was observed that both for the rim and the arms the distribution of displacements and tensions and their size changed from the original model (Figure 21 - Figure 24).

The maximum displacement is about 0.027 mm, and the maximum stress is about 17MPa, values which are under allowable limits.

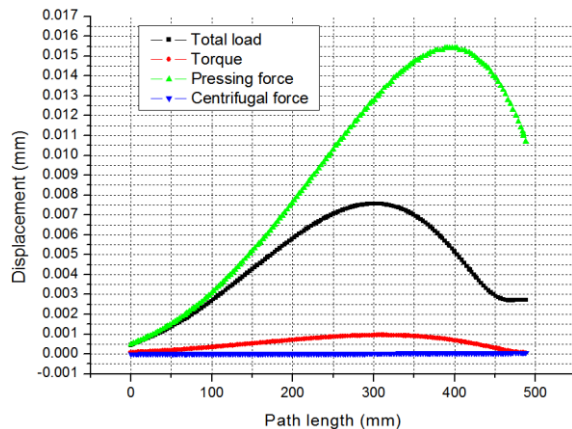


Figure 21. Displacements of arm.

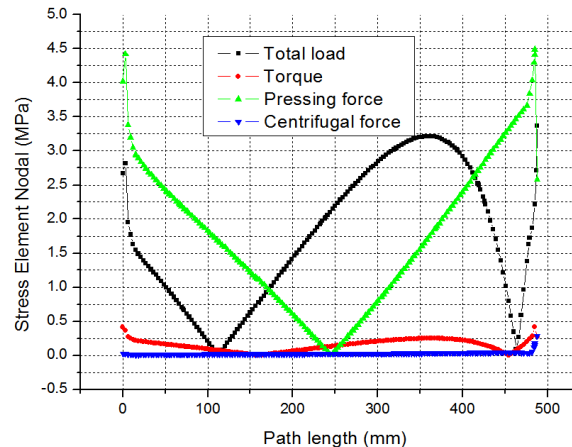


Figure 22. Stresses of arm.

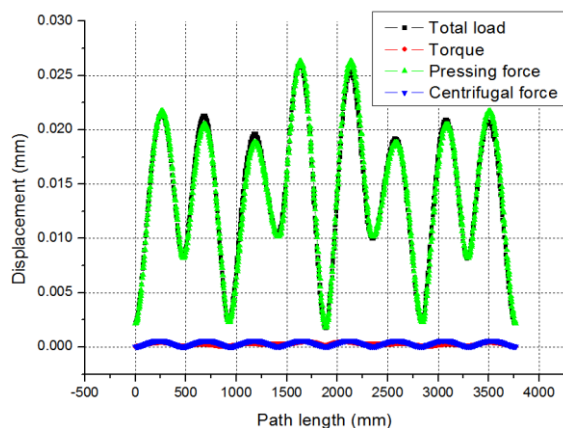


Figure 23. Displacements of rim.

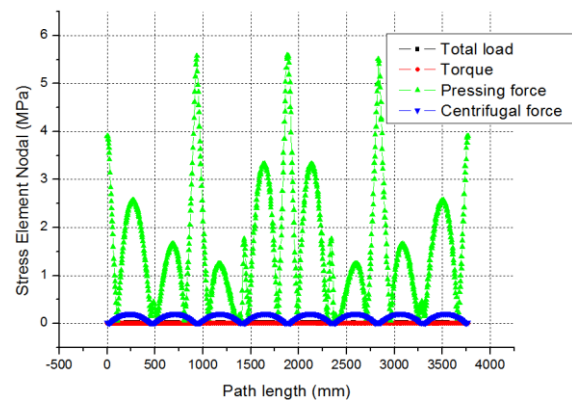


Figure 24. Stresses of rim.

5. Conclusions

Since the drive wheel is subject to complex loads, the determination of the stresses and the displacements in the rim and the arms by the analytical method is complicated. Finite element analysis allows you to determine the behavior of the wheel in the operating conditions, and modification of the 3D model if they found that the limits permitted for specific parameters such as displacement or stress exceeded.

Based on the results of the analysis with finite elements was achieved optimization of the 3D model so that it corresponds to the operating state, but obtained with minimal consumption of material, a condition required for the series production of the wheel. In order to optimize, have been defined the geometric parameters which can be changed, the constrained parameters and the parameter that must be optimized.

It was conducted a finite element analysis for 20 types of wheels and it was chosen the version with the minimum mass and which supplies the conditions regarding the displacements and stresses. As a result of the optimization, have changed the number of wheel arms, their thickness, the angle of inclination of the flank of the arm, and the thickness of the rim.

The results obtained after finite element analysis of the optimized model show that it corresponds to the deformations and maximum admissible stresses. For the optimized model, the effects of each separate load and the cumulative effect of loads were analyzed. The analysis of the results show that the pressing force has the most significant influence on the displacements and stresses occurring in the rim and arms of the wheel.

The displacements and stresses in the arms of the wheel depend on the angular position relative to the direction of the pressing force of the wheel drive cable. Regarding the arms, it is found that the minimum displacement and the highest stress appear at the joint with the wheel hub. The highest stress of the rim is in the contact area with the cable drive, the most significant role having the pressing force of the cable.

Acknowledgments

The work is a part of project MANUNET ERA-NET, Contract nr.19/2015 “Monorail conveyor system for automated production lines”

Nomenclature:

b	-	Width of the rim
d	-	Inner hub diameter
D_1	-	Outer diameter
D_2	-	Inner rim diameter
d_h	-	External hub diameter
$d\alpha$	-	Angular element of cable
F_0	-	Tensioning force
F_1	-	Force in active branch
F_2	-	Force in driven branch
F_c	-	Centrifugal force which acting on the rim
F_{ca}	-	Centrifugal force which acting on the arm
F_{cc}	-	Centrifugal force which acting on the cable
F_f	-	Fricional force
g	-	Gravitational acceleration
G	-	Weight of the flywheel
h	-	Rim thickness
I	-	Moment of inertia
m	-	Mass of the flywheel
M_t	-	Torque
N	-	Normal force
P	-	Transmitted power
R	-	Resulting pressing force
R_0	-	Median radius of the rim
R_1	-	External radius of the rim
R_2	-	Inner radius of the rim
S_a	-	Section Of Arm
S_c	-	Section Of Cable
S_r	-	Section Of Rim
v	-	Linear speed of the flywheel
z	-	Numbers of arms
β	-	The cable winding angle on the wheel
θ	-	Center angle
μ	-	Friction coefficient between flywheel and cable
ρ	-	Wheel material's density
ρ_c	-	Cable material density
ω	-	Angular velocity

6. References

- [1] Sasikumaar G, Lutz T 2017 *Finite Elements - Theory and Algorithms* (Cambridge University Press)
- [2] Hongliang C, Changsheng Z and Peng Y 2014 A Comparison of Analysis Flywheel Stress Distributions Based on Different Material *Applied Mechanics and Materials* **536-537**(2014) 1291-1294
- [3] Sushama G, Bawane A, Ninawe P and Choudhary S K 2012 Analysis and optimization of flywheel *International Journal of Mechanical Engineering and Robotics Research* **1**(2)
- [4] Shaobo W 2014 Analysis of maximum radial stress location of composite energy storage flywheel rotor *Appl Mech* **84** 1007–1013
- [5] Li Z, Jiang T, Bi X, Li H and Li L 2013 Analysis of strength of energy storage flywheel rotating at a high speed *Applied Mechanics and Materials* **251** 42-46
- [6] Choudhary S M and Shahare D Y 2013 Design Optimization of Flywheel of Thresher Using FEM *International Journal of Emerging Technology and Advanced Engineering* **3**(2) 595 - 597
- [7] Yang C, Huang D and Cheng F 2014 Finite Element Analysis and Lightweight Design of Crusher Flywheel *Applied Mechanics and Materials* **684** 297-302
- [8] Chisiu A, et all 1981 *Organe de masini* (Bucharest: Didactic and Pedagogic Publishing House)