IOP Conference Series: Materials Science and Engineering

Finite element analysis of cylindrical gear with mechanical event simulation

IOP Conf. Series: Materials Science and Engineering **393** (2018) 012046 doi:10.1088/1757-899X/393/1/012046

Finite element analysis of cylindrical gear with mechanical event simulation

I Z Miklos[1], C Miklos[1] and C I Alic[1]

[1]Politehnica University of Timişoara, Faculty of Engineering Hunedoara, 5 Revoluției Street, 331128 Hunedoara, Romania

E-mail: imre.miklos@upt.ro

Abstract. In this paper, we present the finite element analysis of a cylindrical gear using the Autodesk Simulation Mechanical software, with mechanical event simulation (MES). MES combines the kinematic, rigid, and flexible-body dynamics with the nonlinear stress analysis. Therefore, MES can be used to simultaneously analyse mechanical events involving large deformations, nonlinear material properties, kinematic motion and the forces caused by this motion for predicting the resulting stresses. It is essential that MES can be used to define the surface contact between the teeth in contact at a given time, the analysis parameters by time steps, respectively the kinematic and kinetostatic parameters, variables defined by certain curves. We use the analysis to find the stress distribution, nodal displacements, etc. in the contact area of the two gear wheels, at different time steps of the kinematic cycle.

1. Introduction

The finite element method, being a numerical integration method applied to the partial derivative equations put in variational form, represents the ideal tool to solve static and dynamic problems in various engineering fields through linear or nonlinear analysis.

The linear analysis assumes linear elastic behaviour and available infinitesimally small angular displacements and strains. In the linear analysis, the costly load incremental and iterative procedures are not required, being obtained direct solutions. The principle of superposition is valid, and the results for various load cases can be factored and combined. Also, the material constants in the linear analysis of isotropic materials are only two, thus leading to very simple stress – strain relationships.

The majority of structural systems are analysed using the linear theory. In numerous cases, however, the structural behaviour is mainly characterised by nonlinear effects that must be included in the analysis. The nonlinear effects can be classified as follows:

- Material nonlinearity, when the material can be characterised by a nonlinear stress – strain relationship;
- Geometric nonlinearity, when the deformation caused by the applied loads cannot be considered small; in these case, the small deflection theory is not valid, being required the use of the large deflection theory;
- Boundary nonlinearity, when the boundary conditions are changing with the load levels.

The nonlinear analysis substantially differs from the linear analysis. The principle of superposition is not valid and only one load case can be treated at a time. Also, the loading history depends on the sequence of load application, as well as on the presence of initial stresses, such as residual stresses or prestressing [1].

The nonlinear analysis can be performed using the Mechanical Event Simulation (MES), by watching a design moving in a dynamic event, such as buckling, swinging, rotation, or oscillation. MES combines the dynamics of rigid and flexible body with the possibilities of nonlinear stress analysis.

As a result, MES can simultaneously analyse mechanical events involving large deformations, nonlinear material properties, kinematic motion and forces caused by that motion, and then predict the resulting stresses.

Some of the main advantages of MES are the need to make fewer assumptions. With MES, there is no need for elaborate hand calculations, interpretation of results or experiments to determine the equivalent loading. The fewer assumptions that must be made reduce the chance of errors [2].

2. Geometric design of the gear

The cylindrical gears are machine parts with widespread use in practice. The designing of cylindrical gears, i.e. their sizing and checking, is carried out in accordance with the fatigue strength conditions provided in ISO 6336, the fatigue being caused by the contact stress on the gear teeth flanks and the bending at the base of teeth. At the analysis using the Finite Element Analysis with MES – nonlinear material model, the obtained results are: stresses, nodal displacements and strains at different areas or points of the geometric model, at different time steps of the kinematic cycle.

The studied gear was designed using Autodesk Inventor Professional software [3-4], based on the initial parameters shown in Table 1, and the results (the values of resulting safety factors), for the two stress cases, are shown in Table 2. It is mentioned that the values of these safety factors are higher than the allowable values. The geometric model of the gear resulting from the computer assisted design is shown in Figure 1.

Table 1. Initial parameters required for gear design.

Gear ratio	Module (mm)	Centre distance (mm)	Helix angle (deg)	Power (kW)	Gear 1 speed (rpm)	Material
2.5	2.75	80	0	12	1500	EN C50

Table 2. Design results.

Gear	Factor of safety from pitting	Factor of safety from tooth breakage	Static safety in contact	Static safety in bending
Gear 1	1.32	2.621	1.23	5.549
Gear 2	1.24	2.679	1.28	5.61

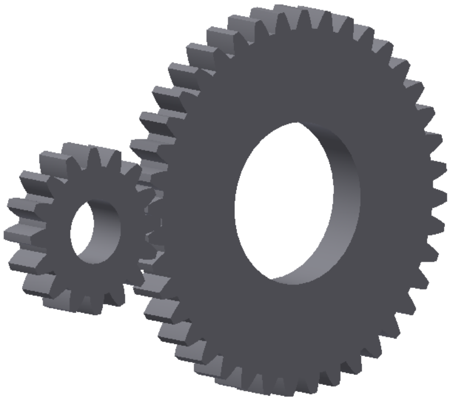

Figure 1. 3D model of the gear.

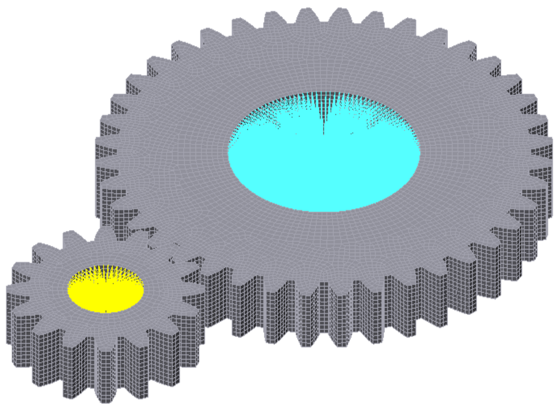

Figure 2. Discretized 3D model.

3. Gear analysis using Mechanical Event Simulation (MES)

3.1. Initial aspects
The analysis using the Finite Element Analysis with MES – nonlinear material model – was carried out using the Autodesk Simulation Mechanical software, going through the specific steps of such an analysis, i.e. discretization of the model, modelling of the rotation couplings, establishing the surface contact between two gear wheels, shaping the constraints and loads, performing the actual analysis (visualization and interpretation of results), and finally generating the analysis report [5].

The geometric model discretization was achieved by establishing an average size for the finite elements and keeping the resulted brick-type element.

The kinematic rotation couplings have been modelled in the bores of the two gear wheels, and the constraints (DOF), the prescribed displacements, and the applied loads are going to be defined in the central nodes. The type of default element for the kinematic couplings is truss, which cannot takeover any kind of moments and therefore turns into beam-type elements. The model obtained so far in this stage can be seen in Figure 2.

3.2. Modelling of surfaces in contact
When transferring loads between the elements of an assembly, the nodes of the contact surfaces are usually interconnected. Following the discretization, the contact between the pairs of nodes is a bonded-type one; therefore, the two gear wheels are considered stiffened together, the engaging movement being no longer possible. In this regard, it is necessary to define a surface-to-surface contact between the surfaces afferent to the teeth in contact at a given time.

This was made in MES by creating a surface-to-surface contact pair, since the surfaces may come into contact with each other during the analysis or are initially in contact, but in the next moment they can separate. The contact pair is made of two distinct surfaces of the two gear wheels, which have been obtained by correspondingly joining the surfaces of the engaging teeth at a given time, as can be seen in Figure 3 and Figure 4 [6-7].

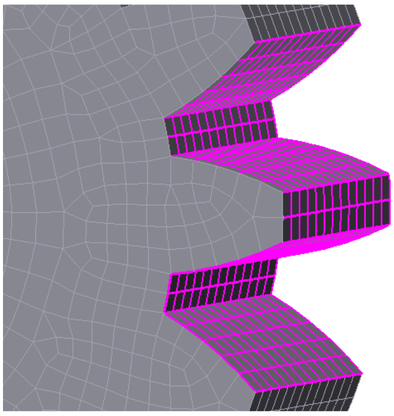

Figure 3. Contact surface for the gear wheel 1.

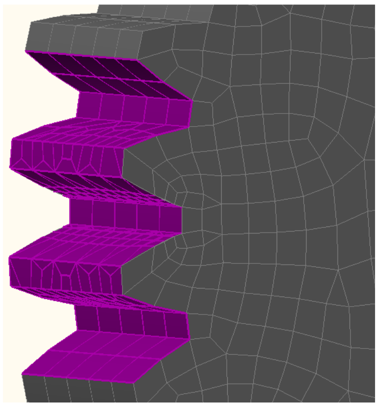

Figure 4. Contact surface for the gear wheel 2.

3.3. Defining the constraints and loads
Defining the constraints means defining the degrees of freedom for the assembly components, i.e. specifying the possibilities of movement that are restricted. Since the two gear wheels, along with the two associated kinematic couplings, execute only rotational movements, they will have only one degree of freedom, i.e. rotational motion around the axis perpendicular to the engagement plane, the other five possibilities of movement being thus restricted, as shown in Table 3. The constraints were defined in the central nodes of the two kinematic couplings.

Table 3. Nodal general constraints.

Part	Tx	Ty	Tz	Rx	Ry	Rz
Kinematic coupling 1	Yes	Yes	Yes	Yes	Yes	No
Kinematic coupling 2	Yes	Yes	Yes	Yes	Yes	No

The rotational movements of the gear components were defined by their rotational speeds around the corresponding axis (Z), for the pair consisting of the driven toothed wheel and its associated kinematic coupling, taking into account the value of the gear transmission ratio (2.5), as shown in Table 4. The rotation centre for the pinion and its associated coupling has been defined in the origin of the reference system, and the value of the distance between the axes (80 mm) in the direction of Y axis was taken into account for the driven gear assembly.

Table 4. Speed values.

Part	Translational velocity (m/s)			Rotational velocity (rpm)			Centre of rotation (mm)		
	X	Y	Z	X	Y	Z	X	Y	Z
Gear 1	0	0	0	0	0	–1500	0	0	0
Kinematic coupling 1	0	0	0	0	0	–1500	0	0	0
Gear 2	0	0	0	0	0	600	0	80	0
Kinematic coupling 2	0	0	0	0	0	600	0	80	0

The loads afferent to a nonlinear analysis may vary depending on the duration of the analysis, these being controlled by the load curves. The multiple loads can follow the same loading curve, but in most cases the individual loads can be assigned to the various load curves. For analysis, two loads assigned to two distinct load curves have been defined [8].

For the first load, a prescribed nodal displacement was applied in the central node of the kinematic couplings associated to the pinion, displacement which at any time of the analysis will depend on the assigned parameter (a rotation), as well as on the current multiplier given by the load curve 1, as seen in Figure 5.

The second load is a nodal moment, defined in the central node of the kinematic coupling associated to the driven toothed wheel by size (191000 Nmm) and direction (around the axis of rotation – Z), its variation during the analysis being described by the load curve 2, as seen in Figure 6.

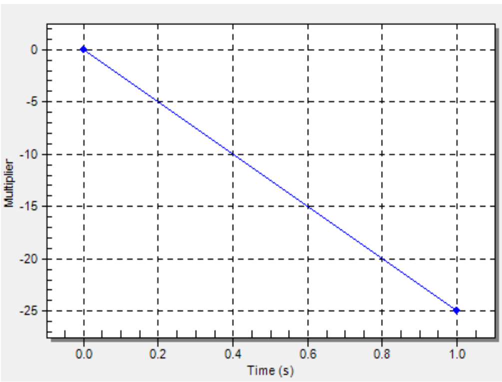

Figure 5. Nodal prescribed displacement – Load case 1.

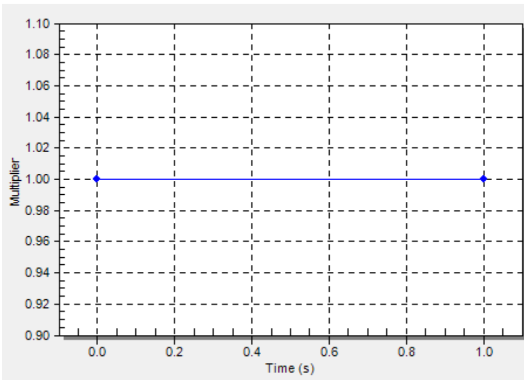

Figure 6. Nodal moment – Load case 2.

3.4. Gear Analysis using MES

In order to carry out the gear analysis, we must define the parameters. The Mechanical Event Simulation (MES) is defined by two parameters, i.e. duration of the event and number of time steps.

The duration of the event is the period of time during which a pair of teeth are in contact, corresponding to a complete rotation of the pinion, and was determined according to its speed. For analysis, we established a number of 10 time steps, the duration of the event being divided into equal time intervals, for which we obtained successive results [9]. The established analysis parameters can be seen in Table 5.

Table 5. Parameters obtained when using MES for analysis.

Duration (s)	Number of time steps
0.0025	10

The obtained results are presented below for three time steps, i.e. the beginning, the middle and the end of the event. We intended to show the Von Mises stress distribution for the gear unit, but also individually for the contact areas of the teeth flanks of the two gear wheels, strains, nodal displacements and safety factors.

- Step 1 – Time: 0.00025 s

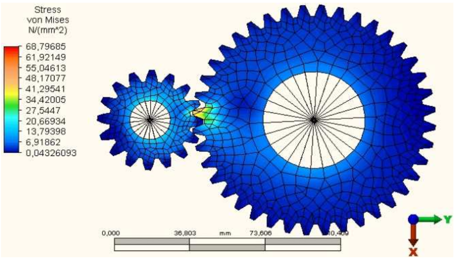

Figure 7. Von Mises stress in gear unit at time step 1.

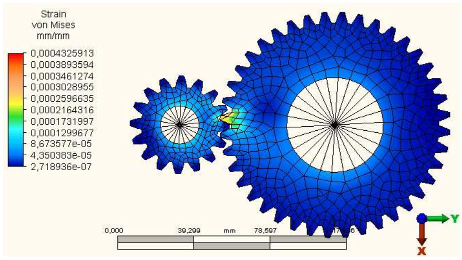

Figure 8. Strain at time step 1.

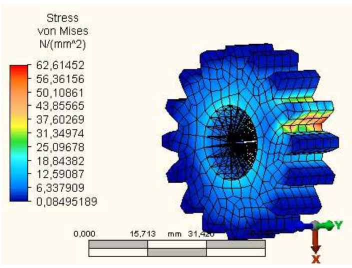

Figure 9. Von Mises stress in the contact area of the pinion at time step 1.

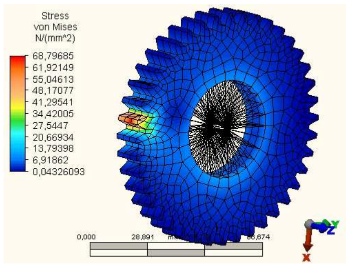

Figure 10. Von Mises stress in the contact area of the driven gear at time step 1.

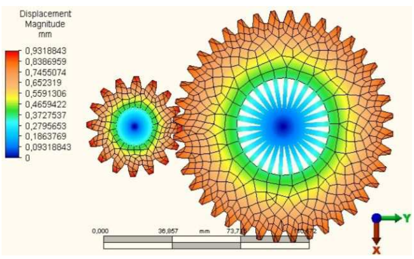

Figure 11. Nodal displacement at time step 1.

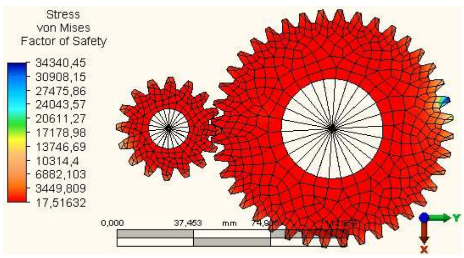

Figure 12. Safety factor at time step 1.

- Step 5 – Time: 0.00125 s

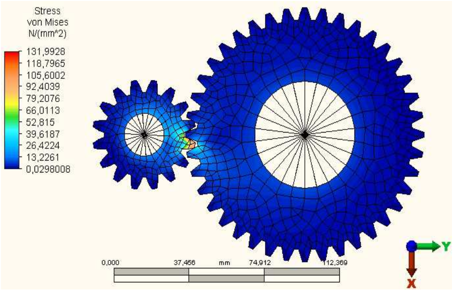

Figure 13. Von Mises stress in gear unit at time step 5.

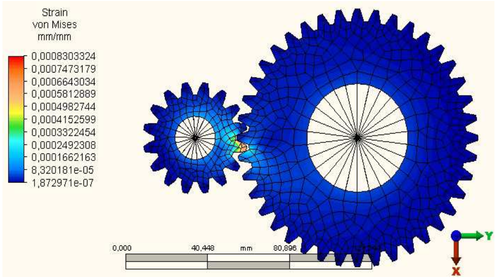

Figure 14. Strain at time step 5.

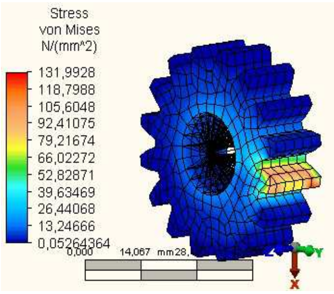

Figure 15. Von Mises stress in the contact area of the pinion at time step 5.

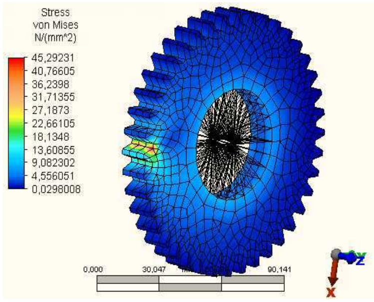

Figure 16. Von Mises stress in the contact area of the driven gear at time step 5.

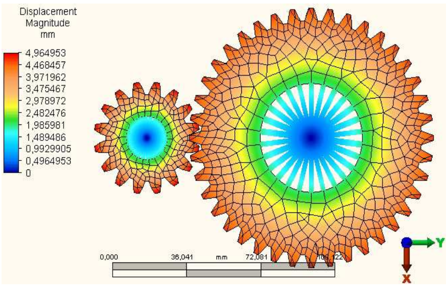

Figure 17. Nodal displacement at time step 5.

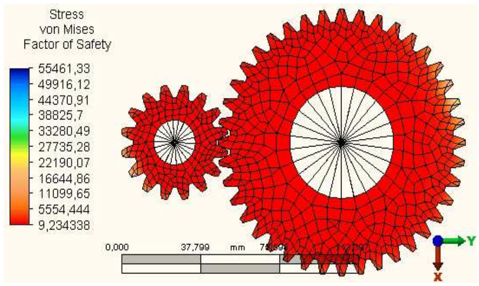

Figure 18. Safety factor at time step 5.

- Step 10 – Time: 0.0025 s

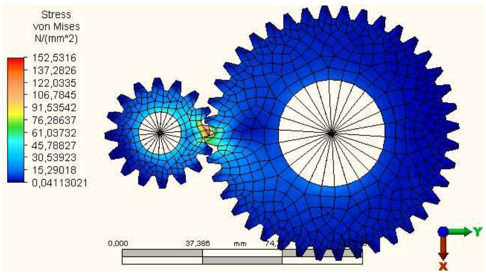

Figure 19. Von Mises stress in gear unit at time step 10.

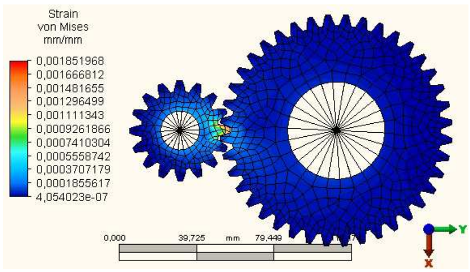

Figure 20. Strain at time step 10.

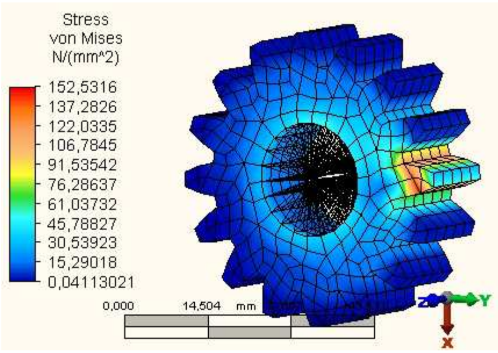

Figure 21. Von Mises stress in the contact area of the pinion at time step 10.

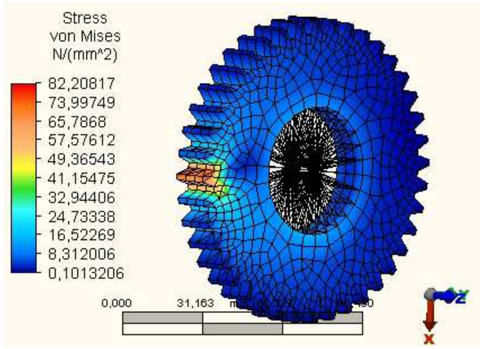

Figure 22. Von Mises stress in the contact area of the driven gear at time step 10.

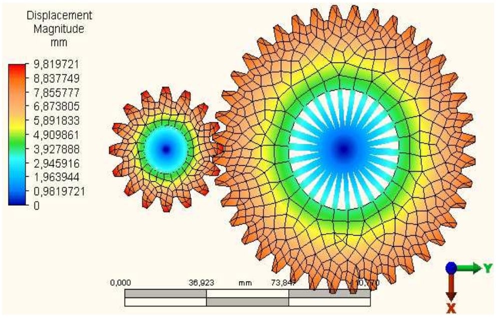

Figure 23. Nodal displacement at time step 10.

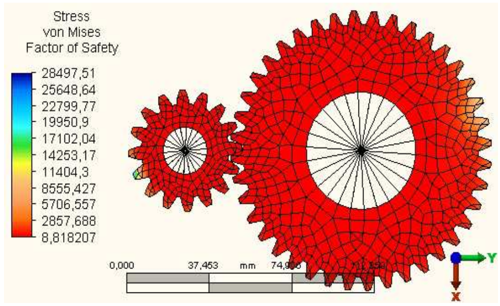

Figure 24. Safety factor at time step 10.

For a correct interpretation of the results presented above, they can be also viewed in graphical form, as a variation of the size studied during the event, in the defined time steps. For example, in figures 25 and 26 it shows von Mises tension maximum values variations and nodal displacements in the 10 time steps.

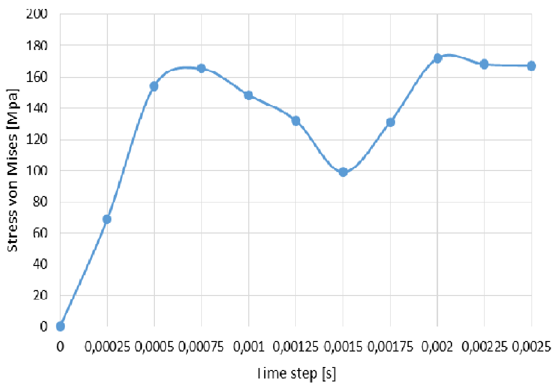

Figure 25. Maximum values of stress von Mises in different time steps.

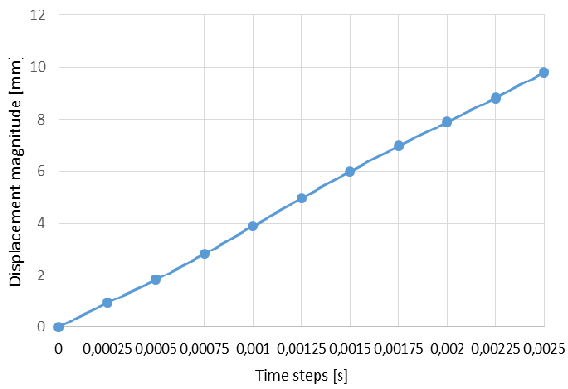

Figure 26. Maximum values of displacement magnitude in different time steps.

Analysing the results presented in Figures 7 - 26, for the above-mentioned time steps, we can specify the following issues:

- For each time step, the Von Mises stresses have maximum values in the area where a pair of teeth is in contact at a given time, being below the values allowable for that gear wheel material;

- The stresses on the flanks have maximum values either in the peak area, or at the base of the teeth, depending on their relative position at a given moment of engagement;
- The relative elongations have low values, within acceptable limits, with a growing trend during the event simulation;
- The nodal displacements are also increasing during the event simulation, the values being high, specific to a nonlinear analysis;
- The safety coefficients have values above the recommended limits, the lowest value that corresponds to the maximum stress at time step 10 being 8.81.

4. Conclusion

On the basis of the above mentioned issues, the finite element method analysis of cylindrical gears can be considered a useful method to be applied in designing. The use of MES with nonlinear material model, by properly determining the parameters of the analysis, enables us to obtain results at different moments of an undergoing event. In this paper, the obtained results showed the distribution and the extreme values of stresses, displacements, relative elongations, and safety factors.

References

[1] Spyrakos C C and Raftoyannis J 1994 *Linear and Nonlinear Finite Element Analysis in Engineering Practice*, Algor Inc. Publishing Division
[2] Simulation Mechanical, *Mechanical Event Simulation 2015*, https://www. knowledge.autodesk. com/support (accessed on December 2017)
[3] Dolga L, Revencu M, Maci C A and Giuchici M 2004 *Parametric and Feature Based Modelling with Applications in Catia and Inventor*, Politehnica Timișoara
[4] Autodesk Inventor, *Design Spur Gears 2017*, https://www.knowledge. autodesk.com/support (accessed on December 2017)
[5] Cioată V G 2008 *Determining the Machining Error Due to Workpiece-Fixture System Deformation using the Finite Element Method*, 19^{th} International DAAAM Symposium, Trnava, Slovakia, October 22-25, pp 253-255
[6] Kim N-H and Bhavani V S 2009 *Introduction to finite element analysis and design*, Wiley
[7] Simulation Mechanical, *Surface to Surface Contact 2017*, https://www. knowledge.autodesk.com/ support (accessed on December 2017)
[8] Cioată V G, Kiss I, Alexa V and Rațiu S A 2017 The Optimization of the Position and the Magnitude of the Clamping Forces in Machining Fixtures, *IOP Conference Series: Materials Science and Engineering* **200** 012015
[9] Laniado-Jacome E 2011 Numerical Model to Study of Contact Force in a Cylindrical Roller Bearing with Technical Mechanical Event Simulation, *Journal of Mechanical Engineering and Automation* **1**(1) 1-7