American Journal of Mechanical Engineering, 2017, Vol. 5, No. 6, 263-268
Available online at http://pubs.sciepub.com/ajme/5/6/6

DOI:10.12691/ajme-5-6-6

Deformation and Strength Properties of Elastic Members of High Precision Positioning Equipment

František Trebuňa, Jozef Bocko, Miroslav Pástor*, Pavol Lengvarský

Department of Applied Mechanics and Mechanical Engineering, Technical University of Košice,
Faculty of Mechanical Engineering, Letná 9, Košice, Slovakia
*Corresponding author: miroslav.pastor@tuke.sk

Abstract During the loading of real machine in operational conditions undergo the machine parts deformations. If the deformation is in elastic area, after unloading the shape of machine element comes back to its original state. In case, the loading level crosses yield point, the machine part undergoes plastic deformations and the relations describing material behavior are changed. In that case in real structures hardening of material occurs and the deformation is irreversible. Such behavior is described by models of material hardening. In principle, there exist three types of such models - isotropic hardening, kinematic hardening, and mixture of previous two, combined hardening.

Keywords: *elastic member, material hardening, stress analysis, deformation, finite element method*

Cite This Article: František Trebuňa, Jozef Bocko, Miroslav Pástor, and Pavol Lengvarský, "Deformation and Strength Properties of Elastic Members of High Precision Positioning Equipment." *American Journal of Mechanical Engineering*, vol. 5, no. 6 (2017): 263-268. doi: 10.12691/ajme-5-6-6.

1. Introduction

The paper deals with analysis of behavior of machine parts under loading in elastic and plastic area by the finite element method. The numerical simulation was realized on elastic members of high precision positioning system. The equipment serves for positioning of frames (bolsters) to which a beam with further technology systems is connected. Positioning of beams is accomplished in five degrees of freedom. Only one direction is fixed – axial direction of beam. Schema of positioning equipment is given in Figure 1. Numerical modelling for analysis the equipment is used because of complicated shapes of machine parts as well as complicated rheological relations describing plastic processes [1].

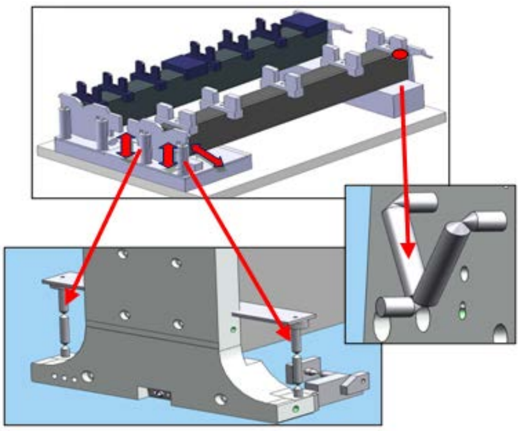

Figure 1. Schematic representation of positioning equipment with elastic members

The material models used were the following: ideal elasto-plastic (Prandtl-Reuss), solid-plastic, as well as models with hardening (isotropic and kinematic) [2,3]. However, not all models will be described in this paper, because of huge amount of results of accomplished computations.

2. Strength and Deformation Analysis of Elastic Member of Positioning Mechanism

In order to verify stiffness and strength properties of proposed mechanism for positioning of heavy objects, the main support members and their parts were analyzed by number of computations. The computations were realized in elastic as well as in plastic area by the finite element method. The numerical computations by the finite element methods were used not only for analysis of structural behavior, but also for the optimization process [4,5,6]. In the paper is described stress and deformation analysis of elastic member (Figure 2) which is a part of equipment for precise positioning of heavy objects.

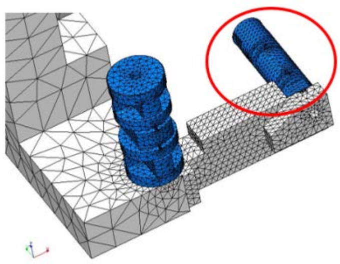

Figure 2. Elastic member of positioning equipment

The computations were realized, as was mentioned above, in elastic as well as plastic area by FE system SolidWorks. Stiffness analysis based on computation of reaction forces and moments in elastic joints was realized by response of given machine part on displacement by 1 mm in given direction x, y, z, respectively or in case of bending and torsion on rotation by angle 1 deg around those axes [3].

In order to model behavior of the whole mechanism, the stiffness of the joints had to be found. As the stiffness is defined as a force (or moment) that belongs to unit deformation, the reaction forces and moments were computed for prescribed unit deformations. The orientation of axes with respect to the joint body is shown in (Figure 3). Elastic joint is made of steel 34CrNiMo6 [7].

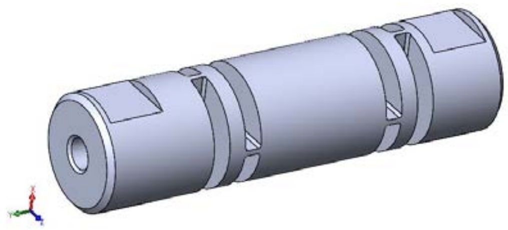

Figure 3. Orientation of elastic member with respect to the used coordination system

Reaction forces and moments on one end of elastic member were established for the prescribed displacement of its second end by 1 mm in direction x, y, z, respectively or in case of bending or torsion as response to rotation by angle 1 deg around these axes. The mesh of finite elements, boundary conditions and directions of prescribed displacements for tensile loading are seen in Figure 4.

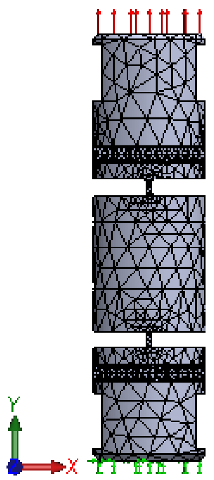

Figure 4. Boundary conditions for the elastic member under tensile load

The results of reaction computations are given in Table 1 to Table 6.

Table 1. Reactions for displacement in direction x (1 mm)

Rx [N]	-309
Ry [N]	16,743
Rz [N]	-0,10251
Mx [Nm]	-0,018162
My [Nm]	-0,0064197
Mz [Nm]	10,8

Table 2. Reactions for displacement in direction y (1 mm)

Rx [N]	0,41182
Ry [N]	-98131
Rz [N]	0,06982
Mx [Nm]	-0,08296
My [Nm]	0,010431
Mz [Nm]	0,021495

Table 3. Reactions for displacement in direction z (1 mm)

Rx [N]	0,051788
Ry [N]	-7,8326
Rz [N]	-150,37
Mx [Nm]	-5,291
My [Nm]	0,0038592
Mz [Nm]	0,0022783

Table 4. Reactions for rotation around x axis (1 deg)

Rx [N]	0,466649804
Ry [N]	10,15829777
Rz [N]	-0,017889392
Mx [Nm]	-0,994970312
My [Nm]	-0,00135089
Mz [Nm]	-0,001413905

Table 5. Reactions for rotation around y axis (1 deg)

Rx [N]	0,485981903
Ry [N]	0,593534702
Rz [N]	-0,37417866
Mx [Nm]	-0,053639373
My [Nm]	-11,65424436
Mz [Nm]	-0,013951404

Table 6. Reactions for rotation around z axis (1 deg)

Rx [N]	1,264135964
Ry [N]	-37,04169736
Rz [N]	-0,282292279
Mx [Nm]	0,011116611
My [Nm]	-0,006184177
Mz [Nm]	-1,007363801

After linear analysis, the computations in plastic area were accomplished. Plasticity occurs due to overloading of the structure. For a one given state of such loading the field of displacement in examined member is given in (Figure 5).

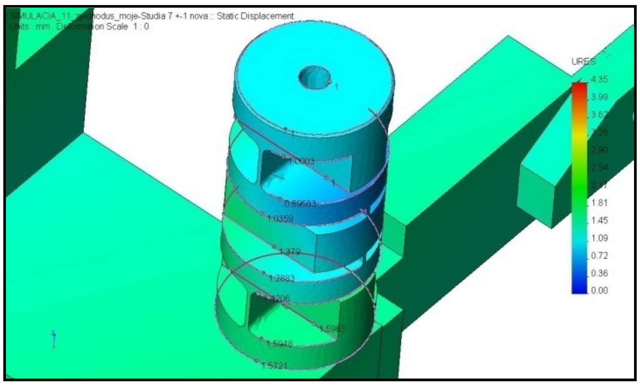

Figure 5. The displacement field of elastic member of positioning equipment

Tensile loading followed by unloading

The elastic member was loaded also by loading that leads to plastic deformations. Because the manipulations with objects can lead also to unloading, for the computation was used loading-unloading process, where the stresses during loading exceed yield point of given material. The behavior of joint member was analyzed with help of isotropic (HF0) as well as kinematic (HF1) hardening model. The parameters of computations were:

- force control method,
- iteration method: Newton-Raphson method,
- integration method: Newmark method.

In Figure 6 is given time-dependent chart of loading (unloading) force for these computations.

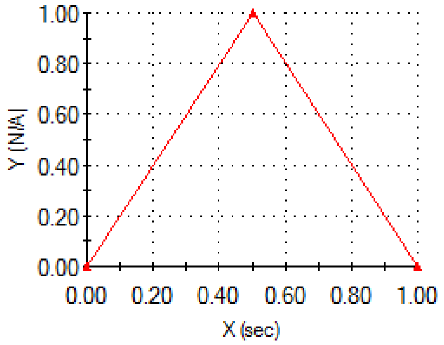

Figure 6. Time-dependent chart of load force

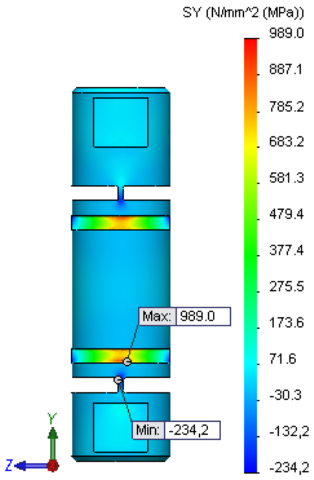

Figure 7. Normal stresses in Y direction (HF0)

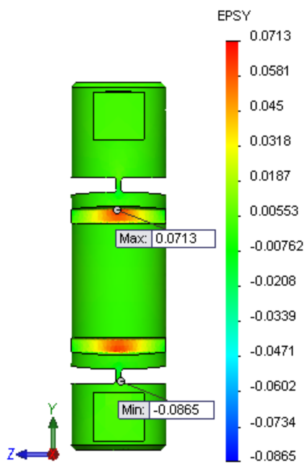

Figure 8. Strains in Y direction (HF0)

The results of numerical simulations for the model with isotropic hardening (HF0) under tensile loading are given in Figure 7 to Figure 10. In Figure 7 is shown field of normal stresses in y direction for the maximum loading force and in Figure 8 are given strains components with respect to the same axis.

The details of stress concentrators are shown in Figure 9. In Figure 10 is given graph of normal stress at the location of stress concentration during whole loading cycle.

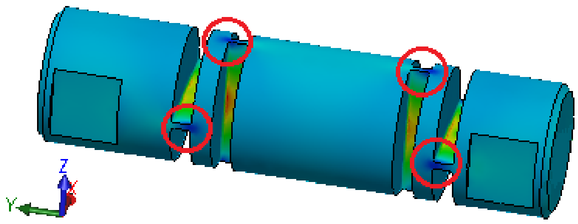

Figure 9. Areas with concentrated stress concentrators

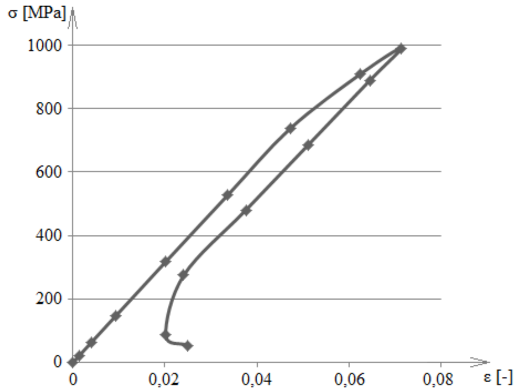

Figure 10. Dependency of stress on strain under tensile loading in one chosen point (HF0)

The results of numerical simulations for the model with kinematic hardening (HF1) under tensile loading are given in Figure 11 to Figure 13.

Figure 11 represents field of normal stresses in y direction for the maximum loading force and in Figure 12 are given strains components with respect to the same axis. In Figure 13 is given graph of normal stress at the location of stress concentration during whole loading cycle. The computations for the models with isotropic and kinematic hardening lead to almost identical results.

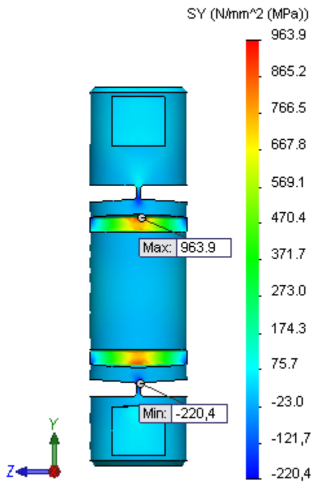

Figure 11. Normal stresses in Y direction (HF1).

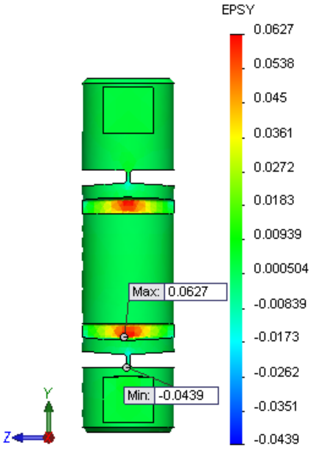

Figure 12. Strains in Y direction (HF1)

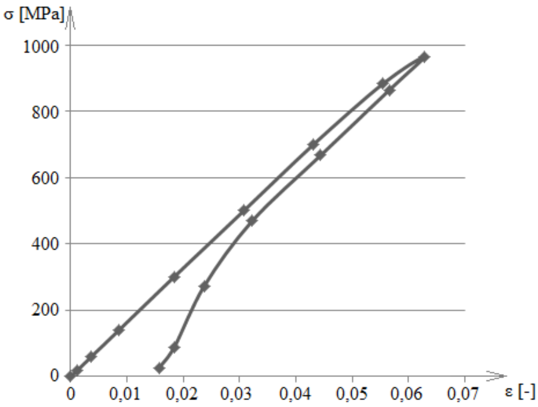

Figure 13. Dependency of stress on strain under tensile loading for one chosen point (HF1)

Torque loading followed by unloading

In the following is described stress and deformation analysis for torque loading of elastic member around y axis. Again, the analyses were accomplished for isotropic and kinematic hardening. The computed machine part with used finite element mesh, boundary conditions and loading is given in Figure 14.

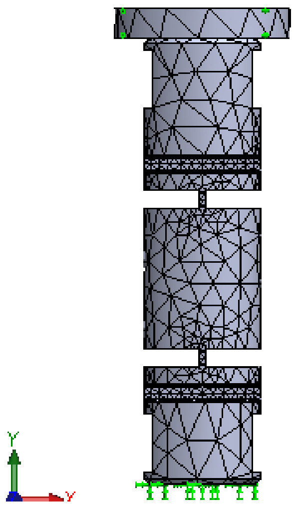

Figure 14. Boundary conditions for the elastic member under torque loading

The chart of time-dependency of loading is given on Figure 15. The parameters of computations were:

- force control method,
- iteration method: Newton-Raphson method,
- integration method: Newmark method.

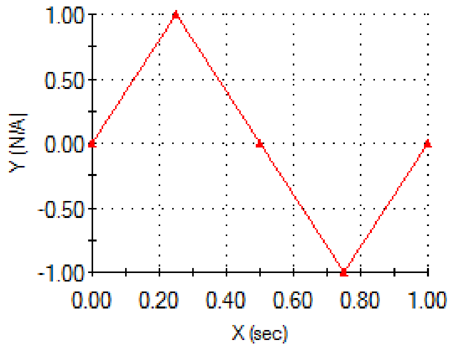

Figure 15. Time-dependent chart of the loading force

The results of numerical simulations for isotropic hardening are shown on Figure 16 to Figure 18 (HF0).

Figure 16 represents field of equivalent von Mises stresses for the highest loading and Figure 17 shows the strains.

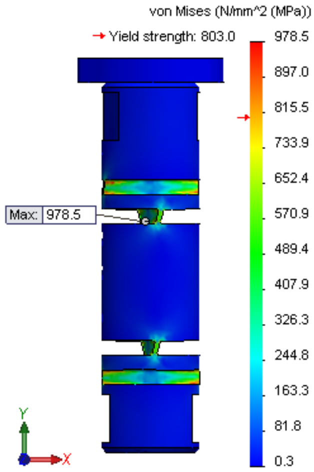

Figure 16. Field of equivalent von Mises stresses (HF0)

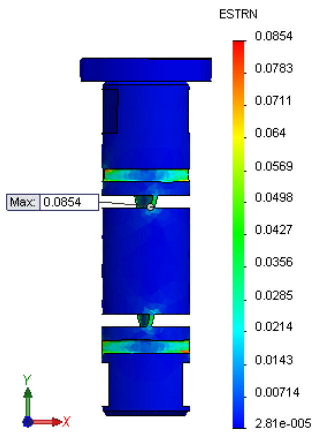

Figure 17. Field of strains (HF0)

The graph of dependencies of equivalent stress on strain in given point is shown in Figure 18. The first cycle is given in grey color, the second one is drawn by red line.

The results of numerical simulations for kinematic hardening are shown on Figure 19 to Figure 21 (HF1).

In Figure 19 is given field of equivalent von Mises stresses and in Figure 20 is a field of strains for maximum loading.

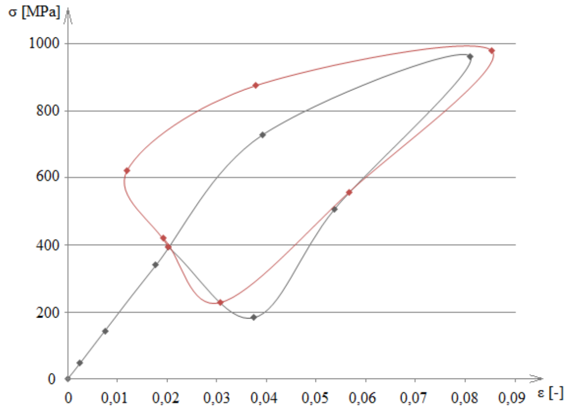

Figure 18. Dependency of stress on strain under torque loading (HF0)

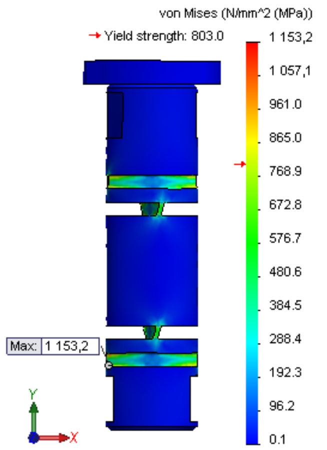

Figure 19. Field of equivalent von Mises stresses (HF1).

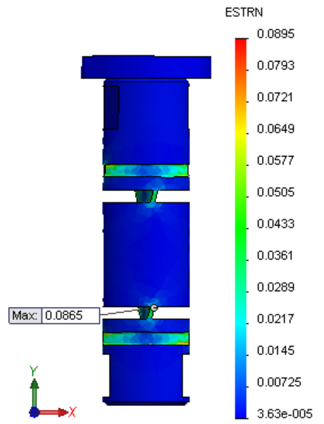

Figure 20. Field of strains (HF1)

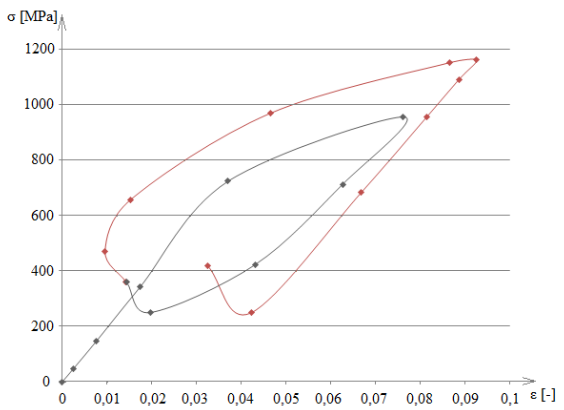

Figure 21. Dependency of stress on strain under torque loading (HF1)

The graph of dependencies of equivalent stress on strain in given point is shown in Figure 21. As before, the first cycle is given in grey color, the second one is drawn by red line.

3. Conclusions

In the paper is described stress and deformation analysis of one from three elastic members proposed for high precision positioning equipment. The computations were accomplished for different types and magnitudes of loadings. The elastic member was loaded in elastic and plastic range using isotropic as well as kinematic hardening. The types of loadings were: tensile loading, torque loading and in elastic area for computation of stiffness also bending. The computations in plastic areas were accomplished for two different prescribed loading cycles that can be in short described as loading and unloading. The main interest of authors was focused to deformation and stress characteristics of given elastic joint, i.e. displacements, strains, stresses, residual stresses after certain loading cycles and so on. General conclusions were established for two main types of loadings – tensile loading and torque. From the computations can be stated fact that the results for both types of hardening methods used are similar.

The results of numerical simulation of elastic members were consequently used for realization of kinematical analysis of movement of supporting system of heavy objects. The aim of such analysis was to verify range of movement of reference points on positioning axes of the object which is given in supports of beam and to find dependencies of their displacements on the movements of actuators. The problem is that this system is statically undetermined and accordingly the stiffnesses of joining elastic members have to be used for the computations. However, the kinematic analysis of the whole equipment exceeds the aims and scope of this paper.

Acknowledgements

This paper was supported by project Stimuli No. Req-00169-0003 Research and development of intelligent mobile robotic platforms and positioning systems with high precision for using in research, development and industry.

References

[1] Vano, M. J. – Sanchis, J. E. – Bocko, J. *Mechanical Behaviour of Materials – Simulation Problems*. Univ Politécnica Valencia, 2013. ISBN 9788490481486

[2] Bocko, J. – Segľa, Š. *Numerical Methods of Mechanics of Solid and Elastic Bodies* (in Slovak). Košice: SjF TU, 2016. 248 p.

[3] Tomko, M. *Material Models with Isotropic and Kinematic hardening and their Application*: *Diploma work* (in Slovak). Košice: TUKE, SjF, 2017. 82 p.

[4] Elakkad, A. – Bennani, M. A. – Mekkaoui, J. EL and Elkhalfi, A. *A Mixed Finite Element Method for Elasticity Problem*. International Journal of Advanced Computer Science and Applications(IJACSA), 4(2), 2013.

[5] Zienkiewicz, O.C., Taylor, R.L., Zhu, J.Z., *The Finite Element Method: Its Basis and Fundamentals*, Butterworth-Heinemann, United Kingdom, 2013.

[6] Bower, A., F. *Applied Mechanics of Solids*. CRC Press, Taylor & Francis Group, Boca Raton, 2010

[7] Trebuňa, F. – Šimčák, F. *Toughness of Elements of Mechanical Systems* (in Slovak). Košice: Emilena, 2004. 980 p. ISBN 80-8073-148-9.