Proceedings of the 17th International Conference on Manufacturing Systems – ICMaS
ISSN 1842-3183
University POLITEHNICA of Bucharest, Machine and Manufacturing Systems Department
Bucharest, Romania

FINITE ELEMENT ANALYSIS OF STATIC, VIBRATION AND FATIGUE BEHAVIOR OF THE ECCENTRIC SHAFT COMPONENT OF A MOTO-MOWER B62

Nicolae COFARU, Adrian PASCU, Valentin OLEKSIK

Abstract: *This paper presents a study through the finite element method applied to an eccentric shaft component of a moto-mower for grass mowing. During the shaft analysis we observed frequent tearing in the eccentric zone. For this study, we carried out three different types of analyses: static, modal and fatigue. The results were focused on the determination of the equivalent von Mises stress and displacements (arrows), but also on the safety factor for the static analyses, on the own vibration modes for the modal analyses and on the safety factor for the fatigue analyses.*

Key words: *numerical simulation, finite element method, static analysis, modal analysis, fatigue analysis.*

1. INTRODUCTION

The increase the retooling degree led to the appearance of various highly automated substitution devices that replaced the classic and traditional equipment that was used in Romania for different agricultural works in farms (mowing, milking of the animals, cultivation of ground, forest exploitation).

The increase of the automation degree of these products brings many advantages to the users:

- significant decrease of work time;
- easy to exploit;
- decrease in the number of malfunctions that use to appear in the product from the previous generations and implicitly, a diminish of the maintenance costs;
- decrease in number of people involved in this performing this work;
- increase in the exploitation productivity;
- substantial decrease of the physical effort;
- decreased risk of accidents.

The above considerations, corroborated with the aging population in the rural areas due to the extensive migration of the young people to the urban or international areas, lead to the necessity of performing the agricultural works using professional equipment.

Replacing the manual mowing with the mechanic mowing using a Bertolini B62 moto-mower falls in the same context.

The B62 moto-mower (Fig. 1) is a professional moto-mower with two forward speeds and two rear speeds; it is also reversible and other accessories can also be adapted to it. The technical characteristics of this moto-mower are: a Honda engine type with 6.5 HP, oil-drowned transmission, two forward speeds and two rear speeds, mowing area length of 115 cm.

In this paper we propose an analysis of one of the machine’s components, which often presents defects and is put out of use.

These “out of use”, apart from the technical reasons, are due to the lack of experience on the part of the users (unaccustomed to such equipments): regarding handling the tool, its maintenance and periodic tuning, as well as proper preparation of the areas to be mowed.

Therefore we believe it is of interest to study and possibly constructively optimize the different parts of such a machine.

Figure 2 shows the active part of the moto-mower, respectively the motion transmission from the equipment’s gearbox to the actives knifes. The main element of this transmission is the main axis number 10.

Fig. 1. General view of the B62 moto-mower.

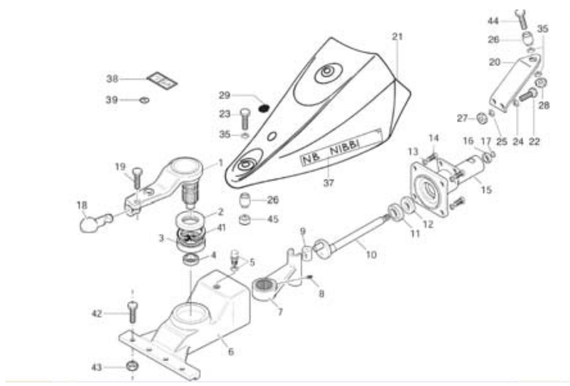


Fig. 2. Exploded view of the B62 moto-mower.

Fig. 3. Defects that appear on the eccentric shaft.

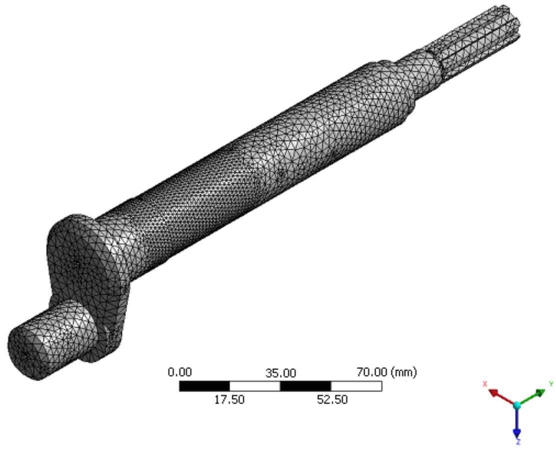


Fig. 4. The meshed model.

This element is an intensely stressed part and exhibits most of the “out of use”, usually through tearing.

Functionally, the 10 axis captures the rotation motion through a groove assemblage and transmits through an eccentric to the special bushed bear number 9, which slides inside the sideway number 8. The sideway 8, united with the crown gear 7, will execute an oscillatory motion transmitted to the arm 1, which transmits it through the mark 6 to the mobile knife of the mower.

The support-centering, axial positioning and axis dilation functions are taken over by the roll bearings 11 and 12.

Many “out of use” of the axis in the eccentric area (Fig. 3) resulted through the current practice, through its effective tearing.

Therefore we propose an analysis through the finite element method in order to elucidate the causes of these defects that appear in the eccentric shaft. The defects could appear either due solicitation that exceeds the static resistance capacity of the material, either due to vibrations or to material fatigue.

2. NUMERICAL SIMULATIONS

The geometric model was achieved using the SolidWorks software, which was then exported to finite element analysis software. For this analysis we used the Ansys software, where the spatial structure of the axis was discretized using the finite elements of tetrahedral shape. For meshing the model of the eccentric shaft we used the "free mesh" method. The meshed model is shown in Fig. 4. During this analysis the eccentric shaft was of quality carbon steel with σ_a = 250 MPa.

2.1. Static analysis of the eccentric shaft

The goal of the static analysis is to determine the stress and strains states, as well as the safety factor for loading the model in a static regime, created by the torsion moment with a magnitude of 50 Nm. This moment value represents the resisting moment of the moto-mower [3, 4].

The analysis of the axis deformation state consists of determining the deformation tendency, through measuring the total nodal displacements (Fig. 5). The maximum value of the nodal displacement appears in the eccentric area and has a magnitude of 0.015 mm.

For the analysis of the stress distribution in the analyzed axis, we used the equivalent von Mises stress (σ_{ecv}) whom the Ansys software calculates it as a quadratic average of normal stresses in the bottom, middle and top of the finite elements, respectively. Analyzing the map of the equivalent stress distribution σ_{ecv}, one can observe that there is a stress concentration point in the area connecting the eccentric shaft and the flange on which it resides. The maximum value of these stresses is 92.63 MPa for the equivalent von Mises stress (Fig. 6). The maximum value of this stress is smaller than the admissible stress of the material of this shaft. In this case, the value for safety factor is 2.7.

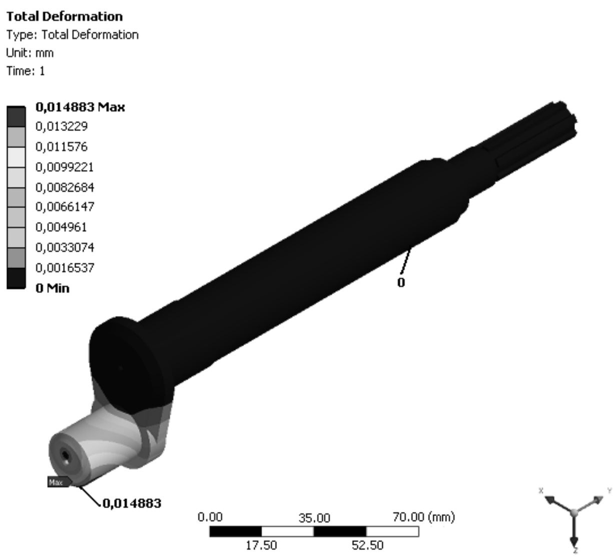


Fig. 5. Distribution of nodal displacements [mm].

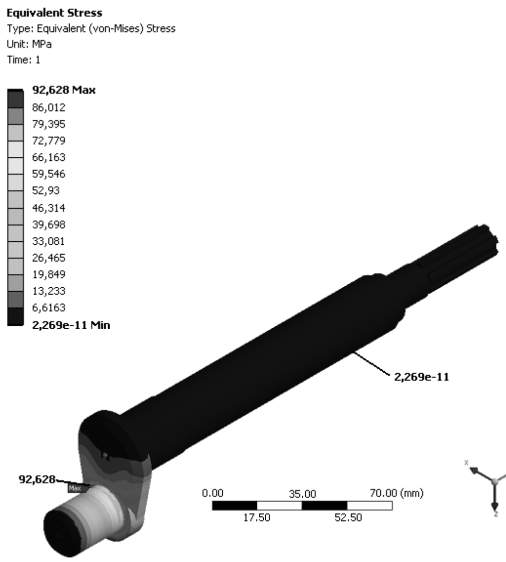


Fig. 6. Distribution of the equivalent von Misses stress [MPa].

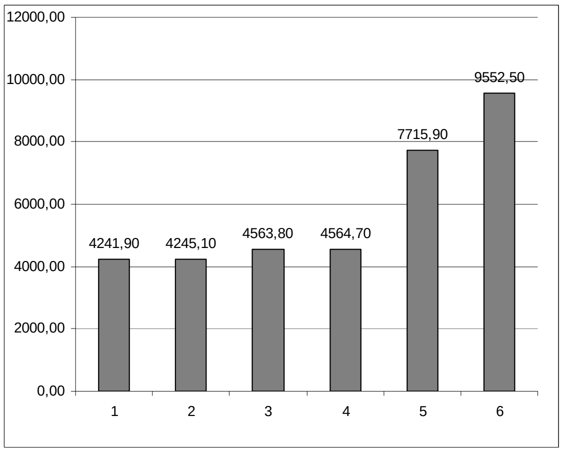


Fig. 7. The values of the first six frequencies [Hz].

2.2. Modal analysis of the eccentric shaft

Modal analysis is a possibility to study the behavior of elastic systems, where the focus is on determining the first own frequencies of the structure, emphasizing some of the weak points of the structure, as well as determining the deformability tendencies in the dynamic domain.

When modal analysis is performed with the Ansys software it is a linear analysis, and the method used to extract the own models is the iteration method on subspaces, which uses the generalized Jacobi algorithm. We chose this method because the accuracy of the results is very good, as this method works with the stiffness and whole mass matrices, thus avoiding the necessity for the user to choose master degrees of freedom.

The normalization of the vibration modes is performed in relation to the masses matrix, in order to enable further analysis with the response to a harmonic excitation.

From the modal analysis, we took the first six own vibration modes. The own frequencies we obtained following the analysis are shown in Fig. 7.

2.3. Harmonic analysis of the eccentric shaft

The dynamic study of the elastic structure, using the finite element method, can be considered an extension of the static analysis for problems where the time dependence of the spatially discretized models through the finite element method is considered.

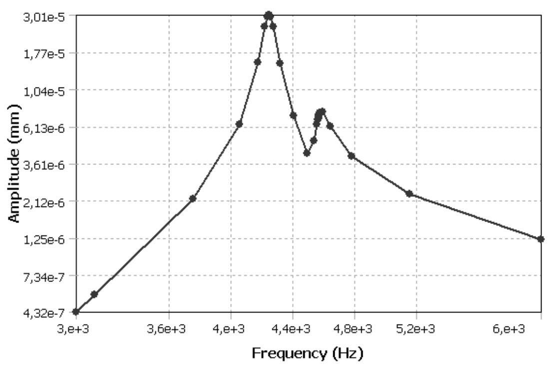
The harmonic analysis is a dynamic analysis method of the mechanical structures loaded with cyclic loads. The harmonic analysis algorithm consists of determining the harmonic answer – generally the vibrations' amplitude – depending on the frequency. Harmonic analysis is a linear analysis, and any nonlinearity (plasticity, contact) is eliminated [2].

The model used in the harmonic analysis is the same with the one used for the static analysis, the difference being in the loading fashion of the structures.

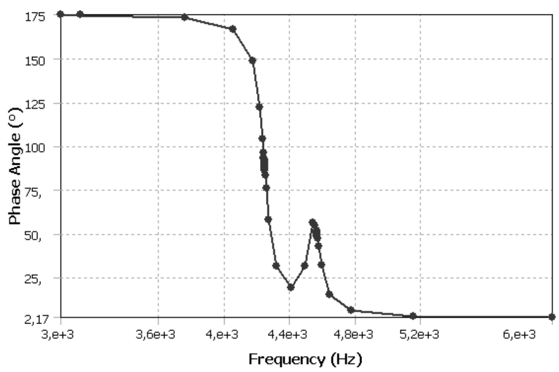
In this analysis the model was loaded through a torsion moment with amplitude of 50 Nm. The time variation of this moment is sinusoidal. The frequency range used in this analysis is 3000 – 5000 Hz, and these are frequencies that cover the first two excitation frequencies spectrum of the eccentric shaft.

For the existing model of the eccentric shaft the amplitude-frequency and phase-frequency characteristics obtained following the harmonic analysis are presented in Fig. 8.

From results of the modal and harmonic analysis, one can observe that the maximum value of the vibrations' amplitude is $3 \cdot 10^{-5}$ mm. Moreover, the frequency when these vibrations appear is 4241.9 Hz, which corresponds to approximately 254000 rot/min. The maximum rotation value for this eccentric shaft is 3000 rot/min, which leads to the conclusion that neither the static solicitations nor those due to the vibrations are the cause for the defects that arise in the eccentric shaft.



a.



b.

Fig. 8. Characteristics amplitude – frequency (a) and phase – frequency (b) obtained following the harmonic analysis.

2.3. Fatigue analysis of the eccentric shaft

It is estimated that 50-90% of the structural failure is due to fatigue, thus there is a need for quality fatigue design tools [1, 5]. The focus of the fatigue analysis in Ansys is to provide useful information to the design engineer, when fatigue failure may be a concern. Also, the fatigue results can be added before or after a stress solution has been performed.

Fatigue, by definition, is caused by changing the load on a component over time. Ansys can perform fatigue calculations for either constant amplitude loading or proportional non-constant amplitude loading. A scale factor can be applied to the base loading if desired.

The classic calculation uses the constant amplitude, proportional loading option. Loading is of constant amplitude because only one set from the finite element stress results along with a loading ratio are required to calculate the alternating and mean stresses. Common types of constant amplitude loading are fully reversed ($R = -1$) and zero-based ($R = 0$). The results of the fatigue analysis are quoted to the fatigue safety factor (Fig. 9).

As can be seen in Fig. 9, the fatigue safety factor is sub-unitary (0.94), which leads to the idea that the failures of this shaft are due to the material fatigue. Also, the number of life cycles that result from the analysis (6.6×10^5) lead to the same conclusion. There are two alternatives to alleviating this problem: changing the shaft material or constructively optimizing the shape of the shaft. Due to the fact that changing the material can be a significantly increase costs, we decided to make a constructive optimization for this eccentric shaft.

3. DESIGN OPTIMIZATION

Considering that the problems arise in theory as well as in practice in the area connecting the eccentric shaft and the flange, we adopted as a constructive solution the introduction of a 10 mm fillet radius in this area. Figure 10 shows the safety factor value for the fatigue simulation.

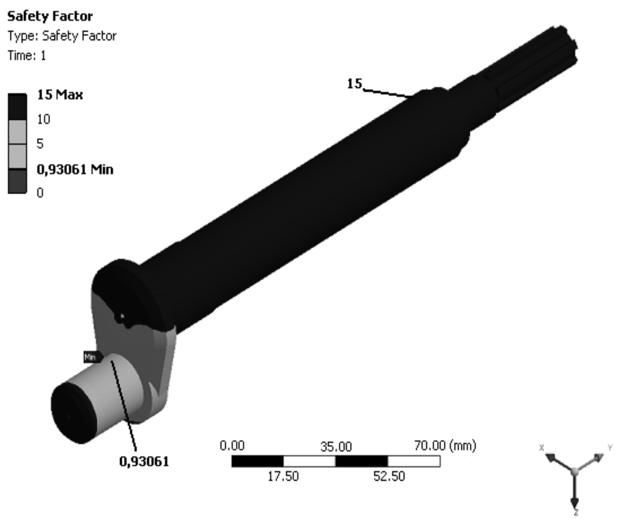


Fig. 9. Safety factor for the fatigue analysis.

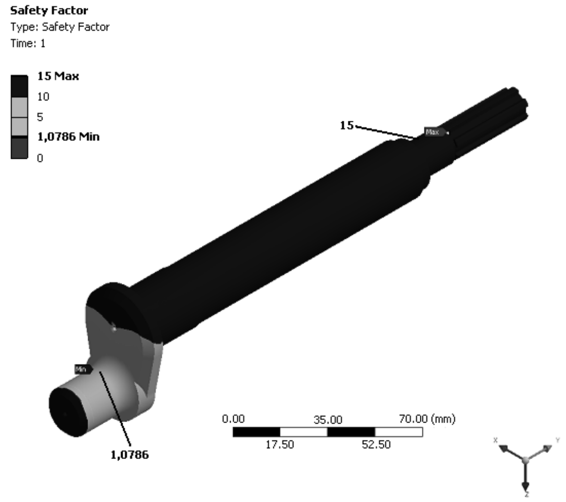


Fig. 10. Safety factor for the fatigue analysis in the optimized model.

4. CONCLUSIONS

Using the finite element method we were able to localize and correct the problems that appear on the eccentric shaft. Hence, for the new model of the eccentric shaft we obtained maximum values of the equivalent von Mises stresses of 79.92 MPa, that correspond to a safety factor of 3.13 for the static analysis and a safety factor of 1.08 for the fatigue analysis.

REFERENCES

[1] Talukdar P., Sen S. K., Ghosh S. K. (2001). *Effect of fatigue damage on the dynamic fracture toughness of En-8-grade steel*. Metallurgical and Materials Transactions, Vol. 32, 2001, pp. 2547-2552, ISSN 1073-5623.

[2] Moens, D., Vandepitte, D. (2006). *Recent advances in non-probabilistic approaches for non-deterministic dynamic finite element analysis*. Archives of Computational Methods in Engineering, Vol. 13, No. 3, 2006, pp. 389-464, ISSN 1134-3060.

[4] Oleksik, V., Pascu, A., Fratila, M. (2007). *Constructive optimization of the elastic element comprised in a mechanic dynamometer by finite element method*. Proceedings of the 24th Danubia-Adria Symposium on Developments in Experimental Mechanics, pp. 135-136, ISBN 978-973-739-456-9.

[5] *** Calculating and Displaying Fatigue Results (2006) Ansys Inc.

Authors:

PhD, Nicolae COFARU, Professor, "Lucian Blaga" University of Sibiu, Engineering Faculty, Department of Machines Building Technology,
E-mail: nicolae.cofaru@ulbsibiu.ro.

PhD, Adrian-Marius PASCU, Lecturer, "Lucian Blaga" University of Sibiu, Engineering Faculty, Department of Machine Design Fundamentals,
E-mail adrian.pascu@ulbsibiu.ro.

PhD, Valentin OLEKSIK, Associate Professor, "Lucian Blaga" University of Sibiu, Engineering Faculty, Department of Machines and Equipment,
E-mail valentin.oleksik@ulbsibiu.ro