International Conference KNOWLEDGE-BASED ORGANIZATION
Vol. XXX No 3 2024

MECHANICAL AND THERMAL ANALYSIS OF A ROCKER ARM SHAFT SUPPORT BASED ON FINITE ELEMENT METHOD

Ioan VIRCA, Tiberiu GIURGIU, Vasile CĂRUȚAȘU, Toma PLEȘANU

“Nicolae Bălcescu” Land Forces Academy, Sibiu, Romania
virca_ioan@yahoo.com, giurgiutiberiu@yahoo.com, v.carutasu@yahoo.com

Abstract: *The Finite Element Method (FEM) is a powerful numerical technique for solving complex physical problems involving differential equations and partial derivatives. This study presents the mechanical and thermal analysis of a rocker arm shaft support using FEM. Employing the ADINA (Automatic Dynamic Incremental Nonlinear Analysis) software, the behavior of the shaft support has been modelled, leveraging ADINA's extensive material libraries, diverse loading conditions, and customizable behavioral properties. The analysis focuses on determining deformations, stresses, and displacements, particularly in the median plane section at the point of maximum stress. Both treated and untreated material conditions are evaluated to assess stress transmission in the transition zone between surface-hardened and base materials. A 2D analysis is conducted, given the geometry of the rocker arm spindle support, to ensure fidelity to real structural behavior while optimizing computational efficiency. The validated model provides insights into design constraints for new materials, supporting industrial applications.*

Keywords: ADINA application, FEM, mesh, rocker arm shaft support

1. Introduction

The Finite Element Method (FEM) allows approximate numerical solving of problems related to complex physical phenomena that are described with differential relations or partial derivatives whose exact solutions are difficult to find [1], [2], [3]. The FEM gained popularity due to its wide degree of generality and applicability in the most diverse fields and due to its capability to ensure a satisfactory convergence of solutions in most situations. Moreover, the spectacular development of computing technology contributed significantly to the growth of FEM–based software applications, which can nowadays solve problems of high complexity.

The finite element method is based on several hypotheses such as:

- the physical system is considered continuous globally.
- the environment is homogeneous locally.
- known functions apply to meshing elements.
- the mesh elements may have different properties.
- the mesh elements are connected only in nodes.
- forces are applied in nodes (concentrated forces).
- deformations and displacements depend on node displacements.

Based on these assumptions, the defined element has physical, geometric, and functional properties depending on the assigned material. Concerning the objectives of the analysis, the elements can receive behavioral properties when stresses and transformation functions are applied

DOI: 10.2478/kbo-2024-0100
© 2024 Ioan Virca et al. This work is licensed under the Creative Commons Attribution-NonCommercial-NoDerivatives 3.0 License.

approximating the variables of the studied system. It can be observed that, for a given physical body, the law of discretization may differ in different areas of its content. The finesse of the mesh is thus a parameter that needs to be established considering its direct impact on the computing time, being a large consumer of computational resources. The modeling of structures based on the finite element method covers practically the entire field of engineering problems, especially structural mechanics.

2. FEM Software Applications

A list and short capability description of the most used FEM-based software is presented in Figure 1, most of them being used for similar research as in [1], [2], [3], [4].

For this study, to model the behavior of the rocker arm shaft support the ADINA (Automatic Dynamic Incremental Nonlinear Analysis) software application was used. ADINA software is continuously evolving and improving, according to the requirements of the industrial field [5]. For this study version 8.1 has been used, which ensures a wide range of capabilities such as: access to several material libraries, various predefined possibilities of characterization of loading conditions (Euler – Lagrangian type), different ways of defining degrees of freedom, multiple possibilities of defining contacts, many ways of meshing and process control, various ways of controlling output data, 16 generic types of elements and types of behavior modeling equations (applicable to each type of material), the possibility to customize toughness and viscosity, defining initial conditions and interfaces. Consequently, it can be said that the application covers the entire spectrum of engineering applications [6], [7]. The main advantage of ADINA software [8] and the reason why it was chosen for this research is that the three main phases of FEM-based software – preprocessing, analysis, and postprocessing can be run independently, allowing considerable time savings when running the simulations.

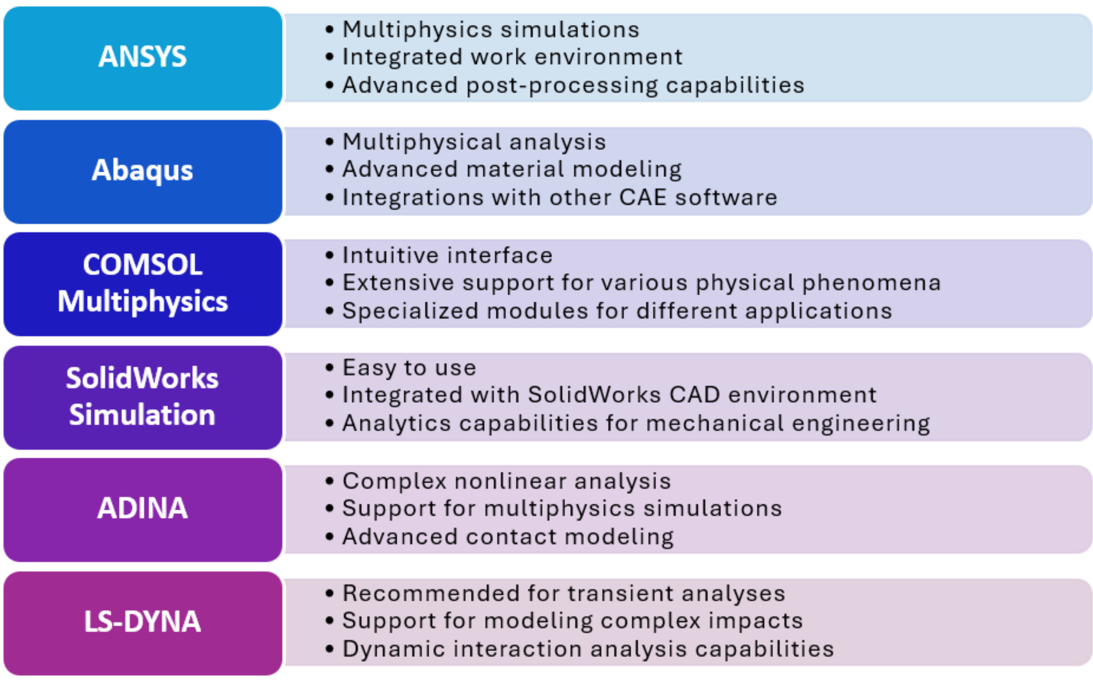

Figure 1: A list of the most known and used FEM-based software

3. Model Description

For the modeling and simulating processes, the following objectives were pursued:

- to determine the deformations.
- to determine the tensions, efforts, and pressures in a median plane section positioned at the point of maximum stresses.

- to evaluate the material displacements in a direction parallel to the geometric axis of the shaft (highlighting the tendency to expel material from the contact surface area).
- to analyze all these requirements for the rocker arm shaft support both treated (superficially hardened) and untreated.

When validating the model, the way of transmitting the stresses in the transition zone from the superficial hardened material to the base material was evaluated. Once the model is validated, from the analysis of stress values of the behavior of the part, it will be possible to determine elements of particularity that constitute restrictions for the design of new materials and under conditions to be evaluated for technical manufacturers.

For this study, given the geometry of the rocker arm spindle support, a three-dimensional analysis of the phenomena occurring in the contact surface area (or in the proximity), is not necessarily mandatory, a 2D analysis being sufficient.

The model obtained by discretizing the structure with finite elements and establishing the behavior of the material must meet the following requirements:

- to represent with sufficient fidelity the real behavior of the structure.
- to allow easy achievement of results expressed as values of tensions and displacements in the relevant sections.
- not to require excessive time for preprocessing and postprocessing.
- to efficiently use the available computing capabilities.

Figure 2: The analyzed rocker arm shaft support

To create the model of the rocker arm shaft support, the real constructive solution, shown in Figure 2, has been used. The dimensions of the rocker arm shaft support are presented in Table 1. For the rocker arm shaft, the diameter set was 19 mm, the same as that of the mounting bearing. This part is considered rigid, the value of the acting force being 25 daN.

Table 1 Dimensions of the rocker arm shaft support

Parameter	**Value**	**Measuring unit**
Thickness of the support	20,50	mm
Large hole diameter	19,00	mm
Small hole diameter	12,30	mm
Width of the support (at the top of the large hole)	7,70	mm

4. Calculation Algorithm Description

The algorithm of structural modeling and finite element analysis (FEA) involves several mandatory steps in the preprocessing phase, mentioned in Figure 3 [8].

The first two steps consist of defining the structure, its geometric and elastic characteristics, applied loads and boundary conditions.

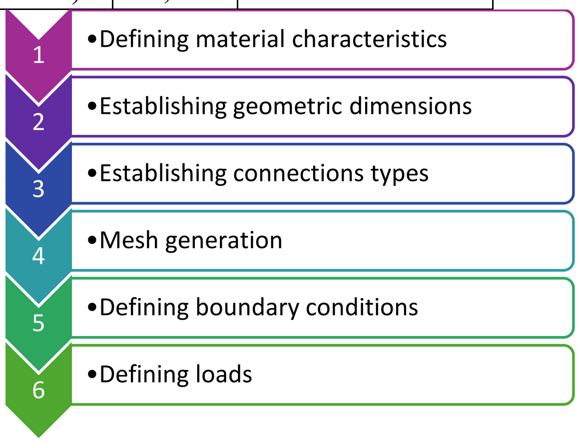

Figure 3: The preprocessing steps

Then, it must be considered the schematization of how the structure behaves and how it takes over the applied loads and the choice of finite element types for modeling the structure, with a direct impact on the modes of deformation and stresses relative to the established load. By completing this step, the material is considered concentrated in the geometric center of the structural element used, preserving the rigidity characteristics of the structure. The discretization (Figure 4) of the structure in finite elements with a sufficiently high degree of finesse is also a key step in terms of efficient use of available computing resources and desired precision.

After checking the correctness of all the input data, the calculation can be performed and finally results will be displayed.

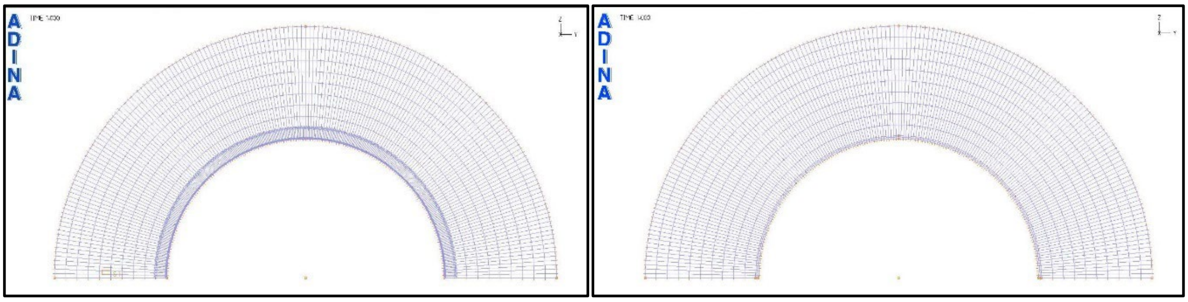

Figure 4: The discretization (mesh) of the model (untreated – left, hardened - right)

5. Results

5.1. Movements on the OZ Axis

Examining Figure 5 it can be noticed that the movements have values around 10^{-16} mm, considered small elastic deformations. If for the control part one can identify a deformation band which may suggest a plastic deformation tendency followed by breaking, when increasing the value of the stress, for the hardened part it is no longer present. The cause is represented by the different physical-mechanical behavior of the material in the two areas.

5.2. Movements on the OX Axis

To analyze the tendency of material expulsion in the OX direction, (perpendicular to the plane of Figure 4), it was necessary to present the deformations along this axis (Figure 6). For the untreated part, their value is around 10^{-10} mm and for the hardened area is around 10^{-15} mm.

These movements give information on the tendency of material removal due to the crashing that occurs between the two contact surfaces. The differences between values are given by the greater rigidity of the material in the treated area.

5.3. Maximum Effort Analysis

The analysis of the maximum efforts presented in Figure 7 enables the drawing of the conclusion that the presence of the thermally treated area leads to a more uniform distribution of the efforts throughout the whole mass of the part, due to its material characteristics.

If in the case of the untreated part, it can be noticed areas of concentration of the efforts at the level of the contact area, in the case of the treated part, these are absent.

5.4. Vectorized Stress Analysis

The analysis of the load vectors in the two analyzed cases (Figure 8) allows the identification of a better distribution of them and it can be appreciated that the presence of the treated area, through the material characteristics, limits the propagation of stresses in the body of the rocker arm shaft support.

5.5. Smooth Strain Analysis

Figure 9 highlights the importance of the treated area in transmitting stresses from the contact surface to the body of the rocker arm shaft support.

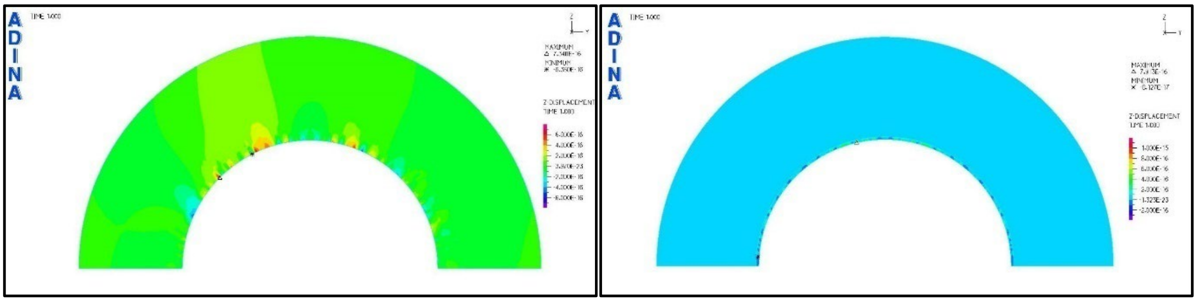

Figure 5: The movements on the OZ axis (untreated – left, hardened - right)

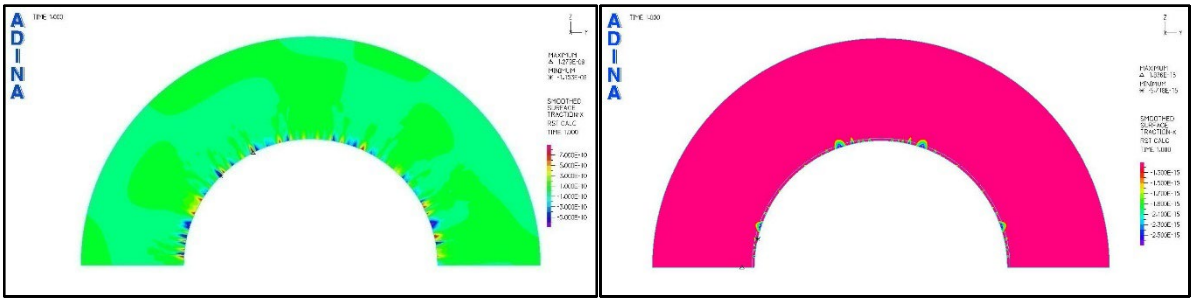

Figure 6: The movements on the OX axis (untreated – left, hardened - right)

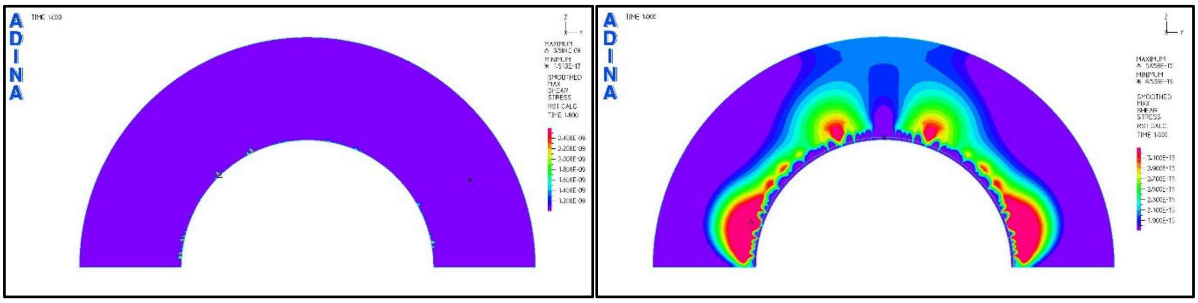

Figure 7: The Maximum Effort Analysis (Untreated – Left, Hardened - Right)

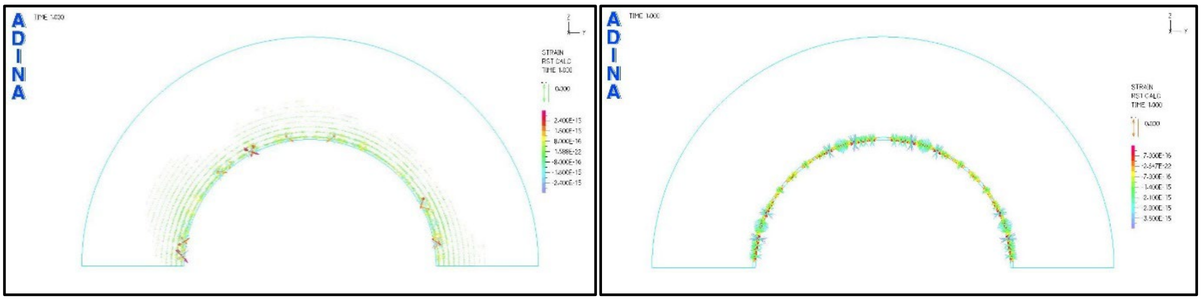

Figure 8: The vectorized stress analysis (untreated – left, hardened - right)

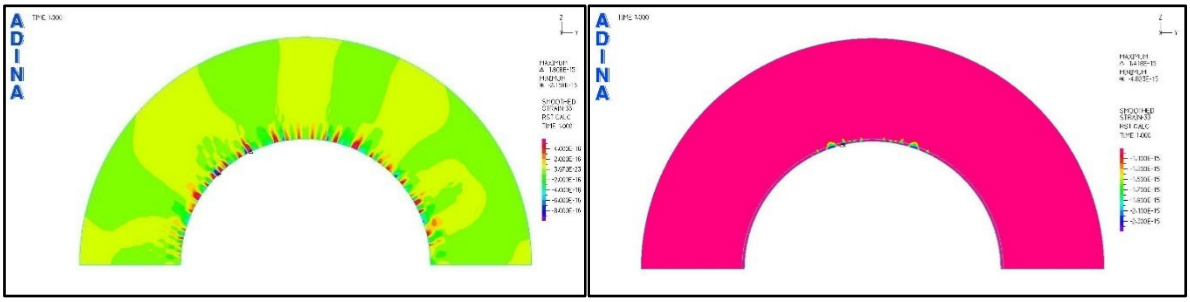

Figure 9: The smoothed strain analysis (untreated – left, hardened - right)

6. Discussion and Conclusions

Referring to the untreated rocker arm shaft support, the analysis of the deformations emphasizes that the elastic deformation of the part is small for real stresses, a fact that suggests good fatigue resistance. Although the stresses are of small value, the efforts are present throughout the analyzed surface (YOZ plane), which suggests the need for a very good homogeneity of the structure to avoid tension concentrators. We can also notice the tendencies of material removal from the contact surface area (OX axis in the figures presented), which leads to higher stresses, and plastic deformations with significant alteration of the geometry of the part and consequently alteration of functional parameters.

Referring to the hardened rocker arm shaft support, the analysis of effort and displacement values highlights the following:

- the presence of the treated area significantly limits the propagation of efforts in the body of the rocker arm shaft support.
- efforts are more evenly distributed in the section of the rocker arm shaft support.
- the tendency to move material away from the contact surface is significantly limited,
- an increase in the rigidity of the assembly.
- stresses that can lead to decreased resistance to fatigue are diminished.

Given the obtained results, practical conclusions can be drawn, such as:

- it has been demonstrated that the thickness of the hardened layer by the method used is satisfactory for low-load-bearing parts.
- the comparative study of total efforts opens the perspective of further developed analyses, referring to loads at certain points or on certain sections, related to the phenomenon that produced them.

Modeling and simulation of bearing behavior is a complex field for which, analyses such as the one presented above can provide, for complex specific solutions, elements of detailed knowledge of values and mode of variation of stresses, and elements regarding resistance and breaking mode. Setting the contact type as surface to surface allows the model to be used for high demands as well, with three-dimensional analysis, including shaft bending moment.

The simulation can be extended to all types of bearing parts, considering that the model is satisfactory in terms of accuracy and simplicity, and can also be used for variable stresses. For the untreated piece, it has been shown that the transmission of stresses throughout the body of the piece is present and may be limited by the presence of the heat-treated area. This highlights the need for surface hardening and demonstrates that the method used in this study is satisfactory concerning the physical, mechanical and dimensional characteristics of the treated layer.

References List

[1] Mawangi P.A.N., et al, (2019), Analysis of Rocker Arm Failure on Diesel Engines Using Finite Element Method, IOP Conference Series: *Materials Science and Engineering*, IOP Publishing,.

[2] Mushiri T., Mbohwa C., (2015), *Finite element analysis of a car rocker arm*, Proceedings of the 2015 International Conference on Operations Excellence and Service Engineering, Orlando, Florida, USA.

[3] Aswara A.Y., et al., Failure Analysis with Material Variations in Rocker Arm using Finite Element Method. IOP Conference Series: *Materials Science and Engineering*, IOP Publishing, 2019.

[4] Fatu A., Arghir M., (2018), *Numerical analysis of the impact of manufacturing errors on the structural stiffness of foil bearings*, Journal of Engineering for Gas Turbines and Power, 140 (4).
[5] *ADINA Theory and Modeling Guide Volume I: ADINA Solids & Structures*, 2005.
[6] Jakubovičová L., Sága M., (2014), Computational analysis of contact stress distribution in the case of mutual slewing of roller bearing rings, *Applied Mechanics and Materials*, (474), pp. 363-368. https://doi.org/10.4028/www.scientific.net/AMM.474.363.
[7] Kania L., Śpiewak S., (2014), Influence of radial forces on change of friction torque in twin slewing bearing, *Machine Dynamics Research*, 38 (2), pp.121-131.
[8] Lee D.W., Cho S.S., Joo W.S., (2008), An estimation of failure stress condition in rocker arm shaft through FEA and microscopic fractography, *Journal of mechanical science and technology*, Vol. 22, pp. 2056-2061.