THE ANNALS OF "DUNAREA DE JOS" UNIVERSITY OF GALATI
FASCICLE XIV MECHANICHAL ENGINEERING, ISSN 1224-5615
2023

STRUCTURAL ANALYSIS OF AN INERTIAL VIBRATOR SHAFT VIA 3D MODELING

Ștefan FLOREA
Dragoș BĂZĂREA
Prof. ec. dr. ing. habil. Adrian GOANȚĂ
Universitatea "Dunărea de Jos" din Galați

ABSTRACT

The paper presents the 3D modeling and structural analysis of an inertial vibrator shaft using Solid Edge and Abaqus. It investigates stress distribution, resistance, and fatigue behavior under operational loads. The simulation results guide geometric optimization for improved performance and durability. Energy consumption in resonance and post-resonance is also analyzed using MATLAB for assessing dynamic behavior.

KEYWORDS: inertial vibration, 3D modeling, finite element analysis,structural simulation

1. INTRODUCTION

3D modeling of mechanical devices plays a key role in the engineering design and analysis process, allowing for performance optimization and simulation of the dynamic behavior of systems. In the case of inertial vibrators, 3D modeling facilitates the study of the interaction between components, the analysis of mass distribution, and the determination of critical parameters that influence their operation.

An inertial vibrator (Figure 1) is a mechanical device used to generate and control vibrations by means of a rotating eccentric mass. It is widely used in areas such as vibrating conveyors, compaction equipment, structural testing, and industrial processes where vibrations are essential for efficient operation.

Figure 1. Inertial vibrator

A shaft in an inertial vibrator is essential for optimal operation, as it supports and distributes the loads encountered during the operational process. Its modeling and analysis become important for understanding the dynamic behavior of the system, as well as for optimizing its design and performance. Also, a correct modeling can prevent structural defects and unwanted vibrations, which can lead to damage and high maintenance costs.

3D modeling of the shaft involves the use of CAD (Computer-Aided Design) software to create an accurate digital representation of its components[1]. In this work, the modeling will be performed using Solid Edge 3D software, a powerful tool for the design, analysis, and optimization of mechanical components[5].

This stage is crucial for the structural analysis and simulation of the device's operation under different operating conditions. Techniques such as the Finite Element Method (FEM) allow engineers to study stress distribution, vibrational behavior, and eccentric mass optimization to achieve an optimal balance between efficiency and durability.

This paper aims to present the steps of 3D modeling of the shaft of an inertial vibrator, highlighting the technical aspects related to geometric design and structural analysis.

2. OPERATING PRINCIPLES OF INERTIAL VIBRATORS

Inertial vibrators are electromechanical devices designed to generate controlled vibrations by transforming the rotational movement of an eccentric mass into centrifugal forces. These forces result from the asymmetric distribution of the mass relative to the axis of rotation. They are transmitted to the supporting structure, serving to induce a vibration regime with well-defined parameters.

The use of inertial vibrators is essential in numerous industrial applications, where vibrations are used for transport, compaction, sorting, or structural testing.

From a constructive and functional point of view, these devices can vary significantly. They can be configured with one or two eccentric masses, having radial or axial positioning in relation to the rotation shaft. The drive mechanisms can be electric, pneumatic, or hydraulic, depending on the requirements of the application, and the vibration frequency can be adjusted by changing the rotation speed, allowing the system to adapt to different types of dynamic loads.

The performance of an inertial vibrator is determined by a number of essential factors, such as the mass of the eccentric, the shaft speed, the geometry and distribution of the masses in the assembly, as well as the structural rigidity of the supporting elements. A larger eccentric mass generates a stronger centrifugal force, while the rotational speed influences the working frequency of the system. The dynamic balance of the assembly is critical, since an uneven distribution of mass can produce parasitic vibrations or additional stresses on the bearings. In addition, the quality of the bearings, the type of mounting, and the general rigidity of the structure influence the efficiency of vibration transmission and the durability of the equipment. Due to these characteristics, inertial vibrators are found in a wide range of industrial applications. In the technological transport sector, they are used to move granular or powdery materials on vibrating belts or chutes. In the construction sector, they are an integral part of compaction equipment, such as vibrating plates and rollers, which ensure the optimal density of filling materials. They are also used in structural test benches to simulate dynamic loads in civil construction or in the aerospace industry. Other applications include sorting and screening processes, as well as vibrofinishing technologies, used to improve the surface quality of machined parts.

Therefore, inertial vibrators are distinguished by their versatility, energy efficiency, and the ability to provide adjustable vibrations adapted to the specific conditions of each industrial application. These characteristics justify the need to study in depth their operating principles, in order to optimize their design and integration into complex systems.

3. 3D MODELING METHODOLOGY

The three-dimensional modeling of the shaft of an inertial vibrator is a fundamental step in the engineering analysis process, as it allows obtaining an accurate digital representation, necessary for simulating the mechanical behavior under working conditions. In this work, the 3D model of the shaft was exclusively created (Figure 2) using the Solid Edge 3D software, with the structural analysis subsequently being performed in Abaqus, a high-performance tool for simulations using the finite element method (FEM).

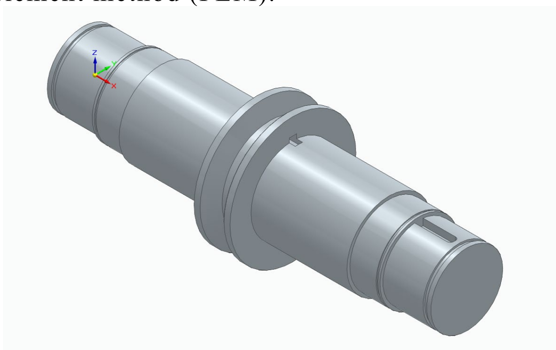

Figure 2. Inertial vibrator shaft

Solid Edge 3D was used to create the geometric model due to its parametric design capabilities and precision in representing construction details. The modeling was initiated by defining two-dimensional sketches of the relevant cross-sections, followed by extrusion operations and applying the dimensional characteristics specific to the shaft. The geometric parameters considered included the total length, the diameters of the support areas, the locations for mounting the eccentric mass, as well as any functional bores or grooves. Particular attention was paid to respecting the real constructive proportions, in order to ensure a solid basis for subsequent simulations. The model was also exported in an Abaqus compatible format (.STEP), thus facilitating integration into the structural analysis environment. At this stage, the entire vibratory assembly was not modeled, but only the critical component – the shaft – given its essential role in transmitting dynamic loads and its susceptibility to complex stresses in the vibration regime.

This approach aims to obtain an analysis model that allows the evaluation of the stress distribution, the natural vibration mode, and the behavior under loads, in order to identify possible structural weaknesses and further optimize the design.

4. STRUCTURAL ANALYSIS AND NUMERICAL SIMULATION

The model was imported into the Abaqus analysis environment, where a tailored finite element mesh was applied, with additional refinement in the areas of interest — especially in the grooved regions — to accurately capture the stress concentrations. The material considered for the shaft is 34CrNiMo6 alloy steel, chosen for its superior mechanical properties, with a high yield strength and good fatigue resistance. The operating conditions imposed in the simulation included a rotation speed of 2000 RPM, corresponding to the normal operating regime of the inertial vibrator. The applied loads included both the effects of centrifugal forces generated by the eccentric masses and the stresses arising from contact with the bearings. The results obtained in Abaqus were interpreted in order to evaluate the tension behavior, structural strength, and possible fatigue risks, and the conclusions are presented below.

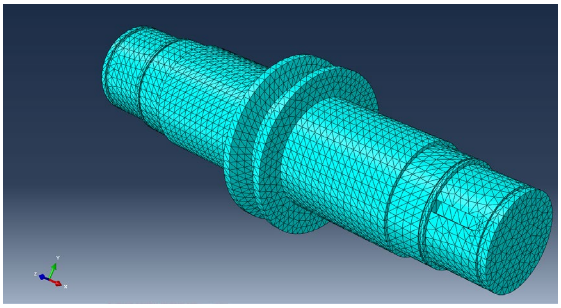

Figure 3. Mesh arbore

I started the analysis and obtained the following results:

4.1 STRESS ANALYSIS

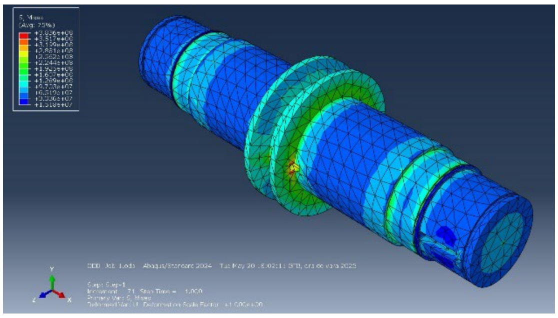

Figure 4. Finite Element Analysis

The finite element analysis (Figure 4) highlighted how the complex geometry of the shaft influences the stress distribution in its structure. With keyways at both ends and a central groove for mounting the drive pulley, the shaft presents a series of geometric discontinuities that act as stress concentration points. These effects are amplified by the presence of four safety grooves, two at each end, positioned in the areas where the bearings and eccentric masses are mounted. The simulation showed that these regions are the most stressed from a stress point of view, being subjected to a combination of bending, torsion, and radial compression[4] stresses generated by the contact with the bearings and the dynamic forces of the eccentric masses. The areas with the highest equivalent stress values are especially around the central keyway, where the pulley is mounted, which suggests that this is a critical area from a strength point of view. Thus, the geometry of the shaft, although functionally necessary, determines a complex stress behavior, and in these areas either local optimization of the shape (connection radii, superior finishes) or rigorous quality control in the manufacturing process is recommended to reduce the risk of crack initiation. The general conclusion is that the shaft operates under normal conditions overall, but requires special attention in design and execution in the grooved regions, where local stresses reach the highest values.

4.2 RESISTANCE CHECK

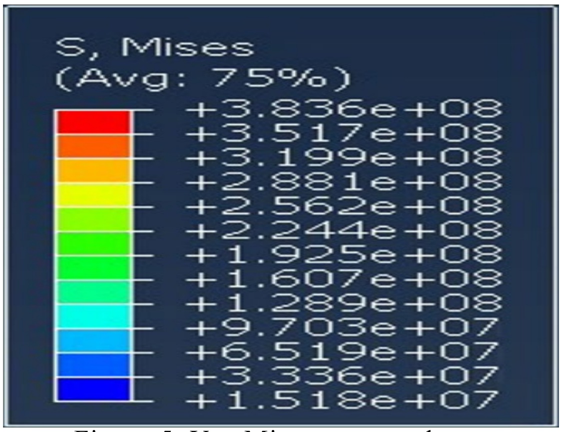

Figure 5. Von Mises stress values

Following the finite element analysis and the evaluation of the maximum values of the equivalent stresses (Von Mises) (figure 5), it was found that the shaft falls within the allowable strength limits for the material used, 34CrNiMo6. This alloy steel is known for its high resistance to mechanical stress, with a yield strength located, depending on the applied heat treatment, around 900–1100 MPa. In the present project, an appropriate safety factor was adopted, and the comparison between the maximum stresses obtained in the simulation and the allowable limit showed that the shaft does not exceed the critical threshold in any analyzed area. The most stressed areas, identified around the central keyway and near the safety grooves, present significant stresses, but which remain below the allowable limit of the material. This result confirms that the shaft is capable of withstanding the combined stresses (bending, torsion, radial stresses) imposed during normal operation of the inertial vibrator[2], without risk of yielding or breakage. Thus, from a static strength point of view, the designed shaft complies with the structural safety requirements, and the choice of the 34CrNiMo6 material has proven appropriate for this type of application.

4.3 FATIGUE BEHAVIOR

Given the cyclic operating regime of the shaft, fatigue behavior is an essential criterion in assessing its durability. Finite element analysis has allowed the identification of areas with high potential for fluctuating stress accumulation, especially around the central keyway, where the pulley is mounted, and in the areas of the safety grooves, located near the bearings and eccentric masses. These regions are subject to repeated stresses, which can favor the initiation and propagation of fatigue cracks. Even if the maximum stress values remain below the admissible limit of the material, under long-term operating conditions, cyclic variations can lead to structural failures if not treated appropriately[3]. The choice of the 34CrNiMo6 material is advantageous from this point of view, due to its good fatigue resistance, especially when used with appropriate heat treatments.

4.4 GEOMETRIC OPTIMIZATION

The FEA analysis provided a clear picture of how the shaft takes on the loads, which opens the possibility of optimizing its shape and mass, without compromising safety.

Based on the results, the following directions can be proposed:

• Diameter reduction in areas with low stresses to reduce mass and moment of inertia,

• Reduction of stress concentrations by introducing smoother transitions between segments of different cross-sections,

• Possible controlled axial perforation, if justified by mass reduction without significantly affecting rigidity.

The goal of optimization is to achieve an effective compromise between strength, weight, and cost, especially in applications where the dynamic balance of the vibrator is critical and the shaft mass significantly influences overall performance.

REFERENCES

[1] Goanţă A.M. – Grafică Asistată şi Modelare geometrică Am simulat mişcarea trizată - Aplicaţii, Editura Galati University Press, 2018, ISBN 978-606-696-139-4, 230 pag

[2] J.E. Shigley, C.R. Mischke, R.G. Budynas, *Mechanical Engineering Design*, 9th Edition, McGraw-Hill Education, 2010. ISBN: 978-0073529288

[3] S. Suresh, *Fatigue of Materials*, 2nd Edition, Cambridge University Press, 1998. ISBN: 978-0521578479

[4] W.C. Young, R.G. Budynas, *Roark's Formulas for Stress and Strain*, 8th Edition, McGraw-Hill Education, 2011. ISBN: 978-0071742474

[5] Siemens Digital Industries Software. Solid Edge 2024. https://www.plm.automation.siemens.com/global/ro/products/solid-edge/