VIRTUAL DEVELOPMENT OF AUTOMOTIVE WHEELS: MODAL, VERTICAL IMPACT AND FATIGUE ANALYSIS AND SIMULATION, USING THE FINITE ELEMENT METHOD

Vinícius Ramos Israel Santos
Federal University of Lavras, Mechanical Engineering Department, Campus, ZIP Code 37.200-900, Lavras, MG, Brazil

Francisco Scinocca
Federal University of Lavras, Mechanical Engineering Department, Campus, ZIP Code 37.200-900, Lavras, MG, Brazil
Corresponding author: francisco.scinocca@ufla.br

Luís Gustavo Nunes de Almeida
Federal University of Lavras, Mechanical Engineering Department, Campus, ZIP Code 37.200-900, Lavras, MG, Brazill

Fábio Lúcio Santos
Federal University of Lavras, Mechanical Engineering Department, Campus, ZIP Code 37.200-900, Lavras, MG, Brazil

Abstract: In recent years, research and development in the automotive industry have increasingly focused on safety, comfort, and energy efficiency. A crucial area of this research is the wheel/tire system, which is vital in ensuring vehicle safety and drivability. This study examines the effects of various types of loads applied to the wheel/tire system. Furthermore, it assesses the safety factor and potential fatigue-related failures in automotive wheels, identifying the areas with significant deformations and displacements. The research employs CAD and CAE software to analyze the behavior of automotive wheels, utilizing the Finite Element Method (FEM), which is commonly used during the automotive design phase to reduce costs associated with prototypes and validation. In the modal analysis, we examined the first four vibration modes, with the natural frequencies and modal shapes ranging from 350 to 610 Hz. For vertical impact analysis, we applied an internal tire pressure of 200 kPa (29 psi) along with a force of 9 kN to the tire, keeping the wheel fixed and the screws secured. This analysis revealed a safety factor of 2.60 for the most critical case. In the lateral fatigue analysis, a force of 1.5 kN was applied to the region around the screws using a 760 mm lever arm. This scenario resulted in a safety factor of 2.66. Lastly, for the radial fatigue analysis, we maintained an internal tire pressure of 200 kPa (29 psi) and applied a force of 7.5 kN to the screw region, simulating the tightening torque. The safety factor for this most critical case was determined to be 1.76.

Keywords: Automotive wheel; Finite element method; Simulation; Modal analysis; Vertical impact; Fatigue.

Received: April 22, 2024 - Accepted: January 04, 2025

INTRODUCTION

In recent years, the automotive industry has shown great interest in developing research in the dynamical behavior of the wheel/tire system (GILLESPIE, 1992),(REIMPELL; STOLL; BETZLER, 2001),(KABE, 2005),(SAVITSKI, 2015) ,(FARRONI; SAKHNEVYCH; TIMPONE, 2016),(D'AMBROSIO; VITOLO, 2018),(FARRONI; SAKHNEVYCH; TIMPONE, 2018). Research including the static and dynamic properties of the wheel/tire system becomes of fundamental importance in order to guarantee high performance to the vehicle, providing energy efficiency, comfort, and vehicle safety. With the increasing use of computational simulations in the automotive systems design, optimized parts of the design could be obtained. Several analyses, such as modal, impact, and fatigue, could be easily performed, improving the vehicle performance as a whole (SHARMA; PARASHAR, 2019). One of the techniques widely used in the development of automotive systems is the Modal Analysis. The Modal analysis is a technique used to determine the vibration characteristics and dynamic behaviors of a system, providing its natural frequencies, modal shapes, and damping factor, known as modal parameters (ZHANG, 2019), (MOHAMED; AZMIR, 2020). The analytical modal analysis uses computational tools, such as Finite Element software, used as an indicator of structural dynamic response and damping determination (RAMU; MOHANTY, 2014),(HASSAN, 2015), (SHARMA; PARASHAR, 2019). In this way, modal analysis presents several relevant points for the development of a component, such as the provision of a faster and more efficient procedure, making it possible to acquire data on the dynamic properties of a structure (MOHAMED; AZMIR, 2020). Fatigue failure is one of the main causes of mechanical component rupture (LI, 2014), (LIU, S. e. a., 2016). Most parts failures are due to the application of time-varying loads, occurring at stress levels significantly lower than the yielding limits of the materials, called fatigue failure. Thus, fatigue is defined as the progressive and localized structural damage that occurs when a material is subjected to cyclic loading. Thus, there are three stages of fatigue failure: crack initiation, crack propagation, and sudden fracture due to unstable crack growth. The first stage can be short-lived, the second stage involves most of the life of the part, and the third stage is instantaneous (LI, 2014). The process of damage to a component is the process in which the strength of the material decreases so that the residual strength of materials can be predicted based on fatigue damage (WEI, 2019). In this way, an important tool related to fatigue is the S-N curve of the material. This S–N curve describes the relationship between stress amplitude and the number of cycles. Thus, the simulation of components can be performed computationally through software based on the Finite Element Method. The numerical method, in which the simulation model and the fatigue parameters are combined to calculate the fatigue life, presents a lower cost and high efficiency. Therefore, the computer simulation replaces the need to manufacture prototypes to carry out tests (LIU, S. e. a., 2016). Particularly, FEM has become a powerful tool for static and dynamic analysis of a wide range of engineering structures and components. It is a computational approach that provides a very accurate approximate solution to engineering problems (SHARMA; PARASHAR, 2019), (MOHAMED; AZMIR, 2020). This analysis has three main stages: pre-processing, solution, and post-processing. In the first step, it is necessary to generate a CAD model of the wheel to be analyzed. Subsequently, in the second step, the mesh is generated, the properties of the constituent material of the analyzed model are assigned and the boundary conditions of this system. Finally, the last step consists of obtaining and analyzing the results, verifying whether they present the desirable physical behavior. Thus, Finite Element Modeling is widely used in industry in order to produce an excellent representation of an engineering structure (SHARMA; PARASHAR, 2019). Currently, there are several studies that include approaches performing modal and fatigue analysis of engineering structures through computer simulations by FEM in order to guarantee greater reliability to the system (YILDIZ; DUZGUN, 2010), (GU, 2017), (KUMAR; SINGH, 2018), (BAHARI, 2019). Thus, the present paper aims to analyze the effect of different types of loads applied to an automotive wheel, analyzing the safety factor and possible fatigue failures of this system. Thus, an approach through CAD (SolidWorks) and CAE (HyperMesh Altair OptiStruct) softwares was carried out to identify the behavior of the automotive wheel. In this way, we performed the approaches of the modal analysis, the vertical impact, the lateral fatigue, and the radial fatigue using the FEM technique. Therefore, it is possible to verify the safety factor and analyze the possible failures due to fatigue in the system. Moreover, it is possible to verify the most significant deformations and displacement areas of the automotive wheel.

MATERIALS AND METHODS

In the present research, the proposed methodology is summarized in Figure 1.

In order to analyze the effects of the wheel/tire system in relation to the load cases performed, the material that presented a high specific resistance is selected. Subsequently, the

https://doi.org/10.31422/taae.v9i1.63

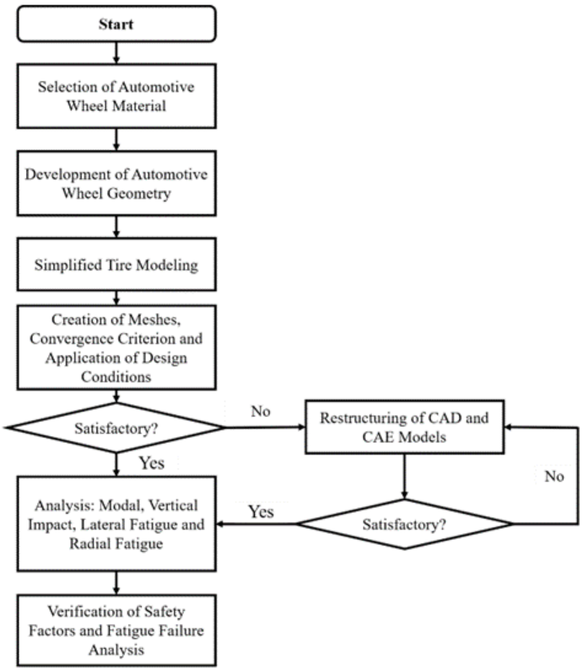


Figure 1: Flowchart of the Proposed Methodology

automotive wheel design is carried out using CAD software. After creating the meshes and applying the boundary conditions to the system, it was possible to carry out the modal analysis, including the vertical impact, the lateral fatigue, and the radial fatigue. Furthermore, it was possible to verify the safety factor, possible fatigue failures, and the greatest stress areas of the system. These satisfaction conditions seek to investigate whether the necessary requirements for virtual simulation analysis are met. If they do not meet the required needs, they must be adjusted to proceed to the subsequent steps.

Material selection and development of automotive wheel geometry

Three types of materials were selected for the manufacture of the automotive wheel, namely: SAE 1020, Aluminum Alloy EN AC 462000-T6 and Magnesium Alloy ZK60-T5, as shown in Table 1. Thus, a comparative analysis was carried out among the three different materials, considering several mechanical properties, especially the specific resistance, which relates the yield point and the specific mass of the analyzed material (MARINI; KEDZIORA, 2019), (ARGUS, 2022).

Through the analysis of Table 1, it is possible to identify that the Magnesium Alloy ZK60-T5 has the best (highest) specific resistance and the lowest specific mass among the three materials analyzed. These factors make this material interesting for the proposed model and simulations of the automotive wheel. In this way, the automotive wheel has good specific mechanical resistance, in addition to being a lightweight product. The mechanical characteristics of the selected material related to fatigue issues, analyzing the relationship between the number of cycles and stress amplitude through the S-N curve (stress amplitude versus number of cycles (log scale)), as illustrated in Figure 2. In addition, the designed automotive wheel will undergo a forging process, so the properties related to the strength of this alloy should be slightly better than the extruded ones (LIU, W. C. e. a., 2009).

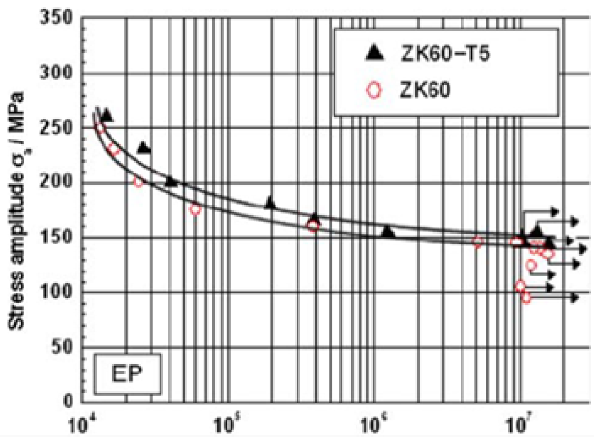


Figure 2: S-N Curves of Magnesium Alloys ZK60-T5 and ZK60 (LIU et al., 2009)

According to MARINI and KEDZIORA (2019), the ZK60-T5 magnesium alloy is an improvement of the ZK60 magnesium alloy, since using shot peening surface treatment in conjunction with T5 ageing, the fatigue strength was increased from 140 to 195 MPa. Therefore, the ZK60-T5 magnesium alloy becomes more attractive to be used in this model. Furthermore, magnesium must be handled with special care, mainly in powder form or shavings due to the risk of fire. This issue can be very important during the manufacturing process of parts, for example, by machining, where a large amount

Table 1: Mechanical Properties of SAE 1020, Aluminum Alloy EN AC 462000-T6 and Magnesium Alloy ZK60-T5 (Adapted of (MARINI; KEDZIORA, 2019))

Properties	SAE 1020	Aluminum EN AC 462000-T6	Magnesium ZK60-T5
Elasticity Modulus [MPa]	205000	71000	45000
Poisson's Coefficient	0.30	0.33	0.35
Specific Mass [kg/m^3]	7870	2800	1700
Yield Limit [MPa]	350	220	273
Specific Resistance $\left[\left(\frac{MPam^3}{kg}\right)\right]$	0.0445	0.0786	0.1606

of chip production occurs.

The ZK60-T5 magnesium alloy displays promising mechanical properties, such as high strength, good toughness, and outstanding corrosion resistance, which make it an attractive material for engineering applications. Studies have shown that factors such as microstructure, zinc content, heat treatment, and plastic deformation play a significant role in determining the alloy's properties. Additionally, the addition of rare earth elements, particularly lanthanum, can improve its corrosion resistance. However, the alloy has limitations, such as low creep resistance at high temperatures and low fatigue strength, which must be taken into account during its application. Further research is needed to optimize the alloy's properties and expand its range of applications. Key studies that explore the alloy's properties include those conducted by (ZHU; WANG; WU, 2019), (CHEN, 2020), (LIU, Q. e. a., 2021), (SUN, 2021).

To protect the surfaces of the automotive wheel from oxidation and corrosion processes, techniques are used, such as electrochemical treatment, which does not increase the component's mass; another protection process is painting, which is traditionally used on the wheels but causes an increase in the mass of the system (MARINI; KEDZIORA, 2019). The developed wheel is suitable for today's market scenario, considering several performance and safety issues of the vehicle system. These issues are related to the cooling of the brake system, geometry of the mechanical component with mass reduction, structure with symmetry planes, and absence of sharp corners. Therefore, the automotive wheel studied in the simulation has the following specifications: 17-7 rim, 5x100, offset 48 mm. Therefore, the automotive wheel was developed based on examples from the BBS Catalog (JAPAN, 2022). The model of the automotive wheel developed in SolidWorks software is illustrated in Figure 3.

Automotive tire modeling

The modeling of the automotive tire with specification 205/40 R17 was carried out, adopting hypotheses and simplifications in the simplified model created, maintaining the main properties and characteristics of the elastomeric compound. These issues were performed to ensure a simulation close to the actual state of the system. The hypotheses are related to the questions of simplification of the elastomer geometry (rectangular shape) and the obtaining of dimensional parameters of the bead stringing (structural reinforcement of the tire) and tread of the automotive tire. Therefore, the dimensional data of these parameters were obtained in loco in the laboratory of the Formula SAE Competition Team (Engineering Students Competition).

Thus, considering the specifications and modeling hypotheses related to the automotive tire, a CAD model of this component was developed. Subsequently, the developed model will undergo computer simulations using the Finite Element Method. In this way, for the modeling of wheel, tire, and structural reinforcement components, two types of contact were applied, called TIE and CONTACT. The TIE contact type has non-existent friction coefficients, as it behaves like a rigid element

https://doi.org/10.31422/taae.v9i1.63

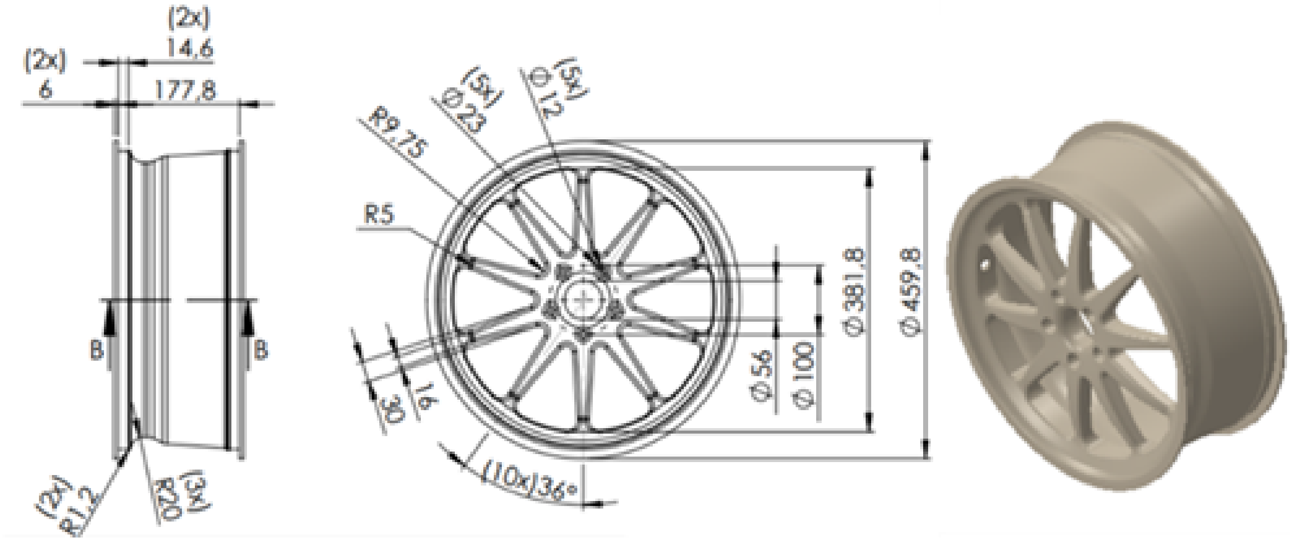
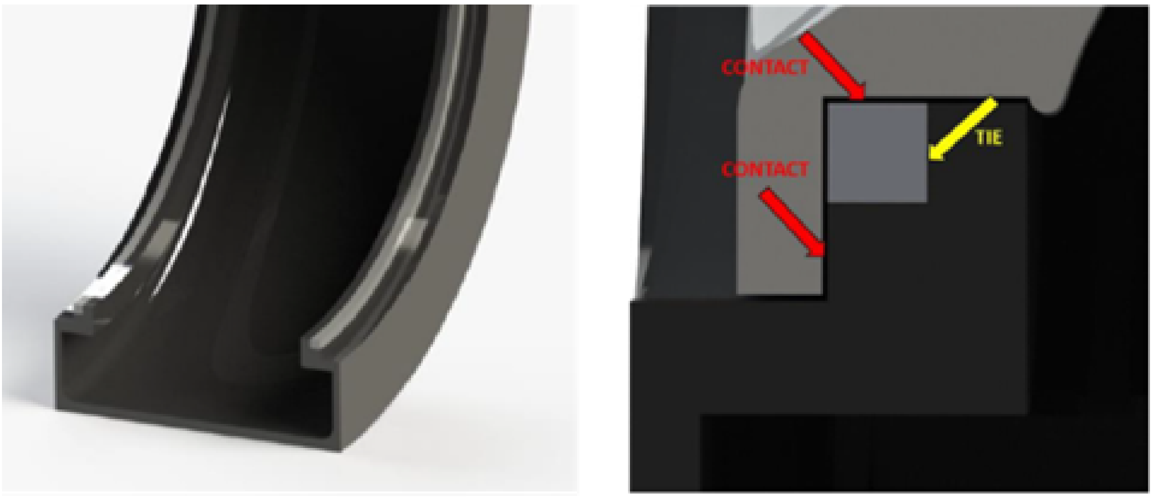


Figure 3: General view of the Automotive Wheel

between the analyzed surfaces; that is, the elements remain connected. The CONTACT type contact, in turn, has a different behavior, as there is displacement between the objects if the applied force is greater than the friction force. Therefore, the TIE type contact was applied between the tire components and the reinforcement, while the CONTACT type contact was between the wheel and tire components and the reinforcement and wheel components. The TIE contact type is employed to rigidly join two surfaces or components, assuming no sliding or separation occurs between them. This implies that the surfaces share the same degrees of freedom, effectively behaving as a single structural entity. Consequently, this type of contact is suitable for regions where components are permanently fixed or bonded, such as the structural reinforcement of an automotive wheel. On the other hand, the CONTACT contact type is used to model interactions where sliding, separation, or impact between surfaces may occur. This type of contact enables the simulation of dynamic and non-linear behaviors, such as friction and deformations. In the context of simulations, this contact is crucial for regions where the tire interacts with the rim of the automotive wheel. In general, the CONTACT contact type is associated with interfaces subject to dynamic variations and external forces, as exemplified by the rim of the automotive wheel in fatigue and impact analyses. Conversely, the TIE contact type is applied to regions with rigid and fixed connections, such as structural reinforcements or internal parts of the automotive wheel where no relative motion occurs between surfaces, as illustrated in Figure 4.

Meshes and design conditions

In this stage of analysis of the development of the automotive wheel, computer simulations were started using the FEM through the CAE software. Subsequently, tetrahedral elements from the Tetra mesh mesh tool were used to perform the system simulation. The utilization of the Tetra mesh element in HyperMesh presents a significant advantage for modeling complex-shaped automotive wheels. Its capacity to accurately represent intricate geometries, accommodating the complex curvatures and irregular surfaces of the wheel, renders it ideal for capturing the wheel's complex geometry. Moreover, the tetra mesh element allows for localized mesh refinement, enabling precise capture of crucial details such as stress concentrations and wheel-spoke junctions. With support for various structural analysis



a) b)

Figure 4: Representation of the tire and contacts (TIE and CONTACT)

methods, including finite element analysis, it empowers a comprehensive assessment of wheel performance, as modal analysis and fatigue analysis are objects of the present research. Thus, the tetra mesh element in HyperMesh stands as a valuable tool for accurately modeling and analyzing complex automotive wheel geometries. Tetra mesh elements are valid for all analyses, including all fatigue approaches of the analyzed structure. Regarding the design conditions (loads and constraint conditions), these were performed according to each type of case and thus applied to the system components in order to analyze the proposed issues. Therefore, regarding the loads used in the proposed cases, all types of force were applied through the rigid element RBE3. The RBE3 element is a type of connection element commonly used in finite element simulations. It functions as an interpolation element, transferring loads or displacements from a master node to multiple dependent nodes. This element is specifically designed to distribute loads or displacements without introducing artificial stiffness into the model. The RBE3 element is particularly advantageous in modal, impact, and fatigue analyses, where accurate force distribution is essential to replicate the realistic behavior of the structure. The constraints, in turn, were applied directly to the nodes present on the surface. Finally, a convergence analysis was performed in order to obtain the number of satisfactory elements for each analysis done in the present research. Figure 5 presents the schematic model employed, and more details are presented in each topic.

The mesh convergence criterion adopted in this study ensures result stabilization for natural frequency, static linear, and fatigue analyses of the automotive wheel. Convergence is achieved when further mesh refinement produces negligible changes in the computed results, indicating that the numerical solution has become independent of the mesh density.

ANALYSIS OF THE AUTOMOTIVE WHEEL BY FEM

Modal analysis

Initially, to enable the analysis of the modal shapes of the component, the mesh was created on the automotive wheel, based on curvature and proximity, aiming at a refined mesh. Subsequently, in the modal analysis, the system

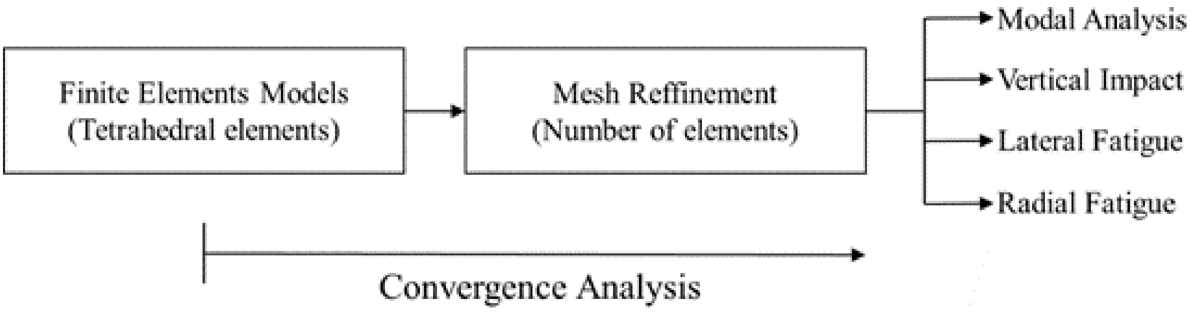


Figure 5: Convergence analysis – Schematic model

was modeled, and there was no application of loads or restrictions on the component to obtain the natural frequencies and the respective modal shapes of the system. In the present study, a mesh convergence analysis was performed for the automotive wheel, combining a reasonable computational cost and the necessary precision, obtaining convergence with 367389 elements and 99789 nodes for the simulation.

Vertical impact analysis

Vertical impact can be described as the application of force in a certain region of the tire with the wheel fixed. For the application of a force of 9 kN on the tire, an RBE3 element was created in a region of the same equivalent to the tire width × 60 mm (length over the tire), as shown in yellow in Figure 6. In addition, an internal pressure of 200 kPa (29 psi) was applied to the wheel and tire. Finally, the setting was performed on the screw-fixing surface. In this way, to carry out the analysis of the component in question, firstly, the symmetry planes of the same were verified, concluding that the model can be simplified in three cases lagged by 18^o. Figure 6 represents the three simulated models of the vertical impact analysis. Since the wheel has a circular pattern every 36^o, it is possible to analyze the other cases from these three proposed conditions.

As the application of the contact case is a quasi-static non-linear analysis, some convergence parameters are needed using the default values provided by the HyperMesh Altair OptiStruct software.

In the present study, a mesh convergence analysis was performed for each of the system elements, combining a reasonable computational cost and a necessary accuracy, obtaining convergence, as shown in Table 2.

Lateral fatigue analysis

The component was divided into two parts to perform the lateral fatigue analysis and enable the creation of the mesh in this wheel model. The primary structure is represented by the color blue, and the border, in yellow, as illustrated by Figure 7.

The case of lateral fatigue can be described as the application of a force by means of a lever arm, in which the rim of the wheel is crimped. A force of 1.5 kN is applied in the region of fixing the screws by means of a lever arm (RBE3) of 760 mm, represented by the green color (see Figure 8). Finally, the setting was carried out on the inner surface of the wheel rim, represented by the color pink, as illustrated in Figure 8.

Analogous to the case of the vertical impact analysis, the symmetry planes of the automotive wheel were verified, concluding that the model can be simplified in three cases lagged by 18^o, which are represented as shown in Figure 9.

In the present study, a mesh convergence analysis was performed for the automotive wheel, combining a reasonable computational cost and necessary accuracy, having 218140 elements and 62309 nodes to perform the simulation.

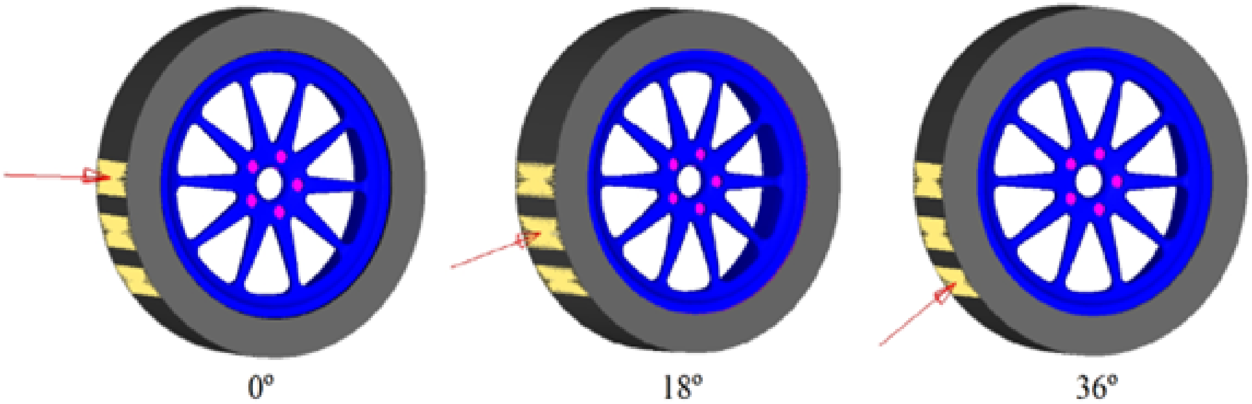


Figure 6: Design Conditions for **Vertical Impact Analysis**

Table 2: Mesh Convergence Parameters for **Vertical Impact Analysis**

Components	Elements Number	Nodes Number
Wheel	169827	50056
Tire	165349	53971
Tire Reinforcement	10015	4442

Figure 7: Division of the Automotive Wheel Elements

Radial fatigue analysis

The case of radial fatigue can be described as the application of a force in the screws fixing region, in which the tire is fixed in a certain area. For the application of a force of 7.5 kN, an RBE3 element was created on the screws fixing surface. In addition, an internal pressure of 200 kPa (29 psi) was applied to the wheel and tire. Finally, the setting was performed on the hub fixing surface, restricting only the displacement in the axial direction of the wheel. The tire surface was restricted referring to the tire width × 80 mm (length over the tire), as shown in yellow in Figure 10. Analogous to the previous cases, the analysis of the component was carried out, verifying its symmetry planes, concluding that the model can be simplified in three cases lagged by 18°. Analyzing Figure 10, it can be seen that the RBE3 element is represented by the pink lines, with the forces applied radially in the center of the wheel. In addition, it can be seen that the restrictions, in yellow, are applied to the center of the wheel, which are responsible for restricting the displacement in the axial direction of the center of the wheel. On the other hand, the restrictions applied to the tire, in yellow, represent the setting of the tire. The yellow color refer to the small triangles indicated around the support of the wheel, as shown in Figure 10.

Analogously to the lateral fatigue analysis, the application of the contact case is a quasi-static nonlinear analysis, thus, some convergence parameters are needed, using the default values provided by the software. In the present study, a mesh convergence analysis was performed for each of the system elements, combining a reasonable computational cost and a necessary

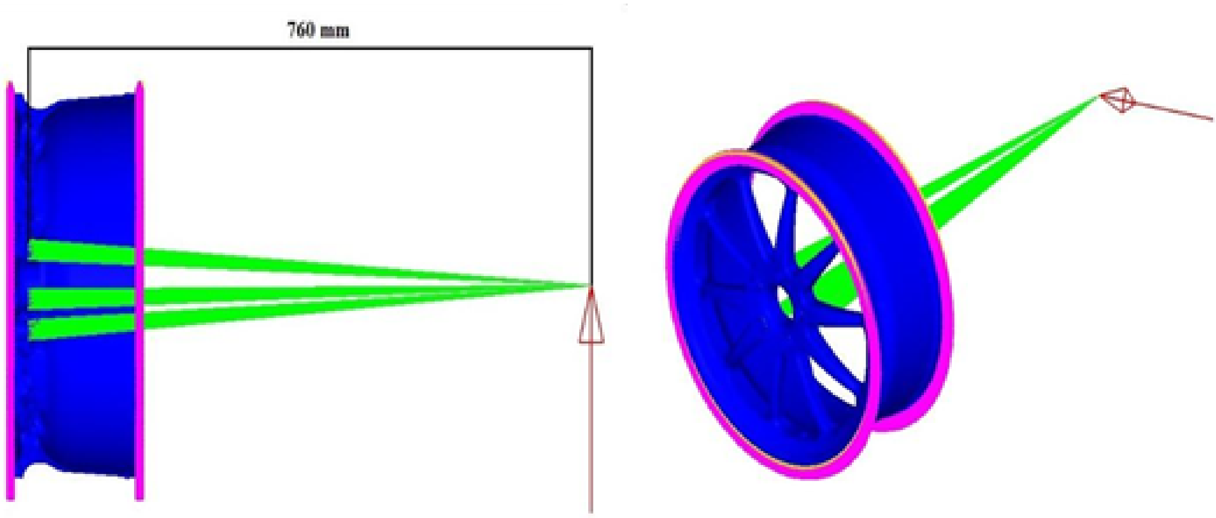


Figure 8: Loading Application for **Lateral Fatigue Analysis**

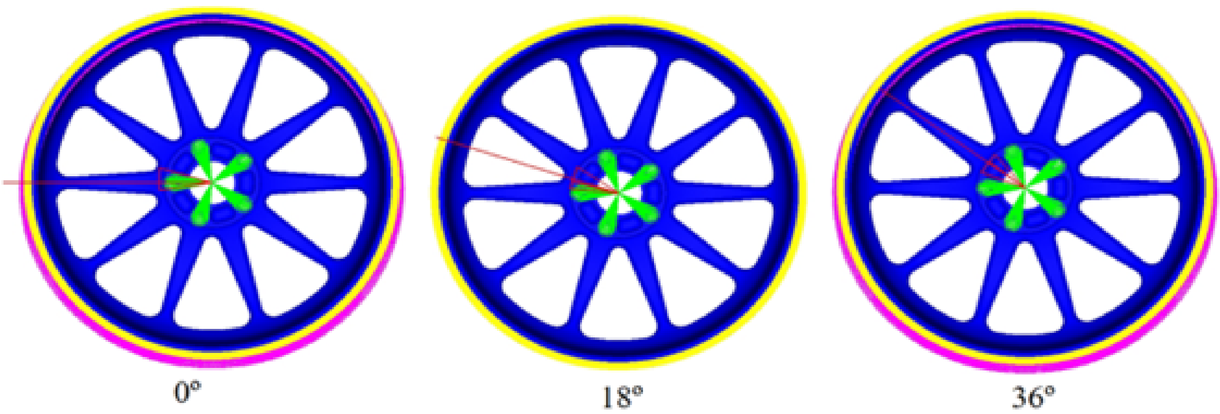


Figure 9: Design Conditions for **Lateral Fatigue Analysis**

accuracy, obtaining convergence, as shown in Table 3.

Some literature and catalogs were used to obtain data and parameters for carrying out this paper. In this way, Table 4 presents the declaration of availability of the data used in the research. The data used are of great importance for the development of this paper. They are directly linked to the material properties of the automotive wheel and fatigue failure analysis, such as S-N curve.

Safety factors and fatigue failure analysis

Finally, to analyze the issues related to the safety of the automotive wheel and the system as a whole approaches related to the safety factor were carried out (see Equations 1 to 3). In addition, it was possible to verify the main points and regions that suffer greater displacements and stresses, as well as possible system fatigue failures. Thus, Equations (1 to 3) govern the analysis of the safety behavior of the system.

$$SF = \frac{\sigma_F}{\sigma_{vM}} \quad \text{(Safety factor)} \tag{1}$$

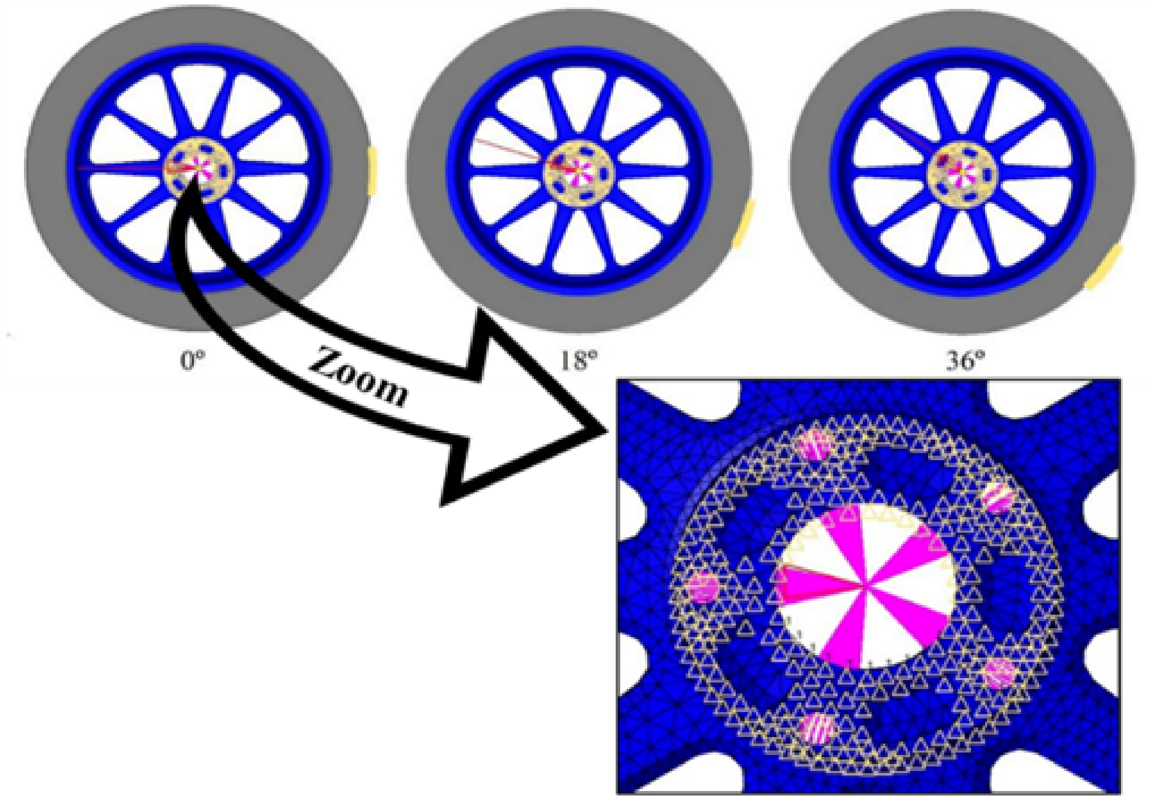


Figure 10: Design Conditions for **Radial Fatigue Analysis**

Table 3: Mesh Convergence Parameters for **Radial Fatigue Analysis**

Components	**Elements Number**	**Nodes Number**
Wheel	172632	49922
Tire	174193	56350
Tire Reinforcement	10017	4442

$$SF_F = \frac{\sigma_C}{\sigma_{MvM}} \quad \text{(Fatigue safety factor)} \tag{2}$$

$$C_C = \sigma_{SN}\, C_R\, C_S \quad \text{(Correct allowable stress)} \tag{3}$$

where SF: safety factor; σ_F: material yield limit; $\sigma_{\sigma_{vM}}$: von Mises stress; SF_F: fatigue safety factor; σ_C: corrected allowable stress; σ_{MvM}: maximum von Mises stress; σ_{SN}: stress obtained through the material's S-N curve relative to the number of cycles analyzed; C_R: correction factor related to system reliability; C_S: correction factor relative to the component surface. In the lateral and radial fatigue approaches, a reliability correction factor (C_R) equal to 0.814 was adopted for 99% component reliability. In addition, a surface correction factor (C_S) of 0.85 was used for material forging processes (NORTON, 2011). In these analyses, only C_R and C_S were considered, since the other design correction factors (σ_{SN}, σ_{vM}, σ_{MvM}) are already incorporated in the CAE software.

RESULTS AND DISCUSSIONS

The first analysis carried out was based on the modal analysis of the automotive wheel, considering its first four vibration modes. In addition, the six rigid body modes were excluded since no restrictions or loads were applied to the wheel. The first four vibration modes are shown in Table 5.

Table 4: DAS – Data Availability Statement

References	Data Availability Statement
(MARINI; KEDZIORA, 2019)	The data that support the findings of this study are available in the University of Luxembourg Library at http://hdl.handle.net/10993/28655, reference number 00114092016. These data were derived from the following resources available in the public domain: http://hdl.handle.net/10993/28655.
(LIU, W. C. e. a., 2009)	The data that support the findings of this study are available in High cycle fatigue behavior of as-extruded ZK60 magnesium alloy. J Mater Sci at https://doi.org/10.1007/s10853-009-3385-z, reference number 44, 2916–2924 (2009).
(JAPAN, 2022)	The data that support the findings of this study are available in BBS Japan Co., Ltd. at `https://bbs-japan.co.jp/en/wp_content/pdf/BBS_Catalog_2018_01_en.pdf`. These data were derived from the following resources available in the public domain:`https://bbs-japan.co.jp/en/wp-content/pdf/BBS_Catalog_2018_01_en.pdf`

Therefore, the system's vibration modes are called eigenvectors, which are directly associated with their respective frequencies, called eigenvalues. In modal analysis, the modes of the deformed body are analyzed. Since the analysis is performed in the free-free condition, the system presents six rigid body modes (three modes related to translation and three modes related to rotation), which are not related to the natural frequency of the structure. These modes refer exclusively to the displacement and rotation movement. Therefore, such modes should not be considered in the analysis. In this way, it is possible to observe a variation between a range of 350 Hz to 610 Hz, directly related to their respective modal shapes. Thus, modal analysis becomes a very important and effective tool, as it guarantees greater reliability and security of the system. This analysis avoids possible cases of resonance since the frequencies will be known through this approach. Because it is a symmetrical wheel, the first and second modes of vibration have a similar modal shape, only being distinguished in different planes of symmetry. The same can be seen for the third and fourth vibration modes.

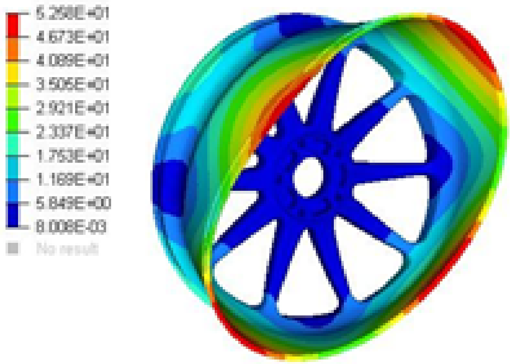
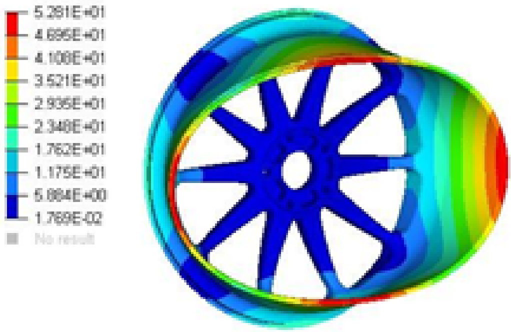
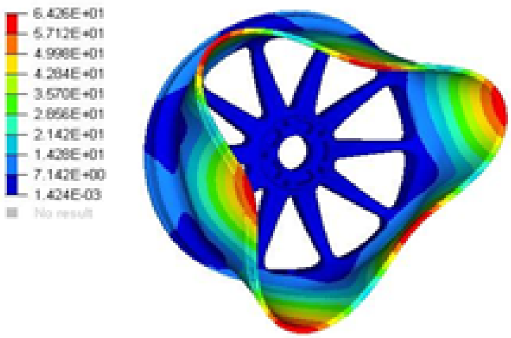
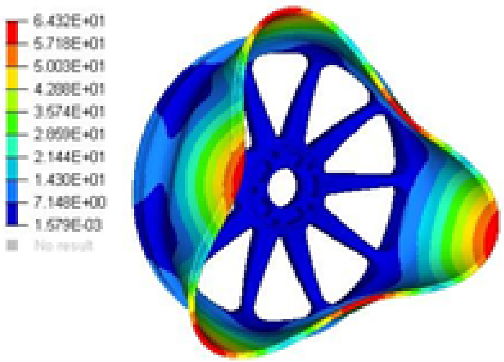
Vertical impact analysis

Subsequently, an analysis was performed regarding the vertical impact, considering three main cases lagged by 18^o due to the automotive wheel symmetry planes. Thus, Table 6 presents the values related to the yield point, von Mises stress, and system's safety factor for the three analyzed cases. Therefore, the safety factors were calculated using Equation 1, while the critical stress is illustrated by Figure 11.

Figure 11 illustrates the von Mises stress for the critical case of vertical impact analysis, with its maximum value equal to $1.051 \times 10^2 MPa$. Thus, the safety factor of the automotive wheel is equal to 2.60. Thus, in the case of vertical impact analysis, considering a reference value equal to 1.0, it is noted that the safety factor found for the critical case is 160% higher. Also, for automotive wheel optimization scenarios, this factor tends to be lower. However, the effects of uncertainties related to the manufacturing process, the material, and the nature of the impacts that the system suffers must be taken into account (CURSI; SAMPAIO, 2015), (SOIZE, 2017), (SCINOCCA; NABARRETE, 2020).

Analyzing Figure 12(a) and Table 6, it is possible to verify the value referring to the maximum von Mises stress equal to $9.366 \times$

Table 5: Automotive Wheel Modal Parameters

Modes	Frequencies [Hz]	Mode Shapes
First Mode	352.82	
Second Mode	353.34	
Third Mode	605.93	
Fourth Mode	606.21	

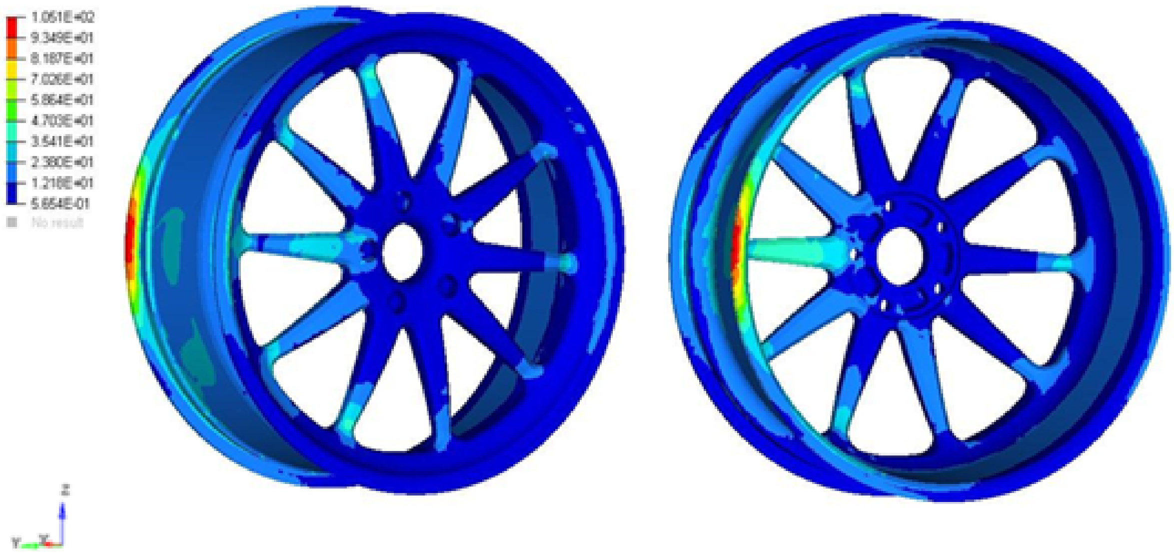


Figure 11: von Mises Stress for Vertical Impact Analysis – Critical Case (36^o)

Table 6: von Mises Stress and Safety Factors for Vertical Impact Analysis

Vertical Impact	0^o	18^o	36^o
Yield Limit [MPa]	273	273	273
von Mises Stress [MPa]	93.66	102.6	105.1
Safety Factor	2.91	2.66	2.60

$10^1 MPa$, with this critical value presented on the left rear edge of the automotive wheel rim. Therefore, the factor of safety is equivalent to 2.91. On the other hand, analyzing Figure 12(b) together with Table 6, it is possible to observe the maximum von Mises stress equal to $1.026 \times 10^2 MPa$, and a safety factor equal to 2.66.

Therefore, the other cases of this approach were analyzed, obtaining the maximum von Mises stresses. In this way, it is possible to identify the points of greatest stress and deformation that the automotive wheel is subject to, as illustrated in Figure 12.

Lateral fatigue analysis

In the case of lateral fatigue analysis of the automotive wheel, the system is subjected to 2×10^4 cycles. Thus, the stress obtained through the S-N curve refers to its respective number of cycles (see Figure 2), having a value equal to 230 MPa. Therefore, through Equation 3, the value of this corrected stress was obtained. Therefore, Table 7 presents the values of the maximum von Mises stress, the corrected stress of the S-N curve and the fatigue safety factor of the system for the respective analyzed cases. This approach is analogous to the vertical impact case. Thus, the critical case for the analysis of lateral fatigue is illustrated in Figure 13, and it is possible to verify the value referring to the maximum von Mises stress. The values obtained for S-N curve stress are associated with 2×10^4 cycles (see Figure 2). The values obtained for corrected allowable stress are associated with Equation 3.

In this way, Figure 13 illustrates the von Mises stress for the two critical cases of the lateral fatigue analysis of the system, with its maximum value equal to 5.987×10^1 MPa. Thus, the safety factor of the automotive wheel is equal to 2.66. In the case of lateral fatigue analysis, it can be verified that the safety factor found for the critical case is 166% higher compared to a standard value equal to 1.0. However, it is necessary to consider the effects of uncertainties related to the

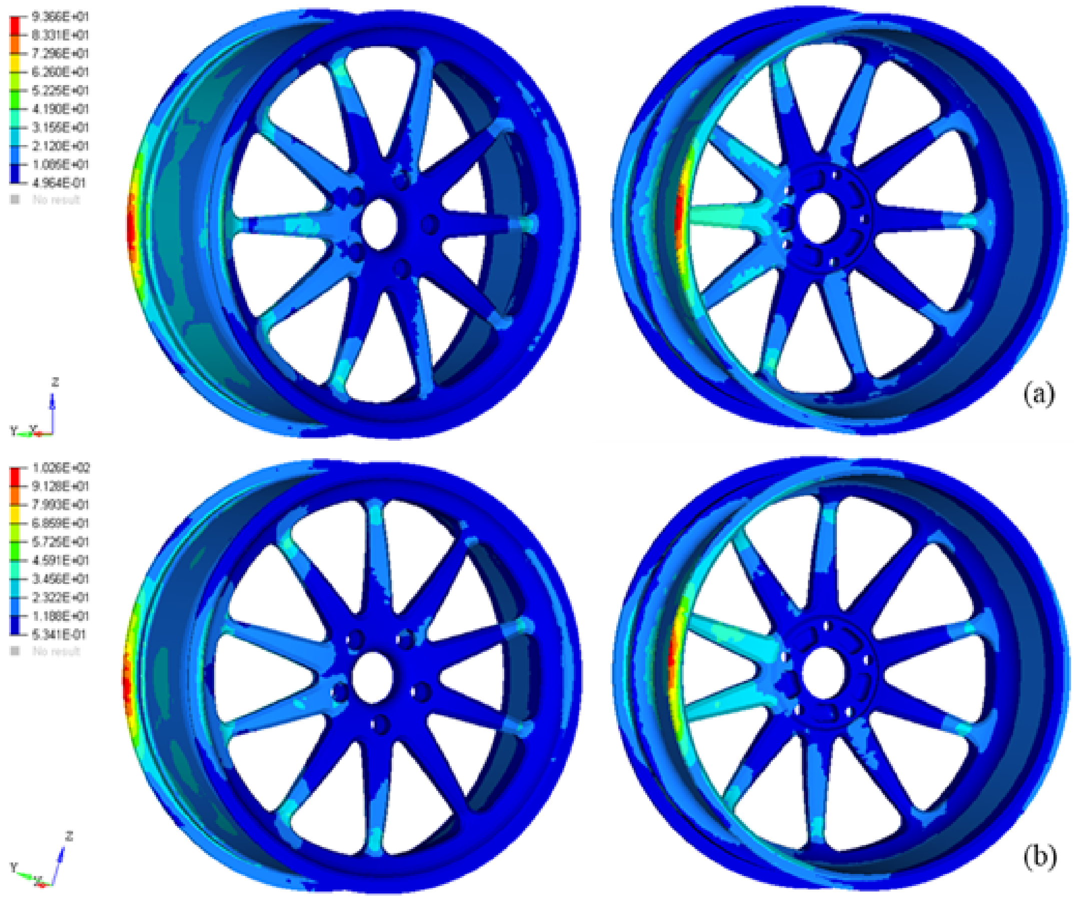


Figure 12: von Mises Stress for **Vertical Impact Analysis: (a) Case** 0^o**; (b) Case** 18^o

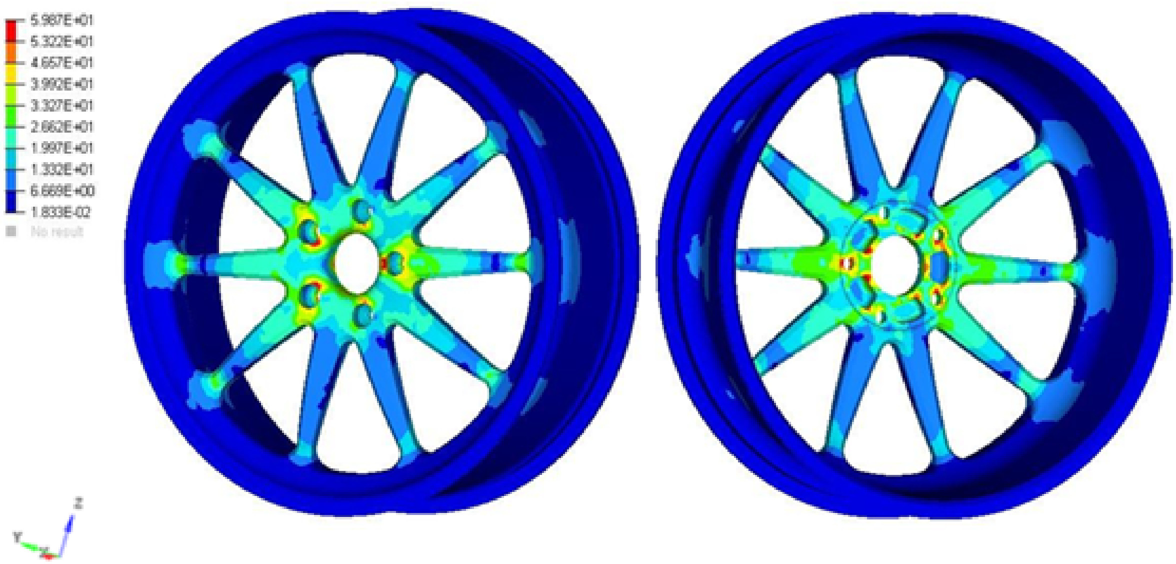


Figure 13: von Mises Stress for **Lateral Fatigue Analysis Critical Cases (**0^o **and** 18^o**)**

Table 7: Maximum von Mises Stress and Safety Factors for **Lateral Fatigue Analysis**

Lateral Fatigue	0^o	18^o	36^o
von Mises Stress [MPa]	59.87	59.87	59.53
S-N Curve Stress [MPa] (2×10^4 cycles)	230	230	230
Corrected Allowable Stress [MPa]	159.14	159.14	159.14
Fatigue Safety Factor	2.66	2.66	2.67

manufacturing process, material properties and the nature of impacts that the tire/wheel system suffers (CURSI; SAMPAIO, 2015), (SOIZE, 2017), (SCINOCCA; NABARRETE, 2020).

Therefore, the case of application of loading at 36^o in relation to the median horizontal plane of the system was analyzed, obtaining the maximum von Mises stress. Thus, it was possible to identify the points of greatest stress and deformation to which the component is subjected, as illustrated in Figure 14. Analyzing Figure 14 and Table 7, it is possible to verify the value referring to the maximum von Mises stress equal to 5.953×10^1 MPa. This critical stress is presented in the screws fixing region of the automotive wheel, resulting in a safety factor equivalent to 2.67.

Radial fatigue analysis

Subsequently, in the case of radial fatigue analysis of the automotive wheel, it is subjected to 4×10^5 cycles. Thus, the stress obtained through the S-N curve refers to its respective number of cycles (see Figure 2), having a value equal to 170 MPa. Thus, through Equation 3, the value of this corrected stress was obtained. Therefore, Table 8 presents the values of the maximum von Mises stress, the corrected stress of the S-N curve and the fatigue safety factor of the system for the respective analyzed cases. Thus, the critical case for lateral fatigue analysis is illustrated by Figure 15. The values obtained for S-N curve stress are associated with 4×10^5 cycles (see Figure 2). The values obtained for corrected allowable stress are associated with Equation 3.

Figure 15 presents the von Mises stress for the most critical case of the radial fatigue analysis of the automotive wheel, with a maximum value equal to 6.701×10^1 MPa. Thus, the safety factor of the automotive wheel is equal to 1.76, being reliable because it is greater than 1.0 (PRASAD; KUMAR, 2013), (MARINI; KEDZIORA, 2019). Thus, in the case of radial fatigue analysis, it is noted that the safety factor found for the critical case is 76% higher than the pre-established standard value. However, once again, it is necessary to consider and analyze the effects of uncertainties related to the manufacturing process, material properties, and the nature of impacts that the tire/wheel system suffers (CURSI; SAMPAIO, 2015), (SOIZE, 2017), (SCINOCCA; NABARRETE, 2020). Furthermore, the safety factor of this system can be reduced through optimization techniques, for example, with the objective of reducing wheel mass.

Thus, the other cases of this approach were analyzed, obtaining the maximum von Mises stresses. In this way, it is possible to identify the points of greatest stress and deformation that the system is subject to, as illustrated in Figure 16.

Analyzing Figure 16(a) and Table 8, it is possible to verify the value referring to the maximum von Mises stress equal to 6.433×10^1 MPa. This critical stress is presented on the left rear edge of the automotive wheel rim, resulting in a safety factor equivalent to 1.83. On the other hand, analyzing Figure 16(b) with Table 8, it is possible to observe the maximum von Mises stress equal to 6.674×10^1 MPa, and a safety factor equal to 1.76.

With the results obtained through these simulations, one can verify the importance of using the Finite Element Method for the development of automotive wheels. Likewise, this tool, the Finite Element Method, has

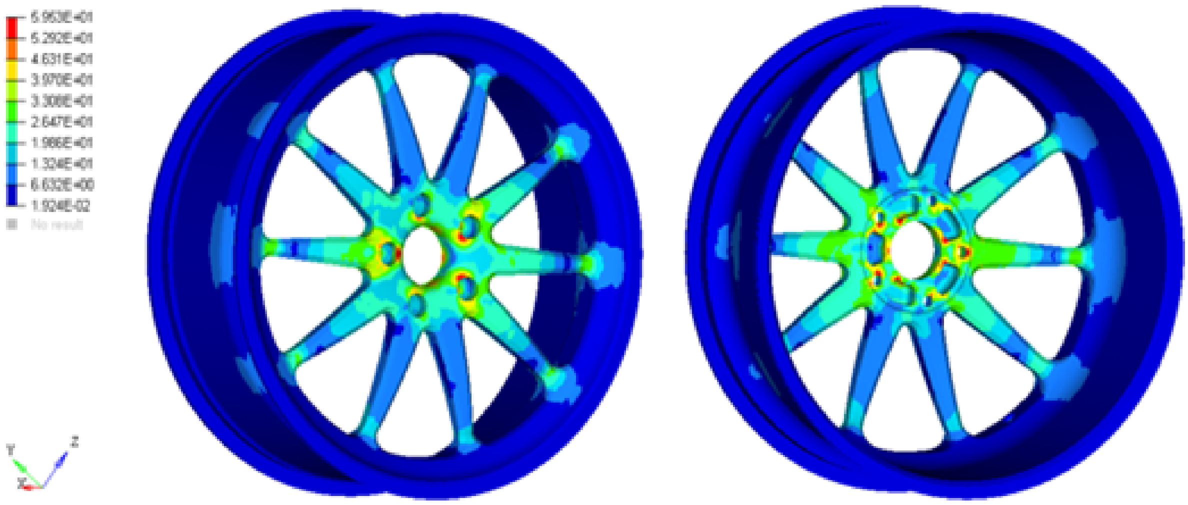


Figure 14: von Mises Stress for **Lateral Fatigue Analysis - Case** (36^o)

Table 8: Maximum von Mises Stress and Safety Factors for **Radial Fatigue Analysis**

Lateral Fatigue	0^o	18^o	36^o
von Mises Stress [MPa]	64.33	67.01	66.74
S-N Curve Stress [MPa] (4×10^5 cycles)	170	170	170
Corrected Allowable Stress [MPa]	117.62	117.62	117.62
Fatigue Safety Factor	1.83	1.76	1.76

high applicability in engineering fields, as well as in several other areas. Today, computer-aided finite element analysis (FEA) and computational fluid dynamics (CFD) was used to solve processes such as metal turning, bone drilling, bone screwing, water jet process and erosion-corrosion processes, the fatigue behavior of implant materials, simulations of COVID-19 and other infections and optimal configuration of implant materials (PRASAD; KUMAR, 2013), (GOK, A. e. a., 2014), (CURSI; SAMPAIO, 2015), (GOK, K. e. a., 2015), (SOIZE, 2017), (ERDEM, 2017), (SCINOCCA; NABARRETE, 2020), (GOK, K. e. a., 2021b), (ADA; ERDEM; GOK, 2021), (GOK, K. e. a., 2021a).

CONCLUSION

The present research confirms the great importance of carrying out structural analyzes using the FEM, and several conclusions could be addressed:

- In modal analysis, it can be noted the great importance of using this approach in order to avoid engineering problems, such as the resonance of components and the system as a whole. In addition, the first four modes above 300 Hz of the automotive wheel, excluding the six rigid-body mode;

- In the vertical impact analysis, it is noted that the safety factors are between 2.60 and 2.91. Thus, these values obtained range from 160% to 191% in relation to a pre-defined default value (safety coefficient equal to 1.0);

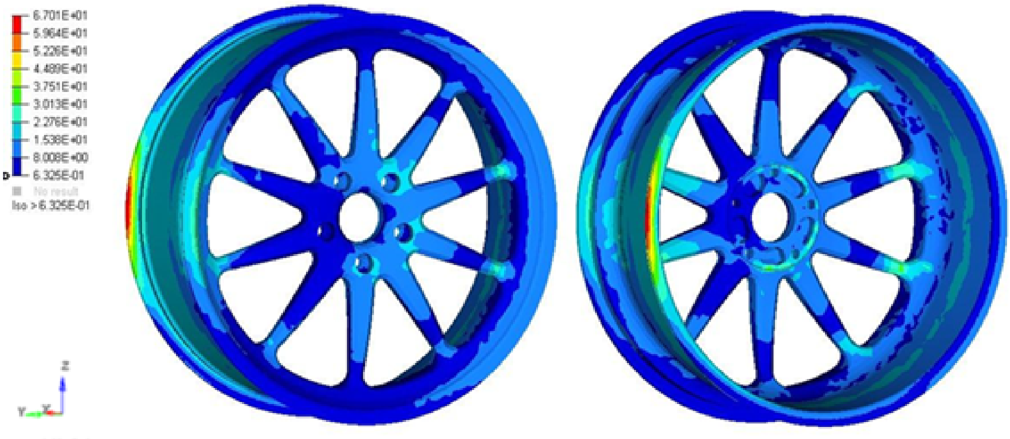


Figure 15: von Mises Stress for **Radial Fatigue Analysis - Case** (18^o)

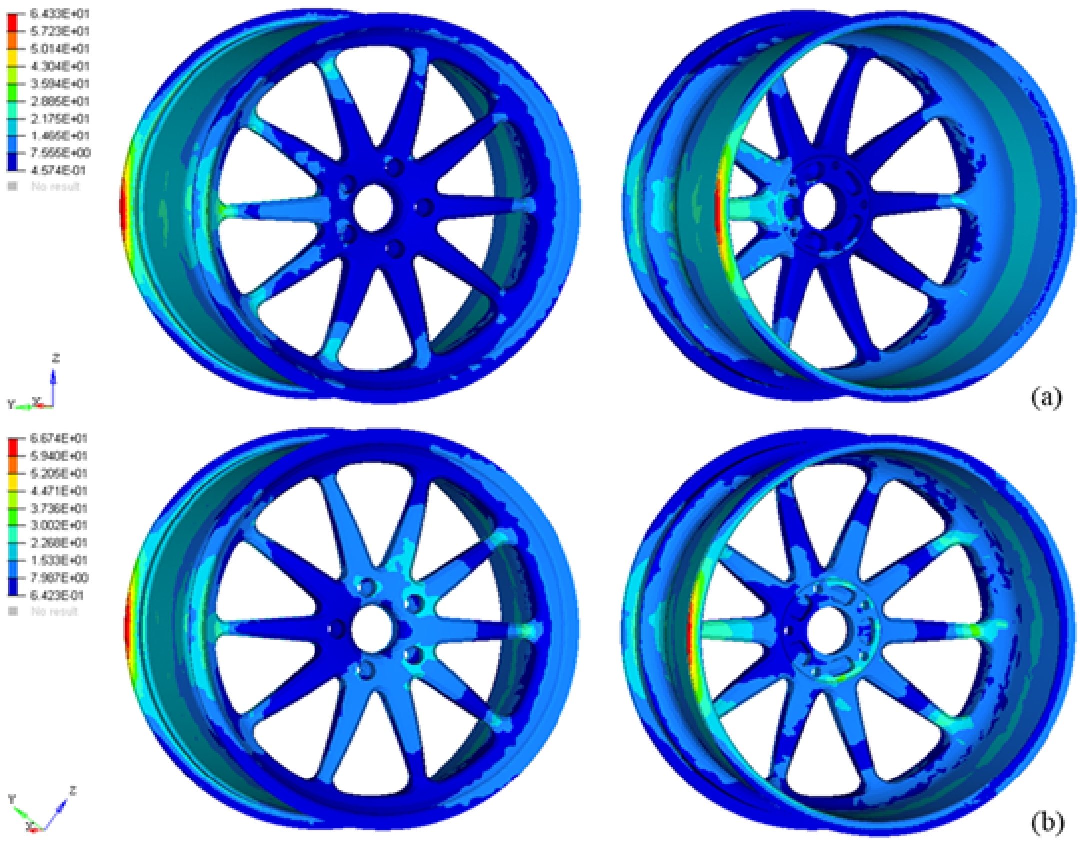


Figure 16: von Mises Stress for **Radial Fatigue Analysis - (a) Case** (0^o) **(b) Case** (36^o)

- In the case of lateral fatigue analysis, the safety factor assumes values equal to 2.66 and 2.67, with a safety margin of 166% for the most critical case;
- In the radial fatigue analysis, the safety factor values are equal to 1.76 and 1.83, with a safety margin between 76% to 83% in relation to the predetermined standardized value equal to 1.0;
- From the results obtained, it is possible to conclude that the types of material and the manufacturing process of components are important points to be considered in design development. Therefore, it can be noted that the type of material associated with the manufacturing process and component geometry have a great influence on issues related to vehicle safety. Such factors confirm the importance of carrying out in-depth and complex studies on these areas in order to ensure the development of effective, reliable, and safe mechanical systems.

REFERENCES

ADA, H.; ERDEM, M.; GOK, K. Computational fluid dynamics simulation of erosion-corrosion in abrasive water jet machining. **Surface Review and Letters**, v. 28, n. 5, p. 2150031, 2021.

ARGUS. **ARGUS MEDIA**. `https://www.argusmedia.com/en/metals`, 2022.

BAHARI, A. R. et al. Investigation on the effects of suspension stiffness using experimental modal analysis and finite element model updating. In: IOP Conference Series: Materials Science and Engineering. IOP Publishing. [S.l.: s.n.], 2019. P. 012043.

CHEN, M.l et al. Influence of zinc content on microstructure and mechanical properties of ZK60 magnesium alloy. **Journal of Alloys and Compounds,** v. 827, p. 154325, 2020.

CURSI, E.S.; SAMPAIO, R. **Uncertainty Quantification and Stochastic Modeling with Matlab**. [S.l.], 2015.

D'AMBROSIO, S.; VITOLO, R. Potencial impact of active tire pressure management on fuel consumption reduction in passenger vehicles. In: PROC IMechE Part D: Journal of Automobile Engineering, [s.l.: s.n.], 2018. v. 233, p. 261–275.

ERDEM, M. et al. Numerical analysis of temperature, screwing moment and thrust force using finite element method in bone screwing process. **Journal of Mechanics in Medicine and Biology**, v. 17, n. 1, p. 1750016, 2017.

FARRONI, F.; SAKHNEVYCH, A.; TIMPONE, F. A three-dimensional multibody tire model for research comfort and handling analysis as a structural framework for a multi-physical integrated system. In: PROC IMechE Part D: Journal of Automobile Engineering, [s.l.: s.n.], 2018. v. 33, p. 136–146.

________. Physical modelling of tire wear for the analysis of the influence of thermal and frictional effects on vehicle performance. In: PROC IMechE Part L: Journal of Materials: Design and Applications, [s.l.: s.n.], 2016.

GILLESPIE, T. D. **Fundamentals of Vehicle Dynamics**. [S.l.]: Society of Automotive Engineers Inc., 1992.

GOK, A. et al. Fatigue behaviors of different materials for schanz screws in femoral fracture model using finite element analysis. **Optoelectronics and Advanced Materials-Rapid Communications,** v. 8, p. 576–580, 2014.

GOK, K. et al. Development of bone chip-vacuum system in orthopedic drilling process. **J Brazilian Soc Mech Sci Eng,** v. 43, p. 1–11, 2021.

________. Development of three-dimensional finite element model to calculate the turning processing parameters in turning operations. **Measurement**, v. 75, 2015.

________. Investigation of the use of silicone pads to reduce the effects on the human face of classical face masks used to prevent from COVID-19 and other infections. In: 5. PROCEEDINGS of the Institution of Mechanical Engineers, Part E: Journal of Process Mechanical Engineering, [s.l.: s.n.], 2021. v. 235, p. 1742–1747.

GU, S. Application of finite element method in mechanical design of automotive parts. In: IOP Conf. Series: Materials Science and Engineering, [s.l.: s.n.], 2017. v. 231, p. 012180. DOI: `10.1088/1757-899X/231/1/012180`.

HASSAN, R. et al. Determination of Rayleigh Damping Coefficient for Natural Damping Rubber Plate Using Finite Element Modal Analysis. In: INCIEC 2014: Proceedings of the International Civil and Infrastructure Engineering Conference. [S.l.]: Springer Singapore, 2015. P. 713–725.

JAPAN, BBS. BBS JAPAN WHEEL CATALOG. In: [s.l.]: `https://bbs-japan.co.jp/en/wp-content/pdf/BBS_Catalog_2018_01_en.pdf`, 21 April 2022.

KABE, K. et al. Tire design methodology based on safety factor to satisfy tire life (simulation approach to truck and bus tire design). **Tire Science and Technology**, v. 33, n. 4, p. 195–209, 2005.

KUMAR, M.; SINGH, N. K. Static, Modal and Buckling Analyses of Automotive Propeller Shaft using Finite Element Methods. In: IOP Conference Series: Materials Science and Engineering. [S.l.]: IOP Publishing, 2018. v. 330, p. 012094.

LI, Z. **Mechanical Analyses of Multi-piece Mining**. [S.l.]: University of Windsor, 2014.

LIU, Q. et. al. Effect of La addition on microstructure and corrosion behavior of ZK60 magnesium alloy. **Materials Science and Engineering**, 2021. DOI: `10.1016/j.msea.2021.141166..`

LIU, S. et. al. Fatigue life assessment of centrifugal compressor impeller based on FEA. **. Engineering Failure Analysis**, v. 60, p. 383–390, 2016.

LIU, W. C. et al. High cycle fatigue behavior of as-extruded ZK60 magnesium alloy. **Journal of Materials Science**, v. 44, p. 2916–2924, 2009.

MARINI, L.; KEDZIORA, S. **Design of Automotive Road Racing Rim with Aid of Topology Optimization**. [S.l.], 2019.

MOHAMED, M. F. B. F.; AZMIR, N. A. B. Study on behavior of water treatment pump before and after modification using finite element modal analysis. In: IOP Conference Series: Materials Science and Engineering. [S.l.]: IOP Publishing, 2020. P. 012004.

NORTON, R. L. **Machine Design: An Integrated Approach.** 4th Edition. [S.l.]: . Pearson, 2011.

PRASAD, B. G. N. S.; KUMAR, M. A. Topology Optimization of Alloy Wheel. In: ALTAIR Technology Conference. [S.l.: s.n.], 2013.

RAMU, I.; MOHANTY, S. C. Modal analysis of Functionally Graded material Plates using Finite Element Method. **Procedia Materials Science,** v. 6, p. 460–467, 2014.

REIMPELL, J.; STOLL, H.; BETZLER, J. W. **The Automotive Chassis: Engineering Principles.** [S.l.]: Butterworth-Heinemann, 2001.

SAVITSKI, D. et al. Influence of the tire inflation pressure variation on braking efficiency and driving comfort of full electric vehicle with continuous anti-lock braking system. **SAE International Journal of Passenger Cars-Mechanical Systems,** v. 8, n. 01, p. 460–467, 2015.

SCINOCCA, F.; NABARRETE, A. Parametric Stochastic Analysis of a Piezoelectric Vibration Absorber Applied to Automotive Body Structure. **Journal of Vibration Engineering Technologies**, v. 8, p. 199–213, 2020.

SHARMA, J. K.; PARASHAR, S. K. .Experimental modal analysis using laser vibrometer and finite element modeling of milling machine arbor. **SN Applied Sciences**, v. 1, p. 1–10, 2019.

SOIZE, C. **Uncertainty Quantification: An Accelerated Course with Advanced Applications in Computational Engineering.** [S.l.]: Springer International Publishing, 2017.

SUN, Y. et al. Influence of microstructure on tensile properties and corrosion behavior of ZK60 magnesium alloy. **Journal of Alloys and Compounds**, 2021. DOI: `https://doi.org/10.1016/j.jallcom.2020.157222..`

WEI, J. et al. Copula-function-based analysis model and dynamic reliability of a gear transmission system considering failure correlations. **Fatigue Fracture of Engineering Materials Structures,** v. 42, n. 1, p. 114–128, 2019.

YILDIZ, Y.; DUZGUN, M. Stress analysis of ventilated brake discs using the finite element method. **International Journal of Automotive Technology**, v. 11, p. 133–138, 2010.

ZHANG, F. L. et al. Structural health monitoring of a 250-m super-tall building and operational modal analysis using the fast Bayesian FFT method. **Structural Control and Health Monitoring,** v. 26, n. 8, .e2383, 2019.

ZHU, S.; WANG, Z.; WU, X. Effect of plastic deformation and thermal treatment on microstructure and mechanical properties of ZK60 magnesium alloy. **Materials Science and Engineering**, 2019. DOI: https://doi.org/10.1016/j.msea.2019.01.073..