Jurnal Asiimetrik: Jurnal Ilmiah Rekayasa Dan Inovasi
Terakreditasi "**Peringkat 3**" berdasarkan SK Dirjendiktiristek, Nomor: 225/E/KPT/2022
Volume 6 Nomor 2 Tahun 2024, Juli 2024, Halaman 283-294
Website : http://journal.univpancasila.ac.id/index.php/asiimetrik/

p-ISSN 2655-1861
e-ISSN 2716-2923

Analysis of Vibration Characteristics in 17-Inch Aluminum Alloy Wheel Rims Using Finite Element Method

Analisis Karakteristik Getaran pada Pelek Paduan Aluminium Ring 17 Menggunakan Metode Elemen Hingga

Victor Indra Wijaya, Riyan Ariyansah, Delvis Agusman*, Rifky, Oktarina Heriyani
Teknik Mesin, Fakultas Teknologi Industri dan Informatika, Universitas Muhammadiyah Prof. DR. HAMKA, Jakarta 13830, Indonesia

---

Article information:

Received:
03/05/2024
Revised:
13/05/2024
Accepted:
26/05/2024

Abstract

The wheel rim serves as a component designed to provide stability and necessary support for vehicle tires. The phenomenon of resonance occurring when the external vibration frequency approaches the natural frequency of the structure has the potential to increase vibration amplitude, which can lead to wheel rim damage. This study aims to investigate the shape patterns and natural frequencies of 17-inch aluminum alloy wheel rims, as well as to analyze the maximum total deformation that occurs. The finite element method is employed to simulate the vibration characteristic of 17-inch aluminum alloy wheel rims under various natural frequencies. Modifications to the wheel rim design are made by altering the spoke angle in variations of 5°, 10°, and 15°. The vibration characteristic data of the wheel rim was obtained through simulation using ANSYS software. The research findings indicate that the natural frequencies range from approximately 364.7 Hz to 723.21 Hz. Furthermore, the maximum total deformation values range from approximately 9.7 mm to 22.5 mm.

Keywords: wheel rim, natural frequency, mode shape, total deformation, ANSYS.

***SDGs*:**

Abstrak

Pelek merupakan komponen yang berfungsi untuk memberikan stabilitas dan dukungan yang diperlukan terhadap ban kendaraan. Fenomena resonansi yang terjadi ketika frekuensi getaran eksternal mendekati frekuensi alami struktur memiliki potensi untuk meningkatkan amplitudo getaran yang dapat merusak pelek. Penelitian ini bertujuan untuk menyelidiki pola bentuk dan frekuensi alami dari pelek paduan aluminium berukuran 17 inci, serta menganalisis deformasi total maksimum yang terjadi. Metode elemen hingga digunakan untuk mensimulasikan perilaku getaran dari pelek roda aluminium 17 inci pada berbagai frekuensi natural. Modifikasi desain pelek dilakukan dengan mengubah sudut spoke dalam variasi 5°, 10°, dan 15°. Data karakteristik getaran pelek diperoleh melalui simulasi menggunakan perangkat lunak ANSYS. Hasil penelitian menunjukkan bahwa frekuensi alami berkisar antara sekitar 364,7 Hz hingga 723,21 Hz. Selain itu, nilai deformasi total maksimum berkisar antara sekitar 9,7 mm hingga 22,5 mm.

Kata Kunci: pelek roda, frekuensi alami, bentuk mode, deformasi total, ANSYS.

---

*Correspondence Author
email : delvisagusman@uhamka.ac.id

1. INTRODUCTION

Wheels constitute a vital component in motor vehicles, responsible for supporting the vehicle's load and ensuring the safety and comfort of the users (Pris, Suyitno and Suhadi, 2019). The selection of wheel materials and appropriate dimensions is a crucial factor in designing a reliable and efficient car wheel. Aluminum alloys are widely used in various applications in the aerospace and automotive industries (Stojanovic, Bukvic and Epler, 2018; Li *et al.*, 2023). In recent studies, aluminum alloys have become popular materials for manufacturing car wheels with various design variations (Vijayakumar *et al.*, 2020; Loganathan and Subramani, 2023). Wheels made from aluminum alloy have advantages in terms of strength, lightweight, and corrosion resistance (Gadwala and Babu G, 2022).

Moreover, another important factor influencing the structural strength of wheels is natural frequency vibration. Natural frequency refers to the inherent frequency of vibrations that occur in a specific structure (Prasetiyo *et al.*, 2022). This property depends on the geometric characteristics, materials, and structural constraints involved. The resonance phenomenon, which occurs when the external vibration frequency approaches the natural frequency of the structure, can cause an increase in vibration amplitude that may potentially damage wheel components (Win and Oo, 2019). Furthermore, repetitive vibrations at frequencies close to the natural frequency can induce material fatigue, leading to structural failure. Thus, research is needed to analyze the influence of natural frequency vibration on the structural strength of aluminum alloy wheels, thereby enabling the development of stronger aluminum alloy wheel designs to withstand natural frequency vibrations.

In efforts to analyze the influence of natural frequency on the structural strength of wheels, the finite element method has become a highly useful and important tool. The finite element method is a mathematical approach used in finite element analysis (FEA) to represent structural behavior. FEA refers to the analysis process using numerical techniques, where a complex structure is divided into small parts called elements. Each of these elements has certain mathematical properties that approximate the original structure's properties. Through this division, the overall behavior of the structure can be numerically estimated using CAD/CAE-based software.

The utilization of CAD/CAE-based software for conducting 3D modeling, prototype design, and specific testing simulations on a physical object or geometric entity is frequently encountered in the current era of digital technological advancement (Ariyansah, 2019; Hermanto, Ariyansah and Gamayel, 2021; Octavianus, Gamayel and Ariyansah, 2022). In this regard, finite element analysis utilizing ANSYS software has been widely employed as a tool to identify and analyze the influence of natural frequency vibrations on the structural strength of a geometric object, including automotive wheel components (Rizqi, Rifky and Ariyansah, 2023; Sofyan, Gamayel and Zaenudin, 2023). ANSYS software encompasses a preprocessor for constructing physical models, a solver for solving mathematical equations, a postprocessor for analyzing results, and additional modules for advanced analysis (Yulianto, Ariyansah and Octavianus, 2023).

Previous literature studies underscore the importance of structural analysis in the development of vehicle wheels made of aluminum 6061-T6 alloy (Alfarisi, Ariyansah and Mugisidi, 2024). This research utilized a load of 3000 N and a pressure of 800 N to compare the performance of three-wheel models using the FEA method. The results indicated that model 3 of the aluminum 6061-T6 alloy wheel exhibited the best performance. Model 3 demonstrated a von Mises stress of 11.02 MPa, deformation of 0.021 mm, equivalent strain of 0.000096, and a safety factor of 25. It was concluded in this study that model 3 wheel is a safer and more efficient choice in the development of aluminum-based vehicle wheels (Alfarisi, Ariyansah and Mugisidi, 2024).

Prior research on the lightweight design of aluminum alloy wheels highlights the significance of sophisticated computer technologies and finite element analysis (FEA) methods in improving vehicle safety and reliability (Guiju and Caiyuan, 2018).

A solid model of the wheel was developed using Solidworks and evaluated with ANSYS to assess its modal properties or vibration characteristics. The analysis revealed that the aluminum alloy wheel experienced maximum stress well below the allowable limit and achieved the required inherent frequencies. The design proposed based on FEA data achieved a weight reduction of 13.9% for the wheel. These results corroborate previous studies, confirming that Computer-Aided Engineering (CAE) technology is effective in optimizing designs, reducing development time, and lowering costs.

Furthermore, previous studies have also emphasized the significance of spoke design in aluminum alloy wheels for MPV vehicles, aiming to enhance safety and performance through spoke design optimization (Naufal, 2019). A numerical study was conducted to develop five variations of wheel designs to analyze the impact of varying the number and thickness of spokes on stress and displacement. The numerical study demonstrated that all wheel designs remained below the permissible stress limit of 190.3 N/mm^2. The research findings indicated that an increase in the number of spokes with consistent thickness resulted in reduced stress and displacement, thereby enhancing the wheel's strength.

Furthermore, previous studies have also investigated the vibration characteristics of alloy rims, which are crucial in enhancing the performance and durability of automotive systems (Somayaji *et al.*, 2022). Specifically, rim edge stiffness has emerged as a critical parameter influencing its lifespan. Finite Element Analysis (FEA) has been widely employed to assess material choices, with Aluminium 7079 alloy reinforced with 8% carbon fibers (CF) emerging as a promising candidate. This study evaluates the modal properties of various materials, including Steel Alloy, Forged Steel, Magnesium Alloy, and Aluminium 7079, alongside the optimized Al7079 composite with 8% CF. The research findings elucidate the superior performance of Al7079 with 8% CF, characterized by higher natural frequencies, thus underscoring its potential to enhance automotive rim performance (Somayaji *et al.*, 2022).

Subsequently, in a study conducted by Sivaraj et al., focus was given to the natural frequency performance of wheels through modal analysis under free and loaded conditions (Sivaraj, Nagendharan and Mohanavel, 2020). Through the utilization of DEWE SOFT modeling software, risks associated with design and manufacturing could be minimized. The analysis compared the natural frequencies of wheels when unloaded and when loaded by a vehicle, with the aim of understanding the impact of free vibration and end vibration on vehicle efficiency. Static and dynamic analysis methods were employed to investigate wheel behavior in various scenarios, while results from both conditions were utilized to recommend the best material for wheel rims, expected to enhance overall vehicle performance and inform future design and material techniques.

Previous investigations have also scrutinized the durability of 17-inch aluminum alloy rims, employing Finite Element Analysis (FEA) to assess three distinct rim designs (Pris, Suyitno and Suhadi, 2019). The results delineate the superiority of rim model 2 over types 1 and 3, demonstrating advantages in terms of displacement values in static simulation 0.008576 mm, Von Mises Stress 96.29 MPa, displacement 0.2189 mm, strain in torque simulation 0.0008832 mm/mm, and displacement in FEA simulation 0.02311 mm. Although there have been studies conducted on the structural analysis of wheel rims, research specifically on modal analysis, particularly on aluminum alloy rims with a size of 17 inches, remains limited. Therefore, the novelty of this research lies in identifying the vibration characteristics in modified designs of 17-inch aluminum alloy rims using the finite element method. The objective of this study is to study the mode shapes and natural frequencies of 17-inch aluminum alloy rims and investigate the maximum total deformation that occurs.

2. METHODOLOGY

In this study, a simulation approach is used to identify the vibration characteristics of 17-inch wheel rims using ANSYS software. The flowchart shown in Figure 1 outlines the steps of the research process in the next section.

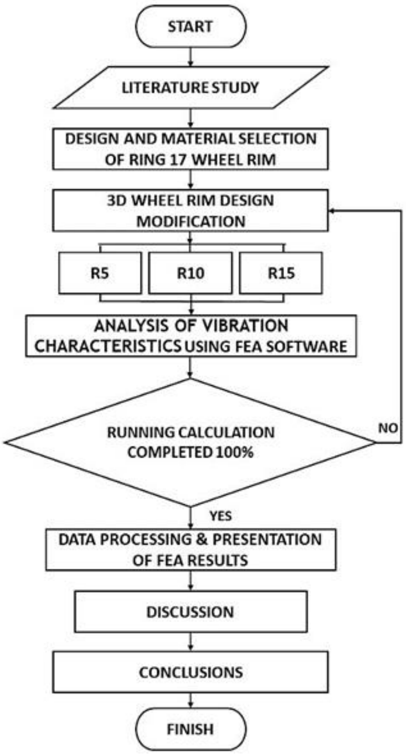


Figure 1. Research flowchart.

Before commencing the simulation, a literature review of relevant sources is conducted. Next, three-dimensional design of the 17-inch wheel rim is carried out using CAD-based software. Following this, simulation is carried out on the 3D wheel rim model using ANSYS R2 2023 student version software. If the simulation results indicate no program errors, they will be processed and presented in the form of data tables or visualized images with color gradients on the 3D wheel rim model. Consequently, data processing and discussion of the previous modal analysis simulation results using ANSYS can be obtained to draw several research conclusions.

2.1. Specifications of Research Tools

2.1.1. Laptop

The simulation with modal analysis on the 17-inch wheel rim requires a laptop or computer with adequate specifications. The laptop used in this simulation has an AMD Ryzen 3 series 5000 processor, 8 GB RAM, 512 GB SSD memory, and AMD RADEON graphics card. The specifications of the laptop used are sufficient to run complex simulations that require intensive processing.

2.1.1. Software ANSYS

In this study, the ANSYS simulation software student version is utilized. ANSYS is a software package that possesses the finite element analysis capabilities required to identify natural frequencies and mode shapes in the structure of the 17-inch wheel rim. This software provides an official license for free for academic research, which has been widely adopted by researchers due to its reliability in producing simulations that approach accuracy and are adequate for research related to vibration characteristic identification.

2.2. Specifications of Ring 17 Wheel Rim

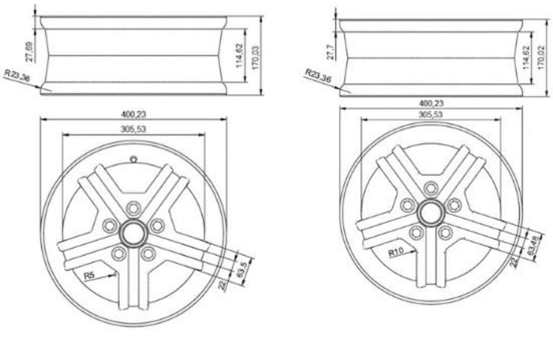
The material used for the 17-inch wheel rim employs aluminum alloy 6061, which is highly suitable for car rim manufacturing. This alloy is renowned for its combination of properties, including high strength, lightweight, corrosion resistance, and ease of processing. Consequently, aluminum alloy 6061-T6 emerges as the ideal choice for car rims. Table 1 below presents several material properties of aluminum alloy, wrought, 6061, T6, which are crucial parameters influencing the strength, stiffness, and elasticity of the material in automotive wheel rim applications.

Table 1. Properties of aluminum alloy, Wrought, 6061, T6 (Ashby, 2016).

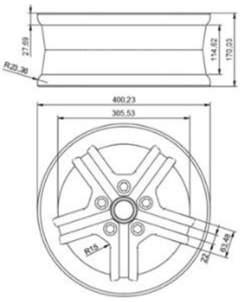
Property	Value	Unit
Density	2713	kg/m^3
Young modulus	69040	MPa
Bulk Modulus	67,686	MPa
Shear Modulus	25,955	MPa
Poisson's Ratio	0,33	N/A

Table 1 indicates that aluminum alloy 6061 in the T6 condition possesses mechanical properties suitable for applications requiring strength, stiffness, and dimensional stability. The high Young's modulus signifies good stiffness, while the relatively low shear modulus indicates the material's ability to withstand shear stresses. The moderate Poisson's ratio suggests that the material is not highly prone to permanent deformation in the direction of the applied stress.

This study presents a modified design of a 17-inch wheel rim, where the diameter remains consistent at 17 inches. Figure 2 illustrates the altered dimensions of the rim spokes, specifically showcasing variations in spoke angle radius by increments of 5 degrees, 10 degrees, and 15 degrees, denoted as R5, R10, and R15 respectively. Despite these modifications, the number of spokes on the rim remains unchanged. The detailed representation in Figure 2 enables a thorough visualization and comparison of the various spoke angle configurations, facilitating valuable insights into their potential impact on the vibration characteristics of the wheel rims.



(a) R5 (b) R10



(c) R15

Figure 2. The modified spoke angle dimensions of the 17-inch wheel rim.

2.3. Meshing of Ring 17 Wheel Rim

During the simulation process, mesh adjustment assumes a critical role, significantly impacting both accuracy and computational efficiency. The meshing phase entails the segmentation of the physical object into smaller, more manageable components referred to as elements. These elements are interconnected via nodes or points, forming either a structured or unstructured network.

At this juncture, the element size of the wheel rim body for each model is standardized to 10 mm, with the mesh physics preference set to mechanical. The quality of the mesh is paramount, given its potential to influence both the accuracy and computational efficiency of the simulation. Table 2 provides comprehensive data concerning the quantity of elements and nodes across various levels of mesh modification, designated as R5, R10, and R15.

Table 2. Mesh data.

Modification	Elements	Nodes
R5	74378	123218
R10	73481	121428
R15	73202	121026

From the data presented in Table 2, it is evident that despite the consistent mesh element size, there is a gradual reduction in the number of elements and nodes from the R5 to R15 modifications. This decrease signifies efforts to enhance computational efficiency by reducing model complexity. In the structural physics scenario, the ANSYS student version software imposes a maximum limit of 128,000 elements or nodes. Consequently, the meshing process conducted has effectively optimized the accuracy of the calculation results performed by the ANSYS student software within the prescribed limitations

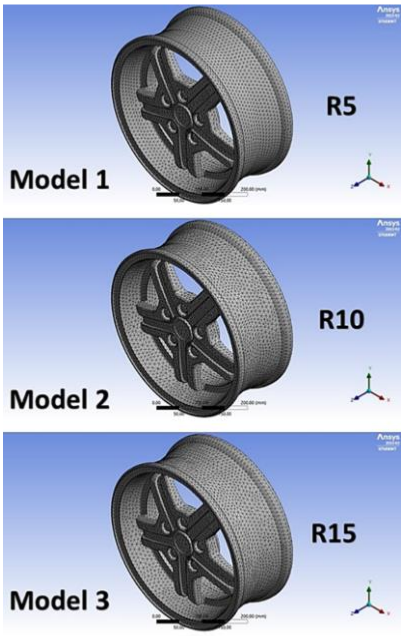


Figure 3. Meshing visualization.

Figure 3 provides a visualization of the meshing process elucidated in the preceding Table 2. Through this visualization, a clearer understanding can be gained regarding how mesh modifications influence the geometric representation of the system within the context of the simulation being conducted. By examining the image, one can observe the distribution of mesh elements and nodes, as well as the effects of these modifications on the complexity and detail of the model representation.

2.4. Boundary Conditions

In this study, boundary conditions are set for fixed support areas on the surfaces of the 5-wheel rim bolts, as illustrated in Figure 4. This implies that these bolt locations are constrained from any movement in all directions, simulating a fixed support condition. Additionally, the type of analysis employed is modal analysis with structural physics. Furthermore, for each model of the 17-inch wheel rim, 6 mode shapes are sought. The fixed support boundary conditions ensure that the wheel rim remains stable and stationary at the bolt locations, allowing for a more accurate modal analysis. By constraining these specific areas, the analysis focuses on understanding the natural frequencies and mode shapes of the wheel rim structure under various loading conditions, without the interference of movement at the bolt locations.

Figure 4. Fix support area.

Furthermore, the boundary conditions must ensure that there is no lateral movement on the ground surface where the vehicle is located during the analysis, allowing a focus on the structural response of the wheel without accounting for additional movements from the surrounding environment. By applying appropriate boundary conditions, this research can provide better insights for designing 17-inch aluminum alloy wheel rims with enhanced resistance to deformation caused by natural frequencies.

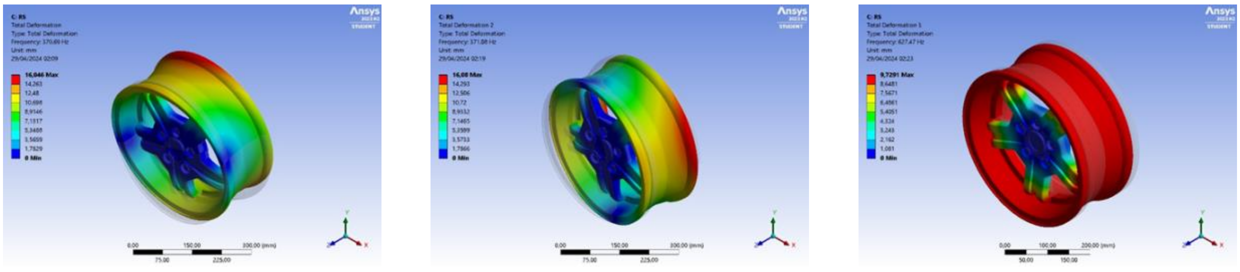
3. RESULTS AND DISCUSSION

The analysis of vibration characteristics, conducted through simulations utilizing ANSYS software, elucidates various vibration modes present within the 17-inch aluminum alloy wheel rim. Each modified model displays distinct frequencies, shapes, and deformations. Figure 5 illustrates the vibration characteristics for the 17-inch wheel rim with a modified spoke angle of 5 degrees, depicted across six different modes. From mode 1 to mode 6, each mode illustrates a distinct vibration pattern within the rim. This analysis of vibration characteristics provides a profound understanding of the structural response of the rim to natural frequencies and deformation due to vibration.

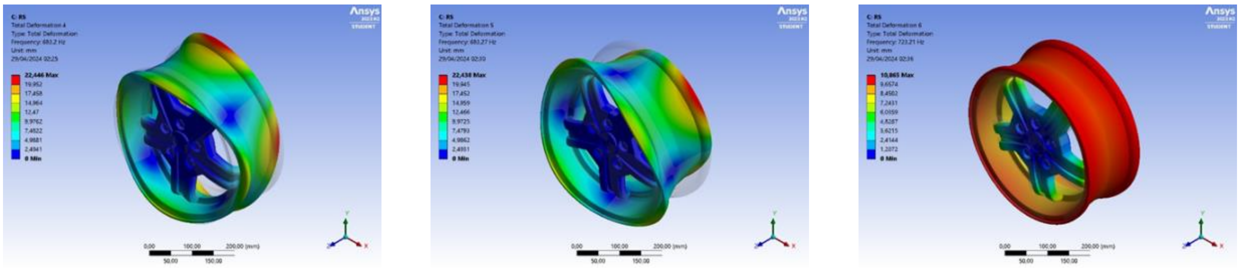
From Figure 5, it can be observed that the first mode for the 17-inch wheel rim design, model 1, is detected at a frequency of 370.69 Hz, with a maximum total deformation of 16.046 mm. The second mode has a frequency of 371.08 Hz with a maximum total deformation of 16.08 mm. Despite the similarity in frequency with the first mode, variations in vibration shape and amplitude indicate different dynamic behaviors within the system. Furthermore, the third mode shape occurs at a frequency of 627.47 Hz with a maximum total deformation value of 9.7291 mm.

The fourth mode occurs at a frequency of 683.2 Hz with a maximum total deformation of 22.446 mm, while the fifth mode occurs at a frequency of 683.27 Hz with a maximum total deformation of 22.438 mm. Finally, the sixth mode is detected at a frequency of 723.21 Hz with a maximum total deformation of 10.865 mm. This analysis illustrates the variation in dynamic response of the rim with model 1 modification across various vibration modes, crucial for a better understanding of the structural behavior.

Subsequently, visualizations of mode shapes resulting from simulations for each mode shape in the 17-inch wheel rim modification model 2 with R10 spoke angle variation are presented in Figure 6. The modal analysis for the 17-inch wheel rim design, model 2, is depicted in the simulation result visualizations as shown in Figure 6. The first mode is detected at a frequency of 364.7 Hz, with a maximum total deformation of 16.055 mm.

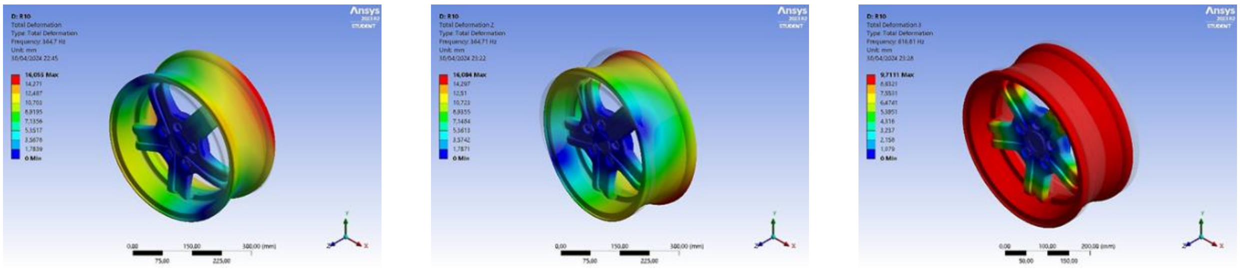


(a) Mode 1 (b) Mode 2 (c) Mode 3

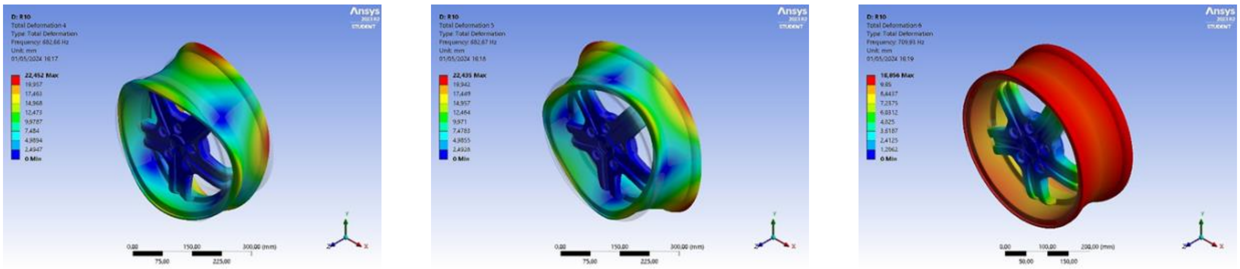


(d) Mode 4 (e) Mode 5 (f) Mode 6

Figure 5. Vibration characteristics for modified 17-inch wheel rims with R5 spoke angle.



(a) Mode 1 (b) Mode 2 (c) Mode 3



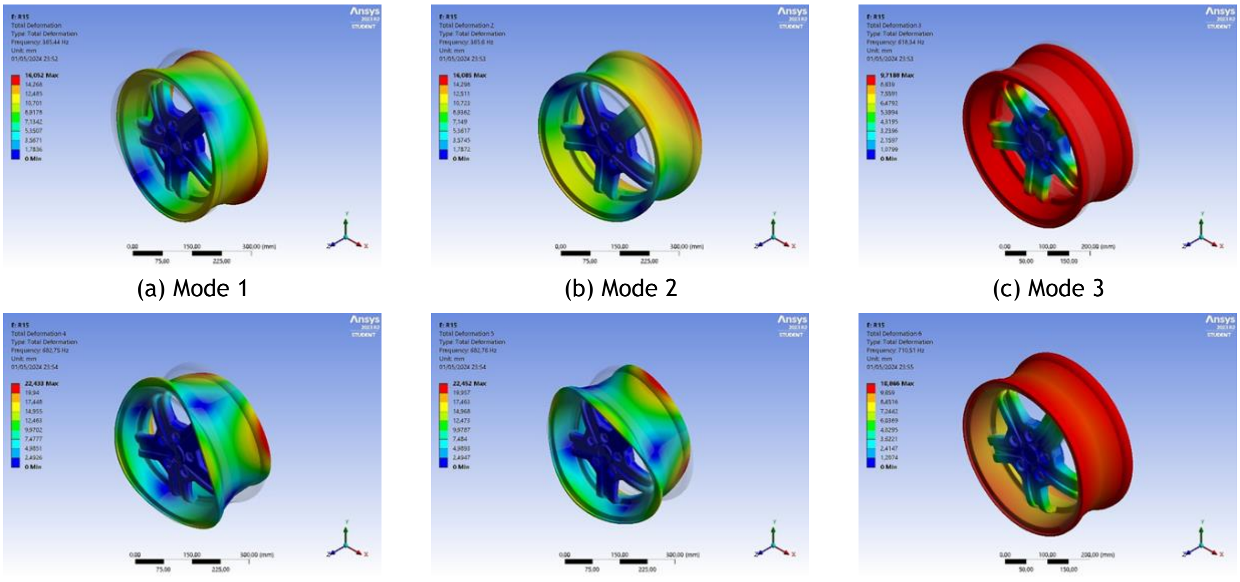
(d) Mode 4 (e) Mode 5 (f) Mode 6

Figure 6. Vibration characteristics for modified 17-inch wheel rims with R10 spoke angle.

Meanwhile, the second mode has a frequency of 364.71 Hz and a maximum total deformation of 16.084 mm. Although both modes have similar frequencies, variations in shape and vibration amplitude indicate differences in dynamic behavior within the system.

Furthermore, the third mode shape occurs at a frequency of 616.81 Hz with a maximum total deformation of 9.71 mm. The fourth mode, with a frequency of 682.66 Hz, exhibits a maximum total deformation of 22.482 mm. Meanwhile, the fifth mode has a frequency of 682.67 Hz with a maximum total deformation of 22.435 mm. Finally, the sixth mode is detected at a frequency of 709.93 Hz with a maximum total deformation of 10.855 mm. This analysis provides insights into the variation in response to natural frequencies of the rim with model 2 modification across various vibration modes.

Next, visualizations of mode shapes resulting from simulations for each mode shape in the 17-inch wheel rim modification model 3 with R15


(a) Mode 1 (b) Mode 2 (c) Mode 3

(d) Mode 4 (e) Mode 5 (f) Mode 6

Figure 7. Vibration characteristics for modified 17-inch wheel rims with R15 spoke angle.

spoke angle variation are presented in Figure 7. The modal analysis for the 17-inch wheel rim, model 3, is depicted in the simulation result visualizations as shown in Figure 7. In the first mode, the detected frequency is approximately 365.44 Hz, with a maximum total deformation reaching 16.052 mm. Conversely, the second mode appears at a frequency of 365.6 Hz, with the maximum total deformation reaching 16.085 mm. Although both modes have nearly identical frequencies, variations in shape characteristics and vibration amplitude indicate differences in dynamic behavior within the system.

Furthermore, the third mode is observed at a frequency of 618.34 Hz, exhibiting a maximum total deformation of approximately 9.7188 mm. On the other hand, the fourth mode, with a frequency of approximately 682.75 Hz, displays a maximum total deformation of 22.433 mm. Additionally, the fifth mode reaches a frequency of approximately 682.76 Hz, with a maximum total deformation of 22.452 mm. Finally, the sixth mode is detected at a frequency of approximately 710.51 Hz, with a maximum total deformation of 10.866 mm. This analysis provides a comprehensive understanding of the variation in response to natural frequencies of the rim with the application of model 3 modification across various observed vibration modes.

Table 3. Data of total deformation modal simulation results for 17-inch wheel rim.

Mode Shape	Total Deformation Maximum (mm)		
	Model 1	Model 2	Model 3
1	16,046	16,055	16,052
2	16,08	16,084	16,085
3	9,7291	9,7111	9,7188
4	22,456	22,452	22,433
5	22,438	22,435	22,442
6	10,865	10,856	10,866

Table 3 presents the maximum total deformation values obtained from the three models for each mode shape. This data is crucial for understanding the structural response of wheel rims to different dynamic loads and ensuring their structural integrity and performance under various operational conditions. The analysis of total deformation provides valuable insights into the extent of shape changes experienced by the wheel rims when subjected to specific loads. The results are presented in Table 3, which includes the six primary mode shapes. These findings will be further discussed to interpret the differences observed between the models and their implications for wheel rim design and application. From Table 3, it can be observed that the maximum total deformation values range from approximately 9.7 mm to 22.5 mm.

The difference in total deformation between the three models is relatively small, indicating that design changes among these models may not significantly impact deformation. Higher total deformation values may lead to decreased structural integrity and potential compromise in the performance and safety of the wheel rim, particularly under dynamic loading conditions.

Table 4. Data of natural frequency modal simulation results for 17-inch wheel rim.

Mode Shape	Frequency (Hz)		
	Model 1	**Model 2**	**Model 3**
1	370,69	364,7	365,44
2	371,08	364,71	365,6
3	627,47	616,81	618,34
4	683,2	682,66	682,75
5	683,27	682,67	682,76
6	723,21	709,93	710,51

Table 4 presents the natural frequencies obtained from the three models for each mode shape. This data is expected to provide insights into the comparison of natural frequencies between the models and to identify the differences and similarities among them. The analysis of natural frequencies is crucial for understanding the vibration characteristics of wheel rims. The natural frequencies obtained from each model are shown in Table 4, which includes the six primary mode shapes. These results will be further discussed to interpret the differences that emerge between the models and their implications for wheel rim design and application.

Regarding the data in Table 4, the modal natural frequencies range from approximately 364.7 Hz to 723.21 Hz. Certain modes have higher frequencies compared to others. Generally, the higher the modal natural frequency, the higher the structural stiffness of the wheel rim, indicating a lower likelihood of significant deformation occurring at that frequency. Although there is slight variation in frequencies among the models, there is a relatively high consistency in vibration patterns. This suggests that the modal characteristics of the wheel rim are relatively stable and reliable, regardless of minor differences in design.

Thus, the findings of this study are consistent with previous research indicating that in mode shapes 1 and 2 of aluminum alloy wheel rims, the natural frequencies are close, with mode 1 at a frequency of 515.27 Hz and mode 2 at a frequency of 515.8 Hz, with only a difference of 0.53 Hz (Somayaji et al., 2022). Similarly, in the findings of this study, for wheel rim model 1, the difference between mode 1 and mode 2 is 0.39 Hz. Furthermore, the same trend is observed for mode shapes 3 through 6, experiencing slight variations in frequency increases, with each frequency exhibiting a different wheel rim mode shape. The differences in natural frequency values and mode shapes in this study are attributed to differences in wheel rim design factors, despite having the same number of spokes, which is 5 spokes.

The observed vibration patterns in the simulation reflect the primary shape modes of the rim, determined by its structural geometry. Dominant modes, indicated in red, exhibit the highest deformations and may occur in specific areas of the rim that are more sensitive to frequencies. Identifying these patterns provides better understanding of areas prone to fatigue or structural failure. Strengthening or improving these areas can enhance overall rim performance and longevity.

Thus, this research supports previous findings suggesting that variations in spoke geometry can influence total deformation values (Dani *et al.*, 2020). This phenomenon arises because modifications to the rim spokes alter the rim's weight distribution. Extreme changes in rim geometry may lead to suboptimal weight distribution, compromising its structural integrity. Therefore, the rim geometry modifications in this study were not excessively drastic to avoid significant loss of structural strength. Optimized designs can yield rims that are stronger, lighter, and more efficient. Moreover, appropriate design can minimize the risk of failure or accidents resulting from structural deformation or unwanted resonance frequencies.

4. CONCLUSION

This study concludes that the highest natural frequency is found in Mode 6 at 723.71 Hz with a modified spoke angle of 5 degrees. Conversely, the lowest natural frequency is observed in Mode 1 at 364.7 Hz with a modified spoke angle of 10 degrees. Furthermore, the maximum total deformation is recorded in Mode 4 at 22.456 mm with a modified spoke angle of 5 degrees, whereas the minimum total deformation is found in Mode 3 at 9.7111 mm with a modified spoke angle of 10 degrees.

The research conclusion underscores the importance of a comprehensive comprehension of structural responses to natural frequencies and deformation induced by vibrations, as emphasized in this study. Furthermore, our findings regarding the impact of modifications to rim spoke geometry on total deformation values offer valuable insights. Alterations to rim spokes can potentially influence the structural integrity of the rim by affecting its weight distribution.

Moreover, this research lays the groundwork for the development of enhanced rim designs by considering modal responses to geometric variations, such as spoke angles and quantities. The application of FEA methods, as demonstrated in this study, can facilitate the optimization of rim designs to attain an optimal balance between performance, safety, and efficiency. Furthermore, future investigations could delve deeper into understanding the ramifications of structural deformation on car rims during both regular and extreme operational conditions. This could aid in pinpointing areas susceptible to material fatigue and in devising suitable repair or reinforcement methodologies.

REFERENCES

Alfarisi, S., Ariyansah, R. and Mugisidi, D. (2024) ‘Finite Element Analysis of Aluminum Based R22 Car Wheel Design Modification’, *Jurnal Asiimetrik: Jurnal Ilmiah Rekayasa dan Inovasi*, 6(1), pp. 1–10.

Ariyansah, R. (2019) ‘Modifikasi Desain Chasis Kendaraan Hybrid Pada Bus Scania K360Ib’, *Teknobiz : Jurnal Ilmiah Program Studi Magister Teknik Mesin*, 7(3), pp. 156–165.

Ashby, M.F. (2016) *Materials and Sustainable Development*. [Print].

Dani, P.A. *et al.* (2020) ‘Influence of Material and Spoke Pattern on the Performance of Automotive Wheels’, in *Materials Today: Proceedings. 2nd International Conference on Materials Manufacturing and Modelling (ICMMM - 2019)*, Tamil Nadu, India: Vellore Institute of Technology (2nd International Conference on Materials Manufacturing and Modelling, ICMMM – 2019, VIT University, Vellore, 29th - 31st March 2019), pp. 1452–1459.

Gadwala, W.K. and Babu G, R. (2022) ‘Modeling And Analysis Of Car Wheel Rim For Weight Optimization To Use Additive Manufacturing Process’, in *Materials Today: Proceedings. International Conference on Advances in Materials and Mechanical Engineering (ICAMME-2022)*, India (International Conference on Advances in Materials and Mechanical Engineering), pp. 336–345.

Guiju, Z. and Caiyuan, X. (2018) ‘Study On Lightweight Design Of Aluminium Alloy Wheels’, *Australian Journal of Mechanical Engineering*, 16(1), pp. 30–36.

Hermanto, Ariyansah, R. and Gamayel, A. (2021) ‘Analisis Kekuatan Struktur Rangka Pembangkit Listrik Sepeda Statis Menggunakan Perangkat Lunak Ansys Workbench’, *Teknobiz : Jurnal Ilmiah Program Studi Magister Teknik Mesin*, 11(1), pp. 20–25.

Li, S. *et al.* (2023) ‘Development And Applications Of Aluminum Alloys For Aerospace Industry’, *Journal of Materials Research and Technology*, 27, pp. 944–983.

Loganathan, K. and Subramani, S.K.V.S. (2023) ‘Design And Optimization Of Aluminium Alloy Wheel Rim In Automobile Industry’, *Materials Today: Proceedings*, Article in Press, pp. 1–6.

Naufal, H. (2019) ‘Studi Numerik Pengaruh Variasi Jumlah Dan Ketebalan Spoke Pada Velg Paduan Aluminium Untuk Mobil MPV’, *Jurnal Teknik Mesin*, 8(1), pp. 39–47.

Octavianus, G., Gamayel, A. and Ariyansah, R. (2022) ‘Simulasi Kekuatan Struktur Model Variasi Cakram Sepeda Motor Menggunakan Metode Elemen Hingga’, *Journal Teknik Mesin, Elektro, Informatika, Kelautan dan Sains*, 2(1), pp. 1-8.

Prasetiyo, A.B. *et al.* (2022) ‘Analisis Frekuensi Natural Velg Ring 16 Menggunakan Finite Element Method’, in *Prosiding Rekayasa Teknologi Industri dan Informasi XVII Tahun 2022 (ReTII). Seminar Nasional Rekayasa Teknologi Industri dan Informasi XVII Tahun 2022 (ReTII)*, Yogyakarta, Indonesia: Institut Teknologi Nasional Yogyakarta, pp. 354–359.

Pris, F.R., Suyitno, B.M. and Suhadi, A. (2019) ‘Analisis Kekuatan Velg Aluminium Alloy 17 Inch Dari Berbagai Desain Menggunakan Metode Finite Element Analysis (FEA)’, *Teknobiz : Jurnal Ilmiah Program Studi Magister Teknik Mesin*, 9(2), pp. 33–39.

Rizqi, R.A., Rifky, R. and Ariyansah, R. (2023) ‘Analisis Dinamika Eksplisit Pada Pelek Ring 15 Dengan Metode Simulasi Uji Impak’, *Teknobiz : Jurnal Ilmiah Program Studi Magister Teknik Mesin*, 13(2), pp. 101–106.

Sivaraj, S., Nagendharan, S. and Mohanavel, V. (2020) ‘Experimental Investigation On Wheel Natural Frequency Performance Using Modal Analysis In Free And Loaded Condition’, in *Materials Today: Proceedings*. *International Conference on Nanotechnology: Ideas, Innovation and Industries*, Ramanathapuram, India: Syed Ammal Engineering College (International Conference on Nanotechnology: Ideas, Innovation and Industries), pp. 3234–3242.

Sofyan, D., Gamayel, A. and Zaenudin, M. (2023) ‘Simulasi Uji Impact Pada Desain Velg Berbahan ASTM A299 Diameter 15 Inch Dengan Standard Uji SAE J175’, *Scientific Journal of Mechanical Engineering Kinematika*, 8(1), pp. 85–93.

Somayaji, A. *et al*. (2022) ‘Studies On Modal Analysis Of Aluminium Based Carbon Fibre Reinforced Optimized Alloy Wheel Used In Automotive Sectors’, in *Materials Today: Proceedings*. *International Conference on Smart and Sustainable Developments in Materials, Manufacturing and Energy Engineering*, Karnataka, India: NMAM Institute of Technology (International Conference on Smart and Sustainable Developments in Materials, Manufacturing and Energy Engineering), pp. 274–277.

Stojanovic, B., Bukvic, M. and Epler, I. (2018) ‘Application of Aluminum and Aluminum Alloys in Engineering’, *Applied Engineering Letters*, 3(2), pp. 52–62.

Vijayakumar, R. *et al*. (2020) ‘Investigation on automobile wheel rim aluminium 6061 and 6066 Alloys using ANSYS WORKBENCH’, in *Materials Today: Proceedings*. *International Conference on Nanotechnology: Ideas, Innovation and Industries 2020*, India: Syed Ammal Engineering College (International Conference on Nanotechnology: Ideas, Innovation and Industries), pp. 3155–3159.

Win, H.H. and Oo, M.Z.T. (2019) ‘Modal Analysis Of Aluminum Alloy Five Spokes Wheel’, *IRE Journals*, 2(12), pp. 173–179.

Yulianto, F.A., Ariyansah, R. and Octavianus, G. (2023) ‘Analisis Modifikasi Desain pada Lokomotif CC-201 Tipe GE U18C dengan Metode Simulasi CFD’:, *Jurnal Asiimetrik: Jurnal Ilmiah Rekayasa dan Inovasi*, 5(2), pp. 161–170.