

Finite Element Based Comparative Structural and Modal Analysis of Steel and Alloy Wheel Rims Using Different Materials

Ankita Kulkarni¹, Nitin A. Kharche², Santosh R. Shekokar³, Yugesh A. Kharche⁴,
Ramakant Choudhari⁵, Dipak Kharat⁶

¹*P.G. student, Department of Mechanical Engineering, Padm. Dr. V. B. Kolte College of Engineering, Malkapur, Maharashtra, India*

^{2,3,4,5,6}*Department of Mechanical Engineering, Padm. Dr. V. B. Kolte College of Engineering, Malkapur, Maharashtra, India*

Abstract:

This study evaluates different types of automobile wheels by comparing their properties based on varying materials used in construction. To evaluate how well-suited these wheel designs are for each particular model in use. By meticulously analyzing design aspects, conducting thorough structural assessments, and evaluating comprehensive operational metrics, this endeavor yields crucial information about the properties and functionalities of carbon fiber and aluminum alloy vehicle wheels. Computer-aided design models, combined with finite element analyses (FEAs) and evaluation methods for assessing system performances, help in examining how stresses spread out, strains accumulate, and shapes deform under various conditions. Through recognizing distinctions between steel and alloy rim types along with their respective characteristics, this initiative contributes significantly to improving both functionality and appearance for vehicle models.

Keywords: Wheel rims, Carbon steel, Aluminium Alloy, CAD, FEA.

I. INTRODUCTION

Today's historians and archaeologists believe that the invention of the wheel was the true beginning of any ancient civilization. The most important invention of antiquity is the wheel. From an enlarged bearing, the wheel has evolved into a crucial component of all contemporary vehicles. To guarantee passenger safety, contemporary motor vehicles are manufactured in accordance with extremely stringent regulations. The materials used to make these wheels have advanced in sophistication, and they can be made of steel or nonferrous alloys like aluminium and magnesium. Over the years, automobile wheels have changed from their first spoke designs made of steel and wood. The rims of contemporary cars are made of cast and forged aluminium alloys and are stamped metal. Numerous cutting-edge techniques for testing that are compatible with experimental stress measurement have been developed since the 1970s.

Choosing automobile wheels involves an important choice because these components greatly affect how well the car performs, its level of security on roads, and overall appearance. Several

widely chosen alternatives include steel wheels and aluminum ones. Despite being historically favored due to their strength and affordability, modern alloys provide benefits such as lighter weights, better fuel economy, and more attractive designs compared to traditional steel wheels. The endeavor compares steel rim designs against those of aluminum alloys in this study. An in-depth scrutiny is conducted regarding their architectural decisions, static strength evaluation, dynamic pressure assessment, modal frequency investigation, and harmonics reaction measurement techniques. A primary emphasis lies within examining both steel and aluminum rims employed by automobile models. The CAD program Creo Parametric facilitates precise three-dimensional representations of prototype steel and aluminum wheels. The Finite Element Analysis technique is utilized through ANSYS Workbench for simulating how rim structures behave in response to different loads.

Furthermore, measurements on sound waves and movements within each wheel segment help understand how they react under stress and identify any natural tones where vibrations might occur frequently. Additionally, an investigation into the durability of both rim designs under repeated load conditions, taking into account their materials' characteristics and how stresses are distributed among them, shall take place. This study predicts the anticipated lifespan of the tires based on their performance during normal usage scenarios.

II. MATERIAL SELECTION

2.1 Composition of Material

Wheel Rim Type	Steel Wheel Rim	Alloy Wheel Rim
Material	Carbon Steel	Aluminum Alloy
Density	7.85 g/cm ³	2.8 g/cm ³
Young's Modulus	200 GPa	73.1 GPa
Poisson's Ratio	0.266	0.33
Yield Stress	250 MPa	505 MPa

Sr. No.	Element / Material	Typical Composition (%)
1	Aluminum	90 – 95 %
2	Magnesium	2 – 5 %
3	Silicon	1 – 3 %
4	Other Metals (Copper, Nickel, Zinc)	1 – 2 %

Sr. No.	Element / Material	Typical Composition (%)
1	Iron	97 – 99 %
2	Carbon	0.2 – 0.5 %
3	Manganese	0.5 – 1.2 %

4	Other Metals (Chromium, Nickel, Copper)	0.2 – 0.5 %
---	--	-------------

Alloy mixtures profoundly affect the characteristics and efficiency of automobile wheels. Wheel rims made of alloys like aluminum, magnesium, and silicon offer reduced weight, superior protection against rusting, and improved cooling efficiency. Adding additional metallic elements enhances qualities such as toughness and longevity.

Steel wheel rims are primarily made of iron along with carbon, manganese, and other metals. These materials provide the rims with strong tensile strength, good impact resistance, and long-lasting performance. The carbon in the steel contributes to its hardness and strength. However, the exact composition of metals can vary between manufacturers, as they may use their own proprietary formulas.

2.2 Comparison of Wheel Rims

2.2.1. Steel Wheel Rims

Steel wheel rims are traditionally favored for their robust construction and cost-effectiveness. The primary advantages of steel rims are their extreme durability and strength, which make them the ideal choice for heavy-duty vehicles and driving in rough or off-road conditions. Because they are manufactured through a less complex stamping process, they are significantly cheaper than their alloy counterparts, offering an economical solution for budget-conscious users.

However, these benefits come with notable trade-offs. The high weight of steel increases the vehicle's unsprung mass, which can negatively impact handling and slightly reduce fuel efficiency. Additionally, steel is highly susceptible to rust and corrosion, especially when exposed to humidity or road salt, necessitating protective coatings and regular maintenance to prevent structural degradation. In terms of performance metrics, steel rims boast high tensile strength to support heavy loads and excellent impact resistance; unlike alloys, steel tends to bend rather than crack under pressure. Their high modulus of elasticity ensures a rigid and stable wheel structure.

2.2.2. Alloy Wheel Rims

Alloy wheel rims, typically composed of aluminum or magnesium, are designed with a focus on performance and aesthetics. The most significant advantage is their low weight, which improves vehicle acceleration, braking response, and fuel economy. They are also highly valued for their attractive appearance, as the casting process allows for intricate, modern designs. Furthermore, alloys offer better resistance to corrosion, maintaining their finish longer than steel with less maintenance.

The disadvantages of alloy rims are primarily centered on cost and fragility. They are more expensive due to the higher cost of raw materials and complex manufacturing processes like forging or casting. Moreover, they are more easily damaged by potholes or harsh impacts, as the material is more brittle than steel and prone to cracking under extreme stress. From a performance standpoint, the reduced weight of alloy rims decreases the vehicle's unsprung

weight, allowing the suspension to react more quickly to the road. They also provide improved heat dissipation, which helps conduct heat away from the brakes, reducing the risk of brake fade during intense or continuous use.

III. DESIGN AND MODELING

3.1 CREO

Creo is a group of Computer-Aided Design (CAD) apps used for creating products, and it's made by PTC. The software has different apps that each help people with specific jobs in product development. Creo works on Windows and includes tools for 3D CAD modeling, both parametric and direct, 2D drawings, simulation and analysis, schematic design, technical drawings, and viewing models. It can also work with Mastercam, a software for machining, to quickly turn designs into physical parts.

3.2 ANSYS

For conducting the analysis of the model created in CREO PARAMETRIC, we employed the finite element solver ANSYS 19.1. ANSYS is a general-purpose finite element analysis (FEA) software package. Finite Element Analysis is a numerical method that breaks down a complex system into small pieces, referred to as elements, of user-defined size. The software applies equations that describe the behavior of these elements and solves them to provide a comprehensive understanding of how the entire system behaves. These results can be displayed in either tabular or graphical formats. This type of analysis is commonly used for the design and optimization of systems that are too complex to analyze manually. Systems that fall into this category are complex due to their geometry, scale, or governing equations. ANSYS is the standard FEA teaching tool in mechanical engineering.

3.3 Geometry of Wheels

3.3.1 Steel

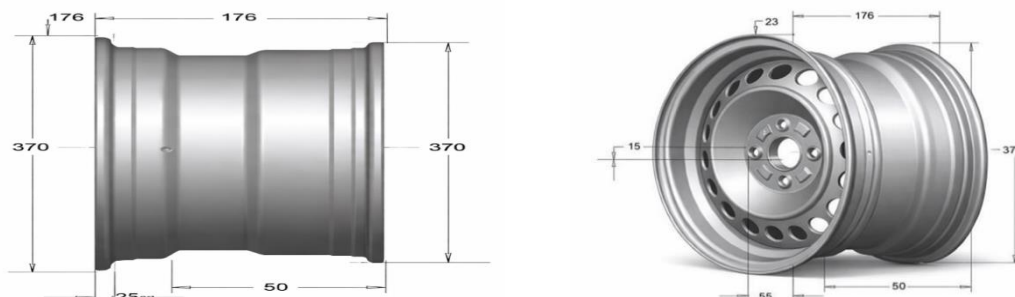


Fig. No. 1- steel wheel geometry

The figure shows the geometry of a steel wheel in both side view and 3D view, highlighting its important dimensions and structural features. The overall diameter of the wheel is approximately 370 mm, which determines its compatibility with the tyre size. The rim width, indicated as 176 mm on both sides, provides proper support and seating for the tyre. The outer edges of the wheel form flanges that help in holding the tyre securely, while the inner portion acts as the bead seat where the tyre fits tightly. The central part of the wheel consists of the hub

mounting area with bolt holes used to attach the wheel to the vehicle axle, ensuring proper load distribution. Additionally, multiple circular holes are provided around the disc for ventilation, which helps in reducing weight and dissipating heat generated during braking. The step dimensions such as 25 mm and 50 mm represent variations in thickness and profile, contributing to the strength and durability of the wheel. Overall, the geometry of the steel wheel is designed to achieve a balance between strength, stability, heat dissipation, and efficient performance.

3.3.1.1 Material



Fig. No. 2- Carbon steel

The figure represents the material model of a steel wheel rim made of carbon steel in ANSYS software. It shows the 3D geometry of the wheel after assigning the material properties required for analysis. Carbon steel is selected due to its high strength, durability, and good load-bearing capacity, making it suitable for automotive wheel applications. The model includes key features such as the central hub, bolt holes, and ventilation holes, which are essential for proper mounting, weight reduction, and heat dissipation. In this stage, material properties like density, Young's modulus, and Poisson's ratio are defined in the software to simulate real-world behavior under loading conditions. This material assignment is a crucial step before performing further analysis such as meshing, deformation, and stress evaluation.

3.3.1.1.2 Meshing

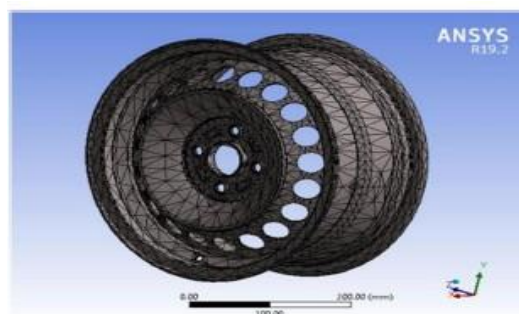


Fig. No. 3- Meshing in steel rim

The figure represents the meshing of the steel wheel rim model carried out in ANSYS for finite element analysis. In this stage, the complete geometry of the wheel is divided into a large number of small elements connected by nodes, forming a mesh structure. This discretization is essential for converting the physical model into a numerical model

so that complex engineering problems such as stress, deformation, and strain can be solved accurately. The mesh shown consists of fine triangular/tetrahedral elements, which are used to capture the detailed geometry of the rim, including holes, edges, and curved surfaces. A finer mesh generally provides more accurate results, especially in critical regions like the hub and bolt holes, but it also increases computational time. Proper meshing ensures that the analysis results are reliable and helps in identifying stress concentration zones and structural behavior under loading conditions.

3.3.2 Alloy



Fig. No. 4- Alloy wheel geometry

The figure represents the geometry of an alloy wheel showing both the front view and side profile with detailed dimensions. The front view highlights the spoke design, which consists of multiple evenly spaced spokes connecting the outer rim to the central hub, providing both strength and aesthetic appeal. The central hub contains bolt holes used for mounting the wheel to the vehicle. Various radii such as R15, R85, and others indicate the curvature of spokes and edges, which are designed to improve stress distribution and reduce weight. The side view shows the rim width and diameter, with dimensions such as approximately 420.98 mm outer diameter and 179.04 mm width, which determine tyre fitting and compatibility. The inner profile ensures proper seating of the tyre and structural stability. Alloy wheels are typically made from lightweight materials like aluminum alloys, which offer advantages such as reduced weight, better heat dissipation, improved performance, and enhanced appearance compared to steel wheels. Overall, the geometry is optimized to achieve a balance between strength, weight reduction, and visual design.

3.3.2.1 Material

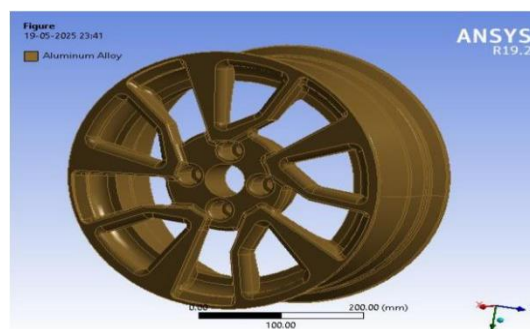


Fig. No. 5- Aluminum Alloy

Figure 5 shows the three-dimensional model of a wheel rim made of aluminum alloy, created and analyzed using ANSYS 19.2 software. The model represents the assignment of a uniform aluminum alloy material to the entire geometry, which is essential for performing accurate simulations such as stress and thermal analysis. Aluminum alloy is selected due to its lightweight nature, high strength-to-weight ratio, good thermal conductivity, and corrosion resistance, making it suitable for automotive applications. The figure also indicates the dimensional scale and coordinate orientation of the model, which are important for defining boundary conditions and analyzing performance in subsequent simulation studies.

3.3.1.1.2 Meshing

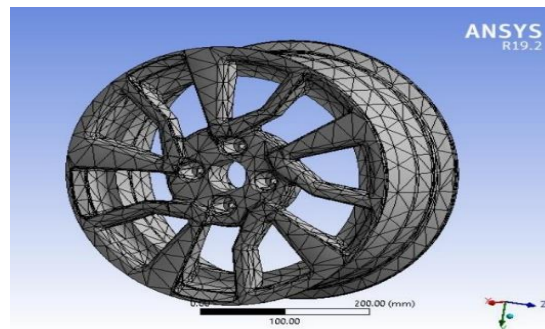


Fig No. 6- Meshing in Aluminium wheel rim

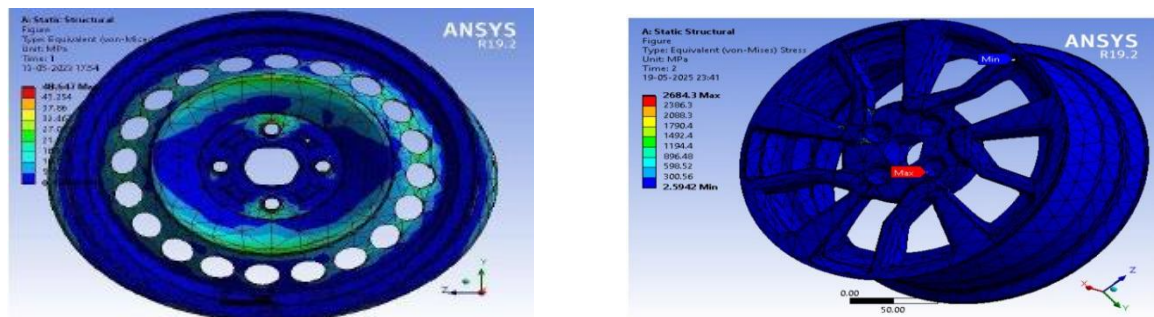


Fig. No. 7 Equivalent stress (Von-mises stress)

Figure 6 shows the meshing of the aluminum wheel rim model performed in ANSYS 19.2. Meshing is the process of dividing the complex geometry into smaller finite elements, which enables accurate numerical analysis during simulation. In this model, the wheel rim is discretized into a network of triangular elements that closely follow its curved and intricate shape. A finer mesh is observed in regions with complex geometry, such as the spokes and hub area, to improve the accuracy of stress and deformation results. Proper meshing ensures reliable simulation outcomes by balancing computational efficiency with solution accuracy, making it a crucial step in finite element analysis.

IV. ANALYSIS OF MATERIAL

4.1 Static structural Analysis

Figure 7 shows the results of the static structural analysis of the aluminum wheel rim in terms of equivalent (Von-Mises) stress obtained from ANSYS 19.2. The color contour represents the distribution of stress across the wheel rim, where red indicates the maximum stress region and blue indicates the minimum stress region. It can be observed that higher stress concentrations occur around the hub and spoke junction areas due to load transfer and geometric discontinuities, while the outer rim experiences relatively lower stress. The Von-Mises stress criterion is used to predict yielding of the material under complex loading conditions, and these results help in identifying critical regions that may require design modification to ensure structural safety and performance.

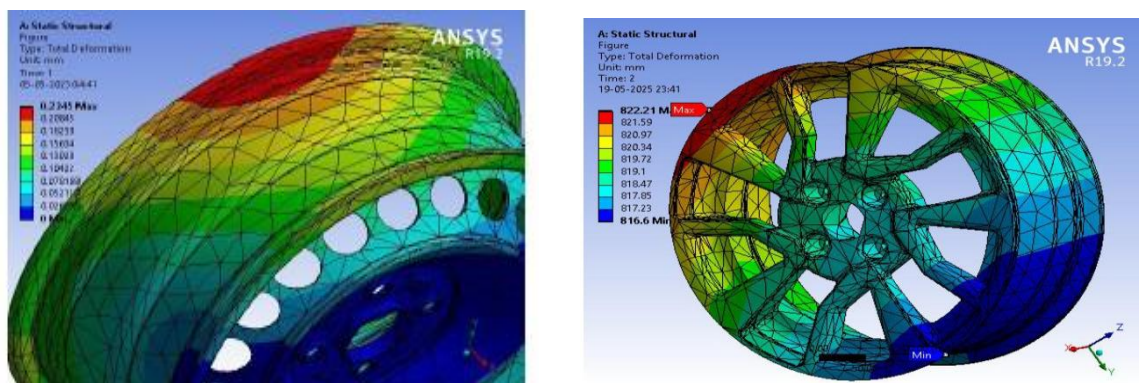


Fig. No. 8- Total deformation

Figure 8 illustrates the total deformation of the aluminum wheel rim obtained from static structural analysis in ANSYS 19.2. The contour plot represents the displacement distribution across the wheel, where red indicates the maximum deformation and blue indicates the minimum deformation. It can be observed that the highest deformation occurs along the outer rim region due to applied loads and boundary conditions, while the hub area shows minimal deformation as it is more constrained. The gradual variation in color from blue to red indicates how the deformation is distributed over the structure. This analysis helps in understanding the stiffness and structural behavior of the wheel rim, ensuring that the deformation remains within acceptable limits for safe and reliable performance.

4.2 Modal Analysis

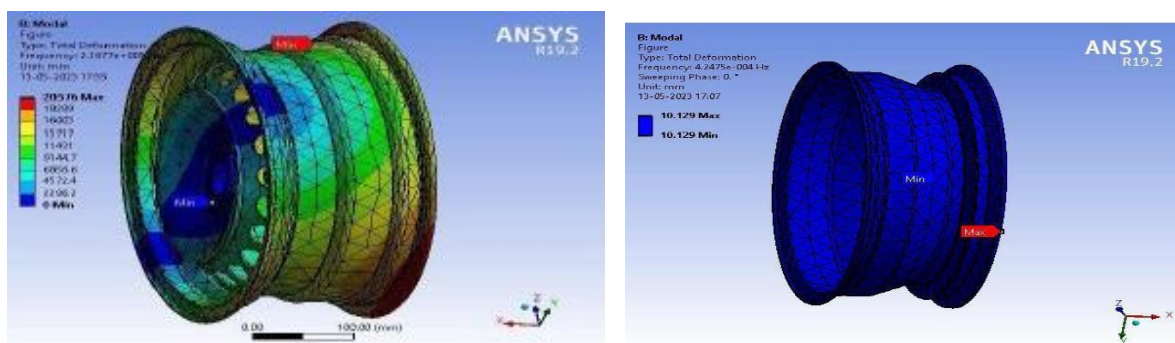


Fig. No. 9- Total deformation

Figure 9 presents the results of the modal analysis of the aluminum wheel rim in ANSYS 19.2, showing the total deformation corresponding to different natural vibration modes. Modal analysis is used to determine the natural frequencies and mode shapes of the structure, which are critical for understanding its dynamic behavior. The contour plots indicate the deformation pattern of the wheel rim at specific frequencies, where red regions represent maximum displacement and blue regions indicate minimum displacement. These mode shapes illustrate how the wheel rim vibrates under dynamic conditions, with certain areas such as the outer rim and spokes experiencing higher deformation. This analysis helps in identifying potential resonance conditions and ensures that the operating frequencies do not coincide with the natural frequencies, thereby preventing excessive vibrations and structural failure.

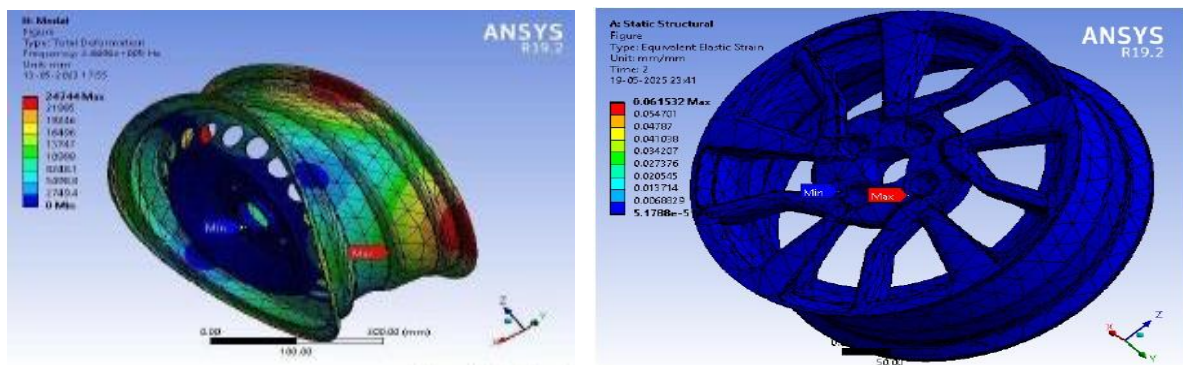


Fig. No- 10 Total deformation

The provided image illustrates two different types of Finite Element Analysis (FEA) conducted on automotive wheel designs using ANSYS R19.2 to evaluate their structural integrity and dynamic behavior. The left figure shows a Modal Analysis of a steel-style wheel, which identifies the natural frequency (approximately 2,604.8 Hz) and the resulting mode shape; the red areas on the outer rim indicate the regions of highest relative displacement during vibration, which is essential for minimizing noise and resonance. In contrast, the right figure displays a Static Structural Analysis of an alloy spoke wheel, specifically measuring Equivalent Elastic Strain. This simulation shows how the material deforms under a constant physical load, with the "Max" indicator pointing to the central hub area near the bolt holes, highlighting this as a critical stress concentration point where the wheel is most vulnerable to fatigue or failure. Together, these simulations allow engineers to optimize the wheel's geometry and material distribution to balance weight with safety and durability.

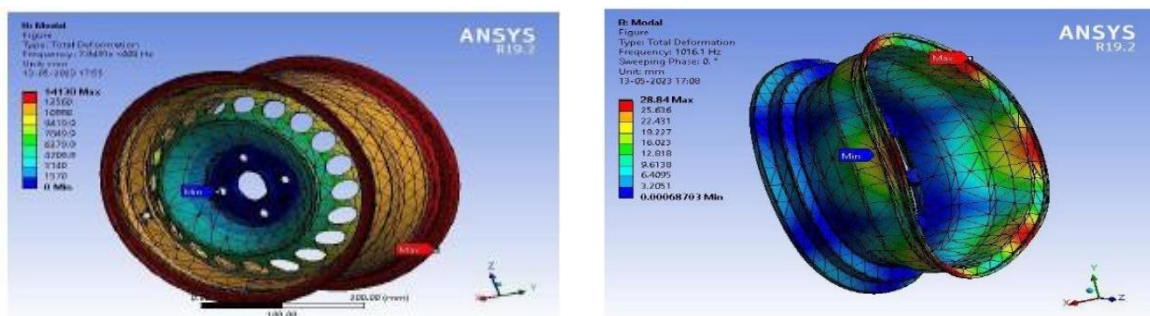


Fig. No. 11- Total deformation

The provided image displays two Modal Analysis results for a steel automotive wheel, showcasing its vibrational characteristics at different natural frequencies. In the left figure, the wheel is vibrating at a frequency of 734.9 Hz, resulting in a mode shape where the entire outer rim experiences high relative displacement (red zones), with a maximum total deformation value of 14,130 mm. The right figure illustrates a second mode at a higher frequency of 1016.1 Hz, which presents a more complex harmonic pattern; here, the deformation is concentrated in specific "lobes" or waves along the rim flange, with a much lower relative deformation scale of 28.84 mm. By analyzing these patterns, engineers can identify the wheel's resonant frequencies to ensure they do not align with vehicle operating speeds or engine vibrations, which would otherwise cause excessive noise, harshness, or structural fatigue.

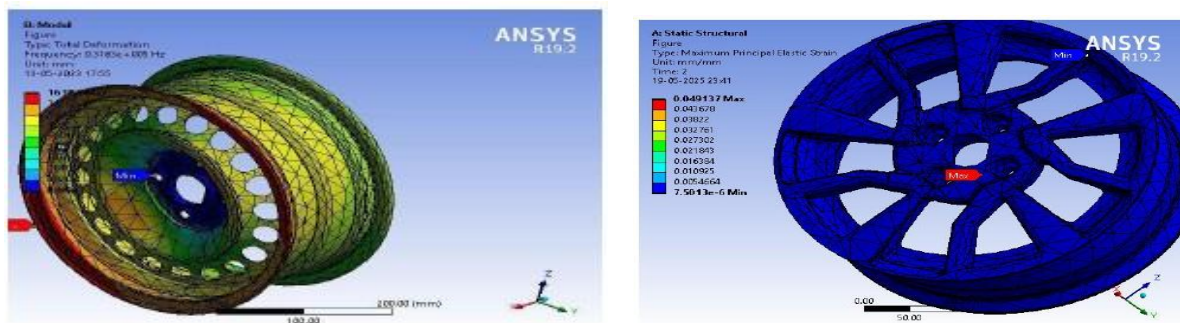


Fig. No. 12- Total deformation

The provided image highlights two critical engineering simulations conducted on automotive wheels using ANSYS R19.2 to assess their performance under different physical conditions. The left figure depicts a Modal Analysis of a steel-style wheel, illustrating its vibration characteristics at a natural frequency of 931.62 Hz; the color map shows "Total Deformation," with red areas indicating where the wheel's rim would flex the most if excited at this specific frequency. Conversely, the right figure shows a Static Structural Analysis of an alloy wheel, focusing on Maximum Principal Elastic Strain rather than total movement. This analysis identifies where the material is stretched or compressed most significantly under a specific load, with the "Max" probe positioned directly on the central hub near the bolt holes. These dual simulations are vital for automotive designers to ensure that the wheel's shape and material can withstand both the constant weight of the vehicle and the dynamic vibrations experienced during driving.

4.3 Harmonic Response

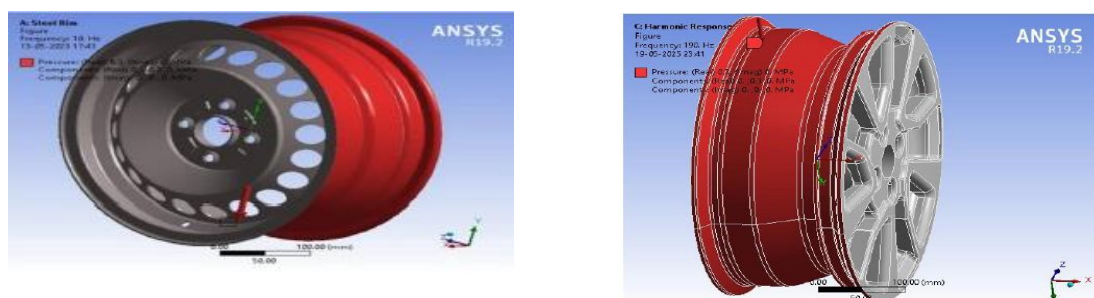


Fig. No. 13- Harmonic Response

This image illustrates the Harmonic Response analysis for two different wheel types, a steel rim (left) and an alloy spoke wheel (right), using ANSYS R19.2. Unlike a simple modal analysis that only finds natural frequencies, a harmonic response simulation predicts how the structure will actually vibrate when subjected to a continuous, cyclic load (like the spinning of a tire or engine vibrations) over a range of frequencies.

Analysis Details

Steel Rim (Left): The simulation is set at a low frequency of 10 Hz. The red arrow indicates the application of a periodic Pressure load on the rim's bead seat. At this frequency, the displacement is relatively controlled, and the analysis helps determine if the steel structure can dampen or sustain these low-speed oscillations.

Alloy Wheel (Right): This simulation is conducted at a much higher frequency of 190 Hz. The red highlighted area shows the pressure distribution across the entire outer circumference of the rim. Because alloy wheels are generally stiffer than steel, this test is crucial for ensuring that high-frequency road noise or mechanical inputs do not lead to structural resonance or fatigue failure.

V. RESULTS

5.1 Alloy wheel Static Structural results

Results						
Type	Total Deformation	Equivalent (von-Mises) Stress	Maximum Principal Stress	Equivalent Elastic Strain	Maximum Principal Elastic Strain	Shear Stress
Minimum	816.6 mm	2.5942 MPa	-2169.6 MPa	5.1788e-005 mm/mm	7.5013e-006 mm/mm	-155.34 MPa
Maximum	822.21 mm	2684.3 MPa	5061.7 MPa	6.1532e-002 mm/mm	4.9137e-002 mm/mm	159.23 MPa
Average	818.98 mm	90.586 MPa	82.011 MPa	1.4457e-003 mm/mm	1.1895e-003 mm/mm	-9.8214e-002 MPa
Minimum Value Over Time						
Type	Total Deformation	Equivalent (von-Mises) Stress	Maximum Principal Stress	Equivalent Elastic Strain	Maximum Principal Elastic Strain	Shear Stress
Minimum	0.26737 mm	4.1186e-002 MPa	-2169.6 MPa	6.738e-007 mm/mm	-1.185e-006 mm/mm	-155.34 MPa
Maximum	816.6 mm	2.5942 MPa	-42.105 MPa	5.1788e-005 mm/mm	7.5013e-006 mm/mm	-4.0706 MPa

Maximum Value Over Time						
Type	Total Deformation	Equivalent (von-Mises) Stress	Maximum Principal Stress	Equivalent Elastic Strain	Maximum Principal Elastic Strain	Shear Stress
Minimum	2.799 mm	55.984 MPa	94.604 MPa	1.1655e-003 mm/mm	9.667e-004 mm/mm	7.9015 MPa
Maximum	822.21 mm	2684.3 MPa	5061.7 MPa	6.1532e-002 mm/mm	4.9137e-002 mm/mm	159.23 MPa

The provided table summarizes the Static Structural Analysis results for an alloy wheel, detailing the mechanical response of the structure under a specific loading condition across three key categories: general results, minimum values, and maximum values over time. The data tracks critical engineering metrics such as Total Deformation, which peaks at 822.21 mm, and Equivalent (von-Mises) Stress, which reaches a maximum of 2,684.3 MPa, a value used by designers to determine if the material will undergo permanent plastic deformation or eventual failure. Additionally, the table records Maximum Principal Stress (5,061.7 MPa) and Equivalent Elastic Strain (0.061532 mm/mm), providing a comprehensive overview of how the wheel's geometry distributes tension and compression. By comparing the average values (e.g., 90.586 MPa for von-Mises stress) against the maximums, engineers can identify high-stress concentration points, typically located at the lug holes or spoke bases, to optimize the wheel for both safety and weight reduction.

5.2 Alloy Wheel Modal Analysis Result

Object Number	1	2	3	4	5	6	7	8	9	10
---------------	---	---	---	---	---	---	---	---	---	----

Results										
Object Number	1	2	3	4	5	6	7	8	9	10
Minimum	10.129 mm	1.4017 mm	7.932e-003 mm	1.5311e-003 mm	1.3784e-002 mm	4.0214e-003 mm	3.9962e-003 mm	6.8703e-004 mm	0.2257 mm	7.5728e-002 mm
Maximum	10.129 mm	13.299 mm	15.264 mm	23.474 mm	15.299 mm	23.777 mm	29.573 mm	28.84 mm	20.078 mm	16.793 mm
Average	10.129 mm	9.4634 mm	5.9321 mm	5.4006 mm	5.8811 mm	5.4133 mm	4.8129 mm	4.8193 mm	10.044 mm	9.2254 mm

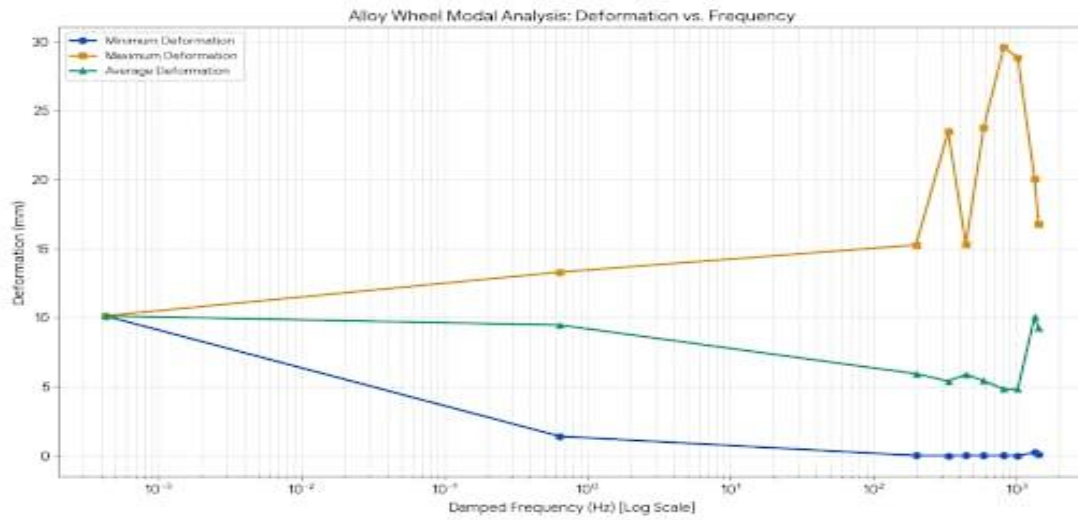
Information										
Object Number	1	2	3	4	5	6	7	8	9	10
Damped Frequency	4.2475e-004 Hz	0.64156 Hz	197.55 Hz	331.7 Hz	440.83 Hz	588.1 Hz	810.99 Hz	1016.1 Hz	1331.5 Hz	1419.4 Hz

The provided table details the Modal Analysis results for an alloy wheel across ten distinct vibration modes, focusing on the Damped Frequency and the resulting Total Deformation for each state. The analysis identifies key resonant frequencies ranging from near-zero (rigid body modes) to higher harmonics like 1419.4 Hz, providing engineers with a "vibration signature" of the wheel to ensure it does not resonate with engine or road inputs. For each mode number, the table records minimum, maximum, and average deformation values (ranging from roughly 10 mm to nearly 30 mm), which represent the relative scaled movement of the wheel's geometry during oscillation rather than physical displacement under a static load. By examining these results, designers can pinpoint which frequencies cause the most significant flex in the wheel structure—such as mode 7 at 810.99 Hz with a maximum deformation of 29.573 mm—and adjust the material distribution or spoke geometry to shift these frequencies outside of the vehicle's typical operating range for improved noise, vibration, and harshness (NVH) performance.

5.3 Alloy Wheel Modal analysis: Deformation Vs Frequency

This graph visualizes the relationship between the Damped Frequency (on a logarithmic scale) and the resulting Deformation for an alloy wheel, effectively plotting the numerical data from the modal analysis table you provided earlier. The plot tracks three key trends—Maximum (orange), Average (green), and Minimum (blue) deformation—across various vibrational modes. At low frequencies (below 1 Hz), the lines are relatively flat and close together, representing rigid body modes where the wheel moves as a single unit without significant internal flexing. However, as the frequency increases past 100 Hz, the "Maximum Deformation" peaks sharply at specific resonant frequencies, such as the major spike near 800–1000 Hz where deformation hits nearly 30 mm. These peaks are critical "danger zones" for engineers; if the car's engine or road surface creates vibrations at these exact frequencies, the wheel could resonate, leading to extreme noise or even structural fatigue. By studying this curve, designers can ensure the wheel's "natural signature" is tuned to avoid common operating vibrations.

Would you like me to explain why the frequency axis is plotted on a logarithmic scale instead of a standard linear one?



5.4 Steel Rim Static Structural Analysis Results

The provided table summarizes the Static Structural Analysis results for a steel rim, outlining how the material responds to a specific mechanical load. Key metrics such as Total Deformation show a maximum displacement of only 0.2345 mm, indicating that the steel rim is quite rigid under the tested conditions. The Equivalent (von-Mises) Stress reaches a maximum of 48.647 MPa, which is well below the typical yield strength of automotive structural steel (usually around 250–350 MPa), suggesting a high factor of safety and no permanent deformation. Additionally, the Maximum Principal Stress of 66.728 MPa and the very low Equivalent Elastic Strain (max 0.00033365 mm/mm) further confirm that the rim is operating comfortably within its elastic limit.

Results					
Type	Total Deformation	Equivalent (von-Mises) Stress	Maximum Principal Stress	Equivalent Elastic Strain	Maximum Principal Elastic Strain
Minimum	0. mm	0.10549 MPa	-21.641 MPa	2.2229e-006 mm/mm	-6.6948e-006 mm/mm
Maximum	0.2345 mm	48.647 MPa	66.728 MPa	3.3365e-004 mm/mm	2.8385e-004 mm/mm
Average	6.5055e-002 mm	5.4395 MPa	2.9567 MPa	3.4552e-005 mm/mm	1.8827e-005 mm/mm

5.5 Steel Rim Modal Analysis Results

Total Deformation										
Object Number	1	2	3	4	5	6	7	8	9	10
Results										
Object Number	1	2	3	4	5	6	7	8	9	10
Minimum	0. mm									
Maximum	20576 mm	20621 mm	24744 mm	24891 mm	12856 mm	25706 mm	25670 mm	1413 mm	16474 mm	16383 mm
Average	8183.2 mm	8338.3 mm	7080.6 mm	7458.2 mm	9311.4 mm	6812.6 mm	6724.3 mm	8861.2 mm	9557.2 mm	9527.6 mm

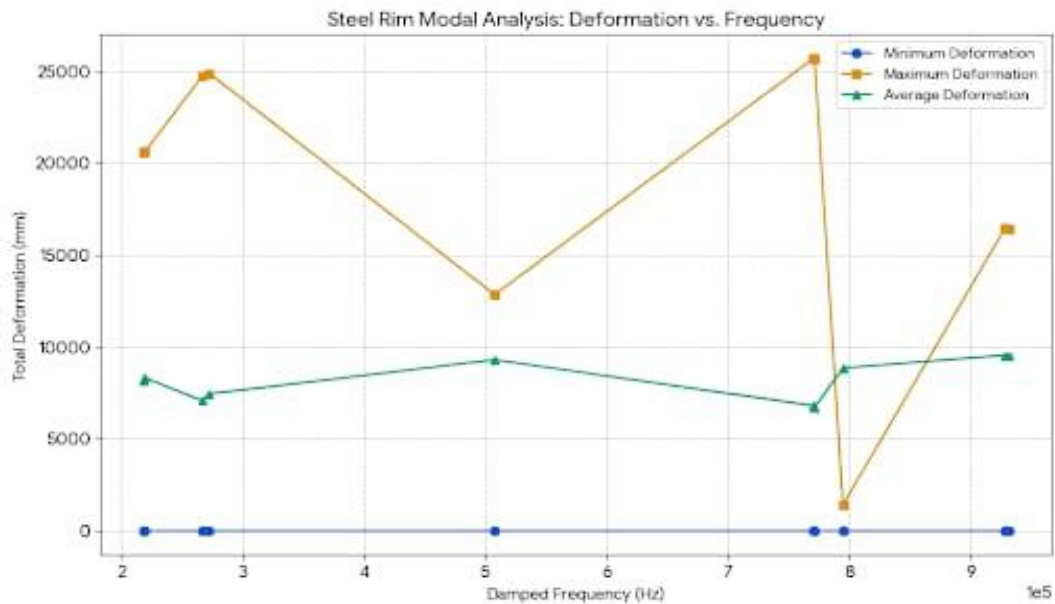
Information										
Object Number	1	2	3	4	5	6	7	8	9	10
Damped Frequency	2.1877e+005 Hz	2.1937e+005 Hz	2.6696e+005 Hz	2.7252e+005 Hz	5.0782e+005 Hz	7.7079e+005 Hz	7.7164e+005 Hz	7.9491e+005 Hz	9.287e+005 Hz	9.3163e+005 Hz

The provided table summarizes the Modal Analysis results for a steel rim across ten distinct vibrational modes, documenting the Damped Frequency and the resulting Total Deformation for each state. A notable characteristic of this specific analysis is the exceptionally high frequency range, with the first mode starting at approximately 218,770 Hz ($2.1877 \times 10^5 \text{ Hz}$) and the tenth mode reaching 931,630 Hz, indicating a very stiff structural response. The "Total Deformation" values, such as the maximum of 25,706 mm in Mode 6, represent relative scaled displacement used to visualize the "mode shape" rather than physical movement under a real-world load. By observing how these frequencies are often grouped in pairs (such as Modes 1 and 2 or 6 and 7), engineers can confirm the structural symmetry of the rim design, ensuring that these high-frequency resonant points do not interfere with vehicle electronics or lead to ultrasonic acoustic issues during operation.

5.6 Steel Rim Modal Analysis: Deformation Vs Frequency

This graph visualizes the relationship between the Damped Frequency and Total Deformation for a steel rim, mapping the specific vibrational modes identified in the previous modal analysis table. The orange line tracks the Maximum Deformation, which fluctuates significantly as the frequency increases, showing major peaks near $2.5 \times 10^5 \text{ Hz}$ and $7.8 \times 10^5 \text{ Hz}$

where the rim experiences its highest relative flexure. The green line represents the Average Deformation, which remains relatively stable between 5,000 mm and 10,000 mm, while the blue line at the bottom indicates that the Minimum Deformation is consistently near zero, signifying that certain nodes or points on the rim remain stationary during these vibrations. In an engineering context, the erratic "sawtooth" pattern of the orange line highlights specific resonant zones that designers must monitor to ensure the rim does not suffer from high-frequency fatigue or produce unwanted acoustic resonance during high-speed operation.



VI. CONCLUSIONS

A detailed parametric model was created using Creo for both carbon steel and aluminum alloy wheel rims. Then, several analyses were done in ANSYS. A force of 625N was applied to check the stress and strain levels.

- 1) The amount of deformation varied for each wheel rim depending on the frequency. The alloy wheel rim showed better stability than the steel one.
- 2) To check how the structures behave under vibrations, a vibrational analysis was done. The harmonic response analysis showed a maximum amplitude of 1.6893, which means the model is stable.
- 3) These results show that the aluminum wheel rim performs better than the steel one in terms of heat dissipation, braking, and how long it can last before failing.
- 4) The results suggest that the aluminum alloy wheel rim is better for the intended use. It has better stability when there are vibrations, it cools better, it stops more effectively, and it lasts longer. These factors make it better than the carbon steel rim.

More research could be done to create better aluminum alloys with improved properties. This might involve trying new mixtures of materials or adding stronger elements to make the material perform even better.

REFERENCES

- [1] Htay Htay win, May zon than Modal Analysis of Aluminium Alloy Five Spokes Wheel's Rim, iconic research and engineering journals, Volume 2 Issue 12, JUN 2019.
- [2] Ameya Malwadkar, Pradnya Mungi, International Research Journal of Engineering and Technology, Meshing and Analysis of Vehicle Wheel Rim, Volume: 09 Issue: 11 | Nov 2022.
- [3] P. Meghashyam, S. Girivardhan Naidu and N. Sayed Baba, Design and Analysis of Wheel Rim using CATIA& ANSYS, International Journal of Application or Innovation in Engineering & Management (IJAIEM), Volume 2, Issue 8, August 2013.
- [4] L. Natrayan, P. Dinesh Kumar, P. Santhakumar, R. Mohan raj, R. Mohandass, Design and Comparative Analysis of Old & New Model Car Wheel Rims with Various Materials, Journal for Research| Volume 02| Issue 02 | April 2016.
- [5] Sudhakar Mishra, Dr. L.P. Singh, Structural and Material Analysis of an Automobile Wheel Rim using ANSYS, International Research Journal of Engineering and Technology, Volume 06 Issue 1, Dec 2019.
- [6] Dhiraj Bhargav. A. S, A. S. Jayaram, Chandan. R, International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056 Volume: 05 Issue: 10 | Oct 2018.
- [7] Mr. Sunil Prashanth Kumar, Muhammad Mansoor P, Vishak Vignesh, Abhijeet Kumar, Mordi Sangma, Design And Analysis of Automobile Wheel Rim International Research Journal Of Engineering And Technology (Irjet) E-Issn: 2395-0056 Volume: 08 Issue: 07 | July 2021.
- [8] Jaslok Pandey, Jayesh Dange, Fatigue Analysis of Wheel Rim Using Different Alloy to Eliminate the Manufacturing Difficulties with Reduced Weight, International Journal of Scientific & Engineering Research Volume 9, Issue 5, May-2018.
- [9] X Jiang, R Lyu, Y Fukushima, M Otake² and D Y Ju, Lightweight design and analysis of automobile wheel based on bending and radial loads, International Conference on Material Strength and Applied Mechanics IOP Publishing Materials Science and Engineering 372 (2018).
- [10] Mr. Chintapalli Shekhar et al. ,International Journal of Advance Scientific Research and Engineering Trends, Volume 5, Issue 7, July 2020.
- [11] B. Lalunayak, K. Musalaiah "Design And Static Modal Analysis Of Car Alloy Wheel Using Finite Element Method: A Review, International Journal of Computational Mathematical IDEAS Publications ISSN 0974-8652, Volume 21, Issue1, Dec 2020.
- [12] Priyam Deka, Faramarz Bagherzadeh, Sundeep Murugesan, Radial Fatigue Analysis of Automotive Wheel Rim(ISO 3006), Asian Journal of Convergence in Technology.

- [13] Kalpesh R. Salunkhe, Prof. Shailesh S Pimpale, Design FEM analysis of alloy wheel Rim of four wheeler, International Advanced Research Journal in Science, Engineering & Technology, Volume 4, Issue 9, Sep 2017.
- [14] Sasankshekhar Panda et al. - International Journal of Engineering Science & Research Technology- 2016.
- [15] Sourav Das ,Design and Weight Optimization of Aluminium Alloy Wheel, International Journal of Scientific and Research Publications, Volume 4, Issue 6, June 2014.
- [16] Ch. Veera Venkata Krishna reddy, V.V. Ramakrishna, Development and Analysis of Wheel Rim by using 3 Different material Al-2024,Al6061 and ZnAl4, International Journal for Research in Science and Technology, volume 11, page 33-39, 2021 march.
- [17] Prasad Satish Divekar, Dr. Thippeswamy Ekbote, Design and Analysis of alloy wheel using Various Mixtures of Alloy, IJSDR, ISSUE10, Volume 4,Oct 2019.
- [18] An analysis of stress and displacement distribution in a rotating rim subjected to pressure and radial loads, P.C. Lam and Srivastav
- [19] Stress Analysis of Wheel Rim International Journal of Mechanical Engineering and Research Volume 1 Issue 1, ISSN: 2277-8128.
- [20] Design and Analysis of Alloy Wheel, journal of V.R. Siddhartha Engineering College, Volume 4 Issue 6, ISSN:2395-0056 and ISSN:2395-0072.
- [21] Design and Comparative Analysis of old & New model car wheel Rims with Various materials, by Natrayan, IETE Journal of Research