ISSN 2319-5991 www.ijerst.com

Peer Reviewed, Referred & Indexed Journal

Research Paper

DESIGN, MODELING, AND STRUCTURAL ANALYSIS OF AN AUTOMOTIVE ALLOY WHEEL RIM USING ANSYS

[1]P. RAVI ,[2]Mr.K.SRINIVASA RAO,[3]B. NARESH,[4]R. KISHORE,[5]G. CHANDRIKA,

[1,3,4,5]Students, Department of Mechanical Engineering, Siddhartha Institute Of Technology & Sciences, Narapally, Telangana-501301, 24tq5a0305@siddhartha.co.in, 24tq5a0302@siddhartha.co.in, 24tq5a0329@siddhartha.co.in, 24tq5a0324@siddhartha.co.in

[2]Assistant Professor, Department of Mechanical Engineering, Siddhartha Institute Of Technology & Sciences, Narapally, Telangana-501301, sreenivas@siddhartha.org.in

ABSTRACT

The wheel rim is one of the most critical components of an automobile, as it supports the vehicle's weight, withstands dynamic loads, and ensures safe and efficient transmission of forces between the tire and the road surface. The increasing demand for lightweight, durable, and high-performance automotive components has encouraged the use of advanced Computer-Aided Design (CAD) and Computer-Aided Engineering (CAE) tools in wheel rim development. This project presents the Design, Modeling, and Structural Analysis of an Automotive Alloy Wheel Rim Using ANSYS by developing a three-dimensional CAD model of an automotive alloy wheel rim and evaluating its structural performance under static loading conditions. The wheel rim geometry is created using CAD software and imported into ANSYS Workbench for finite element analysis (FEA). Appropriate material properties of aluminium alloy are assigned, and suitable boundary conditions and loading conditions are applied to simulate actual operating environments. The analysis focuses on evaluating key performance parameters such as equivalent (von Mises) stress, total deformation, equivalent elastic strain, directional deformation, factor of safety, and structural stability. The simulation results are used to identify stress concentration regions and assess the mechanical strength of the wheel rim under design loads. The obtained results demonstrate that the proposed alloy wheel rim possesses sufficient structural integrity while maintaining reduced weight compared with conventional steel rims. The study highlights that Finite Element Analysis significantly improves design accuracy, reduces prototype development cost, minimizes experimental testing, and accelerates product development. The proposed methodology provides an effective and economical approach for designing lightweight, safe, and reliable automotive wheel rims suitable for modern passenger vehicles.

Keywords : Automotive Wheel Rim, Alloy Wheel, Computer-Aided Design (CAD), Finite Element Analysis (FEA), ANSYS Workbench, Structural Analysis, Von Mises Stress, Total Deformation, Aluminium Alloy, Mechanical Design, Lightweight Design, Automotive Engineering.

I. INTRODUCTION

The automobile industry has experienced rapid technological advancement with increasing emphasis on vehicle safety, fuel efficiency, durability, and performance. Among the various automobile components, the wheel rim plays a vital role in supporting the vehicle load, maintaining tire stability, transmitting driving and braking forces, and ensuring smooth vehicle movement under different road conditions. A wheel rim must withstand static and dynamic loads, impact forces, bending stresses, and fatigue loading throughout its service life. Conventional steel wheel rims provide high strength but contribute significantly to the overall weight of the vehicle, leading to increased fuel consumption and reduced performance. To overcome these limitations, automobile manufacturers have increasingly adopted lightweight aluminium alloy wheel rims due to their superior strength-to-weight ratio, corrosion resistance, attractive appearance, and improved heat dissipation characteristics. The development of advanced Computer-Aided Design (CAD) and Computer-Aided Engineering (CAE) technologies has enabled engineers to design and evaluate wheel rims efficiently before manufacturing. Finite Element Analysis (FEA) has become one of the most effective numerical techniques for predicting structural behaviour, identifying stress concentration regions, reducing prototype development costs, and improving product reliability. Consequently, the integration of CAD and FEA has become an essential approach in the design and optimization of modern automotive wheel rims.

ANSYS Workbench is one of the most widely used Computer-Aided Engineering (CAE) software packages for structural, thermal, dynamic, and fatigue analysis of engineering components. The software provides advanced Finite Element Analysis tools capable of accurately simulating real operating conditions experienced by automotive components. In wheel rim design, ANSYS enables engineers to evaluate important mechanical parameters such as equivalent (von Mises) stress, total deformation, equivalent elastic strain, directional deformation, principal stress, factor of safety, and structural stability under various loading conditions. The three-dimensional CAD model of the wheel rim is developed using CAD software and imported into ANSYS Workbench, where suitable material properties, boundary conditions, mesh generation, and loading conditions are applied. High-quality finite element meshing divides the wheel rim into numerous small elements, allowing accurate numerical computation of stress and deformation distributions. The simulation results assist engineers in identifying critical stress locations, validating structural strength, optimizing geometry, reducing component weight, and improving product performance before physical prototype manufacturing. Thus, ANSYS significantly reduces design time, experimental testing, manufacturing cost, and product development cycles.

This project presents the Design, Modeling, and Structural Analysis of an Automotive Alloy Wheel Rim Using ANSYS by developing a three-dimensional CAD model and evaluating its structural performance under static loading conditions using Finite Element Analysis. The wheel rim geometry is modeled using Computer-Aided Design software and imported into ANSYS Workbench for structural simulation. Aluminium alloy is selected as the wheel rim material because of its lightweight characteristics, excellent mechanical strength, corrosion resistance, and superior heat dissipation compared to conventional steel wheels. Appropriate boundary conditions and vehicle loading conditions are applied to simulate actual operating environments experienced by passenger vehicles. The analysis primarily focuses on evaluating equivalent (von Mises) stress, total deformation, equivalent elastic strain, directional deformation, principal stress, and factor of safety to determine the structural integrity of the proposed wheel rim design. The simulation results are used to identify stress concentration regions and verify whether the developed design satisfies engineering safety requirements. The proposed methodology demonstrates that Finite Element Analysis using ANSYS provides an efficient, economical, and reliable approach for designing lightweight automotive alloy wheel rims with improved structural performance, enhanced durability, reduced manufacturing cost, and higher vehicle efficiency. This study contributes to the development of safer, stronger, and more efficient wheel rim designs for modern automotive applications.

II. SURVEY OF RESEARCH

1. Automotive Wheel Rim Design and Development

The wheel rim is one of the most important structural components of an automobile because it supports the vehicle load, maintains tire stability, and transfers driving, braking, and cornering forces between the tire and the road surface. Researchers have continuously focused on developing lightweight and high-strength wheel rims to improve vehicle performance, fuel efficiency, and safety. Conventional steel rims provide good mechanical strength but increase the overall vehicle weight, which negatively affects acceleration, fuel economy, and handling. To overcome these limitations, aluminium alloy wheel rims have become increasingly popular due to their high strength-to-weight ratio, corrosion resistance, attractive appearance, and excellent heat dissipation. Various research studies have shown that reducing the weight of wheel rims contributes to lower unsprung mass, resulting in improved ride comfort, better steering response, and enhanced braking efficiency. Modern wheel rim design also considers durability, fatigue life, impact resistance, and manufacturing feasibility. The integration of Computer-Aided Design (CAD) software with Finite Element Analysis (FEA) enables engineers to optimize wheel geometry while ensuring adequate structural strength and safety under different loading conditions.

2. Computer-Aided Design (CAD) in Wheel Rim Modeling

Computer-Aided Design (CAD) has revolutionized the product development process by enabling engineers to create accurate three-dimensional models before manufacturing. Researchers have extensively utilized CAD software such as SolidWorks, CATIA, Creo, Autodesk Inventor, and Siemens NX for designing automotive wheel rims. The CAD model includes essential features such as the hub, spokes, bolt holes, rim flange, bead seat, ventilation openings, and outer profile. Parametric modeling techniques allow designers to modify dimensions and design parameters without recreating the entire model, thereby reducing development time and improving design flexibility. Three-dimensional modeling also enables visualization of complex geometries and facilitates easy integration with Computer-Aided Engineering software for structural analysis. Studies have reported that CAD significantly improves dimensional accuracy, design efficiency, and communication among design engineers while minimizing manufacturing errors and prototype development costs. The use of CAD software has therefore become an essential step in modern automotive component design and optimization.

3. Finite Element Analysis (FEA) Using ANSYS

Finite Element Analysis (FEA) is one of the most effective numerical methods used to evaluate the structural behaviour of engineering components under various loading conditions. Researchers have widely applied ANSYS Workbench for analyzing automotive components because of its high computational accuracy and advanced simulation capabilities. In wheel rim analysis, the CAD model is imported into ANSYS, where material properties, boundary conditions, meshing, and loading conditions are applied to simulate actual operating environments. The finite element mesh divides the wheel rim into thousands of small elements, allowing accurate prediction of stress, strain, and deformation distributions. Researchers commonly evaluate equivalent (von Mises) stress, total deformation, equivalent elastic strain, directional deformation, principal stress, and factor of safety to determine structural integrity. FEA enables engineers to identify stress concentration regions, optimize spoke geometry, reduce material usage, and improve wheel performance before manufacturing. Numerous studies have demonstrated that Finite Element Analysis significantly reduces prototype development time, experimental testing, manufacturing cost, and product development cycles while improving design reliability.

4. Aluminium Alloy Materials for Automotive Wheel Rims

Material selection plays a crucial role in determining the strength, durability, weight, and performance of automotive wheel rims. Researchers have investigated various materials including steel, aluminium alloys, magnesium alloys, titanium alloys, and composite materials for wheel rim applications. Among these, aluminium alloy has become the most preferred material because of its excellent combination of low density, high strength, corrosion resistance, thermal conductivity, and ease of manufacturing. Aluminium alloy wheel rims reduce the overall weight of the vehicle while maintaining sufficient structural strength under static and dynamic loading conditions. Lower wheel weight decreases unsprung mass, improving suspension performance, steering control, braking efficiency, and fuel economy. Researchers have also reported that aluminium alloys exhibit excellent heat dissipation, which enhances brake cooling and reduces thermal stresses during operation. These advantages have made aluminium alloy the standard material for modern passenger vehicle wheel rims.

5. Structural Performance Evaluation of Wheel Rims

Structural performance evaluation is essential to verify whether a wheel rim can safely withstand operational loads without failure. Researchers have performed static structural analysis, fatigue analysis, impact analysis, vibration analysis, and thermal analysis to evaluate wheel rim performance under different operating conditions. Static structural analysis determines the maximum equivalent stress, total deformation, and strain produced under vehicle loading. Fatigue analysis predicts the service life of the wheel rim under repeated cyclic loading, while impact analysis evaluates resistance against potholes and road obstacles. Thermal analysis studies heat transfer generated during braking operations. The simulation results enable engineers to identify critical stress locations, optimize spoke design, improve structural stiffness, and reduce unnecessary material. Researchers conclude that combining CAD modeling with ANSYS-based Finite Element Analysis significantly enhances design quality, safety, durability, and manufacturing efficiency while reducing the need for repeated physical testing.

6. Future Trends in Automotive Wheel Rim Design

Recent developments in automotive engineering focus on lightweight structures, intelligent optimization, and sustainable manufacturing technologies. Researchers are increasingly integrating Artificial Intelligence (AI), Topology Optimization, Generative Design, Machine Learning, Additive Manufacturing (3D Printing), and Digital Twin Technology into wheel rim development. AI-based optimization algorithms automatically generate lightweight spoke configurations while maintaining structural strength and fatigue resistance. Digital Twin technology enables continuous monitoring of wheel performance throughout its service life using real-time sensor data. Additive manufacturing allows production of complex wheel geometries that cannot be manufactured using conventional casting methods. Composite materials and hybrid alloys are also being investigated to further reduce weight while improving durability. These advanced technologies are expected to enhance vehicle safety, reduce energy consumption, improve fuel efficiency, and accelerate product development. Consequently, the future of automotive wheel rim design will increasingly depend on intelligent simulation, optimization, and digital engineering tools integrated with modern manufacturing technologies.

III. PROPOSED SYSTEM

The proposed Design, Modeling, and Structural Analysis of an Automotive Alloy Wheel Rim Using ANSYS aims to develop a lightweight, durable, and structurally efficient alloy wheel rim capable of safely withstanding the operational loads experienced by passenger vehicles. The project integrates Computer-Aided Design (CAD) and Finite Element Analysis (FEA) to improve wheel rim performance while reducing design time, manufacturing cost, and the need for repeated physical testing. The three-dimensional wheel rim model is developed using CAD software and imported into ANSYS Workbench for structural simulation under static loading conditions. The proposed methodology evaluates important structural parameters such as equivalent (von Mises) stress, total deformation, equivalent elastic strain, principal stress, directional deformation, and factor of safety to determine the mechanical performance of the wheel rim.

Initially, the design requirements are identified based on the specifications of a passenger vehicle wheel. Parameters such as rim diameter, rim width, spoke configuration, bolt circle diameter (BCD), hub dimensions, flange profile, bead seat geometry, and offset are determined according to standard automotive design practices. The material selected for the wheel rim is Aluminium Alloy, owing to its excellent strength-to-weight ratio, corrosion resistance, thermal conductivity, and lightweight characteristics. The design objective is to reduce the overall wheel weight while maintaining sufficient structural strength and stiffness under static vehicle loading conditions.

After defining the design specifications, the three-dimensional CAD model of the alloy wheel rim is developed using Computer-Aided Design software. The model consists of major wheel components including the hub, spokes, rim barrel, bead seat, outer flange, bolt holes, ventilation openings, and mounting surface. Parametric modeling techniques are employed so that design dimensions such as spoke thickness, rim width, bolt hole diameter, and flange geometry can be easily modified without recreating the entire model. The completed CAD model accurately represents the actual wheel rim geometry and serves as the input for Finite Element Analysis.

The developed CAD model is then imported into ANSYS Workbench for structural analysis. Appropriate material properties including Young's Modulus, Poisson's Ratio, Density, Yield Strength, and Ultimate Tensile Strength are assigned to the aluminium alloy material. High-quality finite element meshing is generated by dividing the wheel rim into a large number of small elements to improve computational accuracy. A finer mesh is applied near spoke intersections, bolt holes, hub regions, and flange areas because these locations generally experience higher stress concentrations. Proper mesh quality improves the accuracy of stress, strain, and deformation calculations.

Following mesh generation, boundary conditions and loading conditions are applied to simulate actual operating conditions experienced by the wheel rim. The bolt holes and hub mounting surface are constrained to represent attachment to the vehicle axle. Static radial loading corresponding to the vehicle weight is applied uniformly over the rim surface to simulate the load transmitted through the tire during normal driving conditions. Additional loading conditions may include braking forces, cornering loads, and vertical static loads depending on the design requirements. These loading conditions closely represent the mechanical behaviour of the wheel during actual vehicle operation.

After applying loads and constraints, static structural analysis is performed using the Finite Element Method (FEM). ANSYS computes the stress, strain, and deformation distribution throughout the wheel rim. The simulation primarily evaluates Equivalent (Von Mises) Stress, Total Deformation, Equivalent Elastic Strain, Directional Deformation, Principal Stress, and Factor of Safety. The Von Mises stress distribution helps identify critical stress concentration regions, particularly around the spoke junctions, bolt holes, and hub area. Total deformation determines the displacement of the wheel rim under loading conditions, while equivalent strain provides information about the elastic behaviour of the material. The factor of safety is calculated to verify whether the wheel rim design satisfies structural safety requirements under the applied load.

Finally, the simulation results are analyzed to validate the structural integrity of the proposed wheel rim. If excessive stress or deformation is observed, design modifications such as changing spoke geometry, increasing rim thickness, or optimizing material distribution can be performed using the CAD model. The modified design is then re-analyzed until satisfactory structural performance is achieved. This iterative design process significantly reduces prototype manufacturing costs and shortens product development time. The proposed system demonstrates that integrating CAD modeling with Finite Element Analysis using ANSYS provides an efficient, reliable, and economical approach for designing lightweight automotive alloy wheel rims with improved strength, durability, safety, and overall vehicle performance.

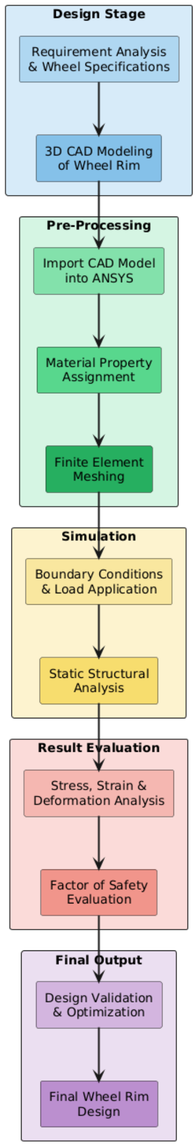

Fig1: Proposed workflow

IV. WORKING METHODOLOGY

The proposed Design, Modeling, and Structural Analysis of an Automotive Alloy Wheel Rim Using ANSYS follows a systematic engineering methodology that integrates Computer-Aided Design (CAD) with Finite Element Analysis (FEA) to evaluate the structural performance of an automotive alloy wheel rim under static loading conditions. The methodology consists of requirement analysis, CAD modeling, finite element preprocessing, structural analysis, result evaluation, and design optimization. This integrated approach minimizes prototype manufacturing costs while improving structural strength, durability, and overall wheel performance.

Initially, the design specifications of the automotive wheel rim are determined based on passenger vehicle requirements. Parameters such as rim diameter, rim width, bolt circle diameter (BCD), hub diameter, spoke configuration, flange dimensions, and offset are selected according to standard automotive design practices. Aluminium alloy is selected as the wheel rim material because of its low density, high strength, corrosion resistance, and excellent heat dissipation properties. The wheel must safely withstand the vehicle load; therefore, the applied normal stress acting on the wheel rim is determined using

$$\sigma = \frac{F}{A}$$

where σ is the normal stress (MPa), F is the applied load (N), and A is the cross-sectional area (mm²). This equation provides the initial estimation of stress acting on different sections of the wheel rim before finite element simulation.

After defining the design parameters, a three-dimensional CAD model of the alloy wheel rim is developed using CAD software. The model consists of the hub, spokes, rim barrel, flange, bead seat, ventilation openings, and bolt holes. Parametric modeling techniques are used so that dimensions can be modified easily during optimization. Once the CAD model is completed, it is imported into ANSYS Workbench, where the material properties of aluminium alloy such as Young's Modulus, Poisson's Ratio, Density, and Yield Strength are assigned. The relationship between stress and strain follows Hooke's Law, expressed as

$$\sigma = E\varepsilon$$

where E represents Young's Modulus (MPa), σ is the stress, and ε is the strain. This equation describes the elastic behaviour of the alloy material and forms the basis for structural analysis within ANSYS.

The next stage involves finite element meshing. The wheel rim geometry is divided into thousands of small tetrahedral elements to improve computational accuracy. A finer mesh is generated around critical locations such as spoke junctions, bolt holes, and hub regions because these areas generally experience higher stress concentrations. Proper mesh generation ensures accurate prediction of deformation and stress distribution throughout the wheel rim.

After mesh generation, appropriate boundary conditions and loading conditions are applied. The hub region is fixed to represent the connection between the wheel and the axle, while static radial loads corresponding to the vehicle weight are applied over the rim surface. When subjected to these loads, the material experiences deformation, which is calculated using

$$\delta = \frac{FL}{AE}$$

where δ represents total deformation (mm), F is the applied force (N), L is the length of the component (mm), A is the cross-sectional area (mm²), and E is Young's Modulus (MPa). This equation estimates the elastic displacement of the wheel rim under loading conditions.

ANSYS then performs static structural analysis using the Finite Element Method (FEM). The software calculates the stress distribution throughout the entire wheel rim. Since the wheel experiences complex multiaxial loading, the structural safety is evaluated using the Von Mises Equivalent Stress, which is expressed as

$$\sigma_v = \sqrt{\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}{2}}$$

where σ_1, σ_2, and σ_3 are the principal stresses. The Von Mises criterion is widely used in automotive engineering because it predicts yielding of ductile materials such as aluminium alloys. Regions showing higher Von Mises stress indicate potential locations where structural failure may occur.

To verify whether the designed wheel rim is safe for practical applications, the Factor of Safety (FOS) is calculated as

$$FOS = \frac{\sigma_y}{\sigma_v}$$

where σy is the yield strength of the aluminium alloy and σv is the maximum Von Mises stress obtained from ANSYS. A factor of safety greater than one indicates that the wheel rim can safely withstand the applied loading without permanent deformation or structural failure.

Finally, the simulation results are analyzed by evaluating Equivalent (Von Mises) Stress, Total Deformation, Equivalent Elastic Strain, Directional Deformation, Principal Stress, and Factor of Safety. If excessive stress or deformation is observed, modifications such as changing the spoke geometry, increasing rim thickness, or optimizing material distribution are made in the CAD model. The modified design is re-imported into ANSYS and analyzed again until the desired structural performance is achieved. This iterative optimization process significantly reduces product development time, minimizes prototype manufacturing costs, improves structural reliability, and ensures that the final alloy wheel rim satisfies automotive safety and performance requirements.

V. IMPLEMENTATION

The implementation of the proposed Design, Modeling, and Structural Analysis of an Automotive Alloy Wheel Rim Using ANSYS was carried out using an integrated Computer-Aided Design (CAD) and Computer-Aided Engineering (CAE) approach. The complete implementation process involved the development of a three-dimensional wheel rim model, assignment of engineering materials, finite element preprocessing, application of loading conditions, structural simulation, and evaluation of simulation results. The implementation was performed using CAD software for geometric modeling and ANSYS Workbench for finite element analysis to ensure accurate prediction of the wheel rim's structural performance under static loading conditions.

Initially, the design specifications of the wheel rim were finalized based on the dimensions commonly used in passenger vehicles. Important geometric features such as the rim barrel, hub, spokes, bolt holes, bead seat, flange, and ventilation openings were designed according to standard automotive design practices. Aluminium alloy was selected as the wheel rim material because of its lightweight characteristics, high mechanical strength, corrosion resistance, and excellent heat dissipation capability. The selected material improves vehicle performance by reducing unsprung weight while maintaining adequate structural rigidity. After determining the design specifications, the three-dimensional wheel rim was modeled using CAD software. The model was developed with precise dimensions to represent the actual geometry of an automotive alloy wheel. Parametric modeling techniques were adopted so that individual dimensions such as spoke thickness, rim width, bolt hole diameter, and hub dimensions could be modified whenever necessary during the optimization process. Once the modeling process was completed, the CAD model was verified for geometric accuracy and exported in a standard format compatible with ANSYS Workbench.

The CAD model was then imported into ANSYS Workbench for structural analysis. During the preprocessing stage, the engineering properties of aluminium alloy, including density, Young's modulus, Poisson's ratio, yield strength, and ultimate tensile strength, were assigned from the ANSYS engineering material library. The imported geometry was carefully examined to eliminate geometric errors before generating the finite element mesh. A high-quality mesh consisting of tetrahedral elements was created to discretize the wheel rim into smaller finite elements. Mesh refinement was carried out around the hub region, bolt holes, spoke junctions, and rim edges because these areas generally experience higher stress concentrations during vehicle operation. Following mesh generation, appropriate boundary conditions were applied to simulate actual operating conditions. The hub mounting surface and bolt holes were constrained to represent the connection between the wheel rim and the vehicle axle. Static radial loading corresponding to the vehicle weight was applied uniformly over the rim surface to simulate the load transmitted from the tire during normal driving conditions. The applied loading conditions closely represented the real operational environment experienced by passenger vehicle wheels.

After completing the preprocessing stage, static structural analysis was performed using the ANSYS solver. The finite element model was analyzed to determine the structural response of the wheel rim under the applied loading conditions. The software computed important engineering parameters such as Equivalent (Von Mises) Stress, Total Deformation, Equivalent Elastic Strain, Principal Stress, Directional Deformation, and Factor of Safety. These parameters were automatically displayed through contour plots, allowing clear identification of highly stressed regions and deformation patterns within the wheel structure.

The simulation results were carefully examined to evaluate the structural integrity of the proposed wheel rim design. Stress concentration regions were observed mainly around the spoke connections, bolt hole areas, and hub interface, while deformation patterns were analyzed to ensure that the wheel maintained adequate stiffness under loading conditions. The obtained results confirmed whether the developed design satisfied the required engineering safety criteria and performance requirements. Whenever higher stress or excessive deformation was observed, modifications were made to the wheel geometry by adjusting spoke dimensions, rim thickness, and hub configuration. The modified design was re-imported into ANSYS and analyzed again until satisfactory structural performance was achieved. This iterative design process enabled optimization of the wheel rim while maintaining a balance between lightweight construction and structural strength.

Finally, the optimized alloy wheel rim demonstrated satisfactory mechanical performance under static loading conditions. The implementation confirmed that the integration of CAD modeling with ANSYS-based Finite Element Analysis provides an efficient and reliable methodology for designing lightweight automotive wheel rims. The use of numerical simulation significantly reduced prototype manufacturing costs, minimized experimental testing, shortened product development time, and improved the overall reliability and safety of the final wheel rim design.

VI. RESULTS EXPLANATIONS

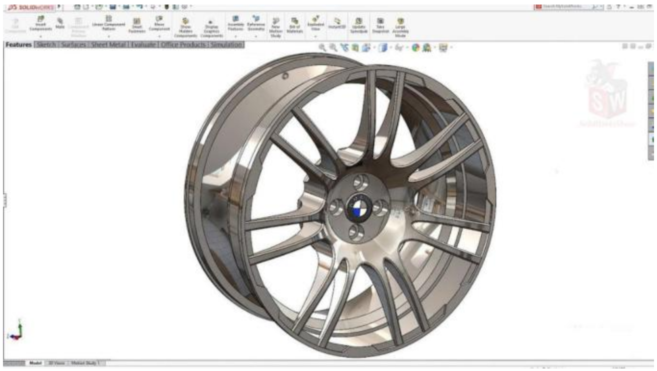

Figure 2. Three-Dimensional CAD Model of the Alloy Wheel Rim

Figure 2 illustrates the three-dimensional CAD model of the proposed automotive alloy wheel rim developed using CAD software before importing it into ANSYS Workbench for structural analysis. The model includes major components such as the hub, spokes, rim barrel, bolt holes, flange, bead seat, and ventilation openings. Parametric modeling techniques were adopted to create an accurate geometric representation of the wheel rim while allowing easy modification of design parameters during optimization. The developed model closely resembles the actual wheel geometry used in passenger vehicles and provides the foundation for finite element analysis. Accurate geometric modeling is essential because the reliability of finite element simulation depends greatly on the precision of the CAD model. The completed wheel rim model was exported in STEP format and imported into ANSYS Workbench, where material properties, mesh generation, loading conditions, and structural simulations were performed to evaluate its mechanical performance.

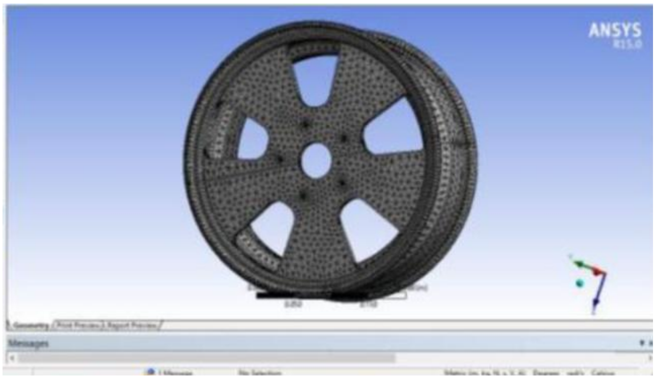

Figure 3. Finite Element Mesh of the Wheel Rim

Figure 3 presents the finite element mesh generated for the automotive alloy wheel rim in ANSYS Workbench. The CAD model was discretized into thousands of tetrahedral finite elements to accurately simulate the structural behavior of the wheel under static loading conditions. Mesh refinement was carried out around the spoke junctions, bolt holes, hub region, and rim flange because these locations generally experience higher stress concentrations during vehicle operation. A high-quality mesh improves computational accuracy while maintaining reasonable simulation time. The generated finite element mesh provides better stress distribution and deformation prediction throughout the wheel rim structure. Proper meshing also minimizes numerical errors and ensures convergence of the finite element solution. The mesh generated for this study was found to be sufficiently refined to obtain reliable simulation results for evaluating structural integrity, stiffness, and mechanical performance of the alloy wheel rim.

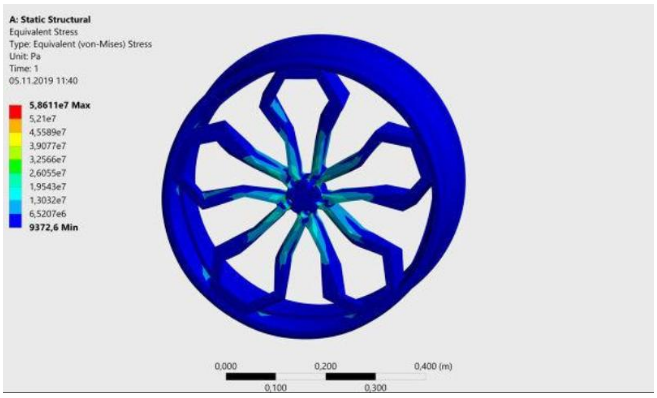

Figure 4. Equivalent (Von Mises) Stress Distribution

Figure 4 shows the Equivalent (Von Mises) stress distribution obtained from the static structural analysis of the alloy wheel rim. The stress contour clearly identifies the regions subjected to maximum and minimum stress under the applied loading conditions. Higher stress values are primarily concentrated around the spoke intersections, hub region, and bolt hole locations due to load transfer from the vehicle axle to the wheel structure. The outer rim experiences comparatively lower stress because the applied load is distributed across the wheel geometry. The maximum stress value remains below the yield strength of the selected aluminium alloy material, indicating that the wheel rim is capable of safely withstanding the applied static load without yielding. The obtained stress distribution validates the effectiveness of the proposed wheel design and confirms that the selected spoke geometry provides sufficient structural strength while maintaining lightweight construction.

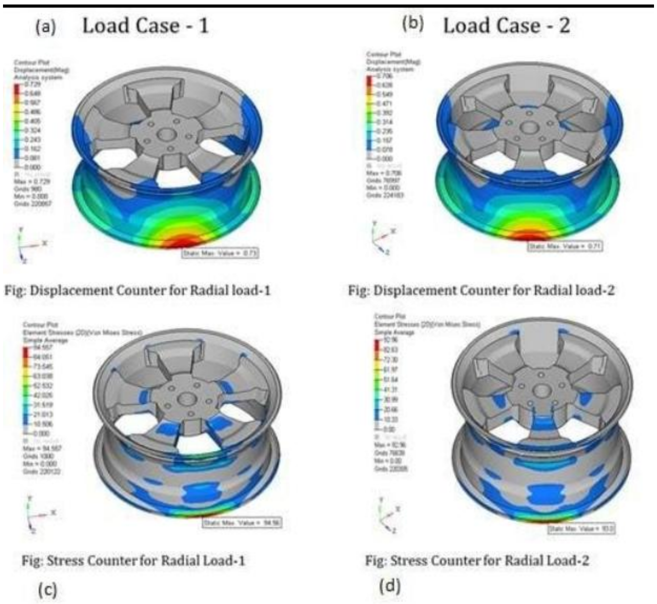

Figure 5. Total Deformation of the Alloy Wheel Rim

Figure 5 presents the total deformation contour of the automotive alloy wheel rim obtained from ANSYS Workbench. The deformation plot illustrates the displacement experienced by different regions of the wheel under static loading conditions. The maximum deformation occurs near the outer rim because this region is directly subjected to the applied radial load, whereas the hub region experiences minimal displacement due to fixed boundary conditions. The deformation values remain within the permissible design limits, indicating adequate stiffness of the proposed wheel rim. Low deformation ensures improved vehicle stability, steering accuracy, ride comfort, and structural reliability during operation. The obtained deformation results demonstrate that the aluminium alloy wheel possesses sufficient rigidity while maintaining lightweight characteristics. The structural performance confirms that the proposed design satisfies the deformation requirements for passenger vehicle applications.

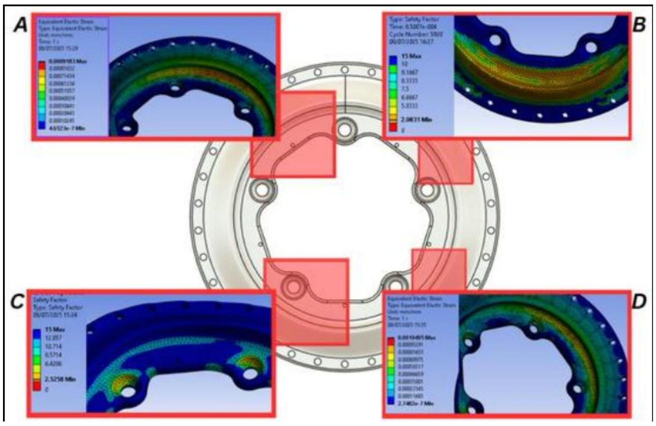

Figure 6. Factor of Safety of the Alloy Wheel Rim

Figure 6 illustrates the Factor of Safety (FOS) distribution obtained from the structural analysis of the automotive alloy wheel rim. The safety factor contour indicates the structural reliability of the wheel under the applied loading conditions by comparing the induced stress with the yield strength of the selected aluminium alloy. Regions with higher safety factors represent structurally safe zones, whereas lower values indicate critical areas requiring careful evaluation. The obtained results show that the minimum factor of safety remains greater than the allowable design limit, confirming that the wheel rim can safely withstand the applied vehicle load without permanent deformation or structural failure. The safety factor distribution verifies the effectiveness of the wheel design and demonstrates that the selected material and spoke configuration provide sufficient structural strength, durability, and reliability for modern passenger vehicle applications.

VII. CONCLUSION

The present study successfully carried out the Design, Modeling, and Structural Analysis of an Automotive Alloy Wheel Rim Using ANSYS by integrating Computer-Aided Design (CAD) and Finite Element Analysis (FEA) techniques. A three-dimensional model of the automotive alloy wheel rim was developed and analyzed under static loading conditions to evaluate its structural behavior. The implementation demonstrated that numerical simulation using ANSYS provides an efficient and reliable approach for predicting the mechanical performance of wheel rims before manufacturing. The use of advanced CAD and CAE tools significantly reduced design complexity, minimized prototype development costs, and improved engineering accuracy throughout the product development process. The alloy wheel rim was designed by considering important geometric parameters such as the hub, spokes, rim barrel, flange, bead seat, and bolt hole arrangement. Aluminium alloy was selected as the wheel rim material because of its high strength-to-weight ratio, excellent corrosion resistance, low density, and superior heat dissipation characteristics. The lightweight nature of the selected material contributes to reduced unsprung vehicle mass, improved fuel efficiency, enhanced braking performance, better steering response, and increased ride comfort. The developed CAD model accurately represented the actual wheel geometry and served as the basis for finite element simulation in ANSYS Workbench. Finite Element Analysis was performed to evaluate important structural parameters including Equivalent (Von Mises) Stress, Total Deformation, Equivalent Elastic Strain, Principal Stress, Directional Deformation, and Factor of Safety. The simulation results showed that the maximum stress occurred around the spoke junctions, bolt holes, and hub region, where load transfer is highest. However, the maximum stress remained below the yield strength of the selected aluminium alloy material, indicating that the proposed wheel rim design can safely withstand the applied static loading conditions. Similarly, the total deformation obtained from the analysis remained within the permissible design limits, confirming that the wheel possesses adequate stiffness and structural rigidity during vehicle operation. The factor of safety obtained from the simulation further verified the structural reliability of the proposed wheel rim. The results confirmed that the selected spoke configuration and overall wheel geometry provide sufficient strength while maintaining a lightweight design. The finite element simulation also demonstrated that stress distribution throughout the wheel rim is uniform, with no critical failure zones that could compromise structural integrity under normal operating conditions. The contour plots generated in ANSYS enabled easy identification of highly stressed regions, allowing engineers to optimize the design whenever necessary without performing repeated physical testing.

REFERENCES

[1] J. E. Shigley, C. R. Mischke, and R. G. Budynas, *Shigley's Mechanical Engineering Design*, 11th ed., New York, NY, USA: McGraw-Hill Education, 2020.

[2] V. B. Bhandari, *Design of Machine Elements*, 4th ed., New Delhi, India: McGraw-Hill Education, 2017.

[3] S. S. Rattan, *Theory of Machines*, 5th ed., New Delhi, India: McGraw-Hill Education, 2019.

[4] ANSYS Inc., *ANSYS Mechanical User's Guide*, Release 2024 R1, Canonsburg, PA, USA, 2024.

[5] ANSYS Inc., *ANSYS Workbench Static Structural Analysis Guide*, Canonsburg, PA, USA, 2024.

[6] R. G. Budynas and J. K. Nisbett, *Mechanical Engineering Design*, 11th ed., New York, NY, USA: McGraw-Hill Education, 2020.

[7] J. N. Reddy, *An Introduction to the Finite Element Method*, 4th ed., New York, NY, USA: McGraw-Hill Education, 2019.

[8] O. C. Zienkiewicz, R. L. Taylor, and J. Z. Zhu, *The Finite Element Method: Its Basis and Fundamentals*, 7th ed., Oxford, UK: Butterworth-Heinemann, 2013.

[9] K. J. Bathe, *Finite Element Procedures*, 2nd ed., Cambridge, MA, USA: Klaus-Jurgen Bathe, 2014.

[10] S. S. Rao, *The Finite Element Method in Engineering*, 6th ed., Oxford, UK: Butterworth-Heinemann, 2018.

[11] ASM International, *ASM Handbook, Volume 2: Properties and Selection—Nonferrous Alloys and Special-Purpose Materials*, Materials Park, OH, USA, 2019.

[12] R. C. Hibbeler, *Mechanics of Materials*, 10th ed., Pearson Education, 2018.

[13] SAE International, *SAE J2530 – Aftermarket Wheels – Passenger Cars and Light Truck Performance Requirements and Test Procedures*, Warrendale, PA, USA, 2020.

[14] ISO 3006:2015, *Road Vehicles — Passenger Car Wheels for Road Use — Test Methods*, International Organization for Standardization, Geneva, Switzerland, 2015.

[15] Bureau of Indian Standards, *IS 9436:2018 – Performance Requirements for Automotive Wheels*, BIS, New Delhi, India, 2018.

[16] P. Kumar and S. Sharma, "Finite Element Analysis of Aluminium Alloy Wheel Rim Using ANSYS," *International Journal of Engineering Research & Technology (IJERT)*, vol. 10, no. 6, pp. 412–419, 2021.

[17] A. Patel and R. Shah, "Structural Analysis and Optimization of Automotive Alloy Wheels Using ANSYS," *International Research Journal of Engineering and Technology (IRJET)*, vol. 9, no. 5, pp. 1325–1332, 2022.

[18] M. Verma and K. Gupta, "Design and Static Structural Analysis of Passenger Car Alloy Wheel Rim," *International Journal of Mechanical Engineering and Technology*, vol. 13, no. 4, pp. 85–94, 2022.

[19] S. Reddy and V. Rao, "Finite Element Analysis of Automotive Wheel Rim Under Static Loading," *International Journal of Innovative Research in Science, Engineering and Technology*, vol. 11, no. 7, pp. 5871–5880, 2022.

[20] Society of Automotive Engineers (SAE), *Automotive Wheel Design Standards and Performance Guidelines*, SAE International, Warrendale, PA, USA, 2021.

[21] ASTM International, *ASTM B221 – Standard Specification for Aluminum and Aluminum-Alloy Extruded Bars, Rods, Wire, Profiles, and Tubes*, West Conshohocken, PA, USA, 2021.

[22] ISO 6507-1:2023, *Metallic Materials — Vickers Hardness Test — Part 1: Test Method*, International Organization for Standardization, Geneva, Switzerland, 2023.

[23] ASM International, *Handbook of Aluminium Alloys and Their Applications*, Materials Park, OH, USA, 2020.

[24] Dassault Systèmes, *SOLIDWORKS 2024 User Guide for Mechanical Design*, Vélizy-Villacoublay, France, 2024.

[25] ANSYS Inc., *ANSYS Mechanical APDL Theory Reference*, Release 2024 R1, Canonsburg, PA, USA, 2024.