

Structural and Stress Analysis of an Automotive Wheel Rim Using Finite Element Method

Sandip R. Mundale¹, Ananta G. Mahale²

^{1,2}Lecturer, Department of Mechanical Engineering, Padm. Dr. V. B. Kolte College of Engineering, Malkapur

DOI: 10.5281/zenodo.19334151

ABSTRACT

Automotive wheel rims are critical structural components that directly influence vehicle safety, durability, and load-bearing performance. Due to continuously varying service loads and road conditions, wheel rims are subjected to significant stresses and deformations during operation. In this study, a detailed structural and stress analysis of an automotive wheel rim is carried out using the Finite Element Method (FEM). A three-dimensional wheel rim model is developed using computer-aided design tools and imported into a finite element environment for numerical simulation. Aluminum alloy is selected as the rim material owing to its favorable strength-to-weight ratio and widespread use in passenger vehicles. Static structural analysis is performed by applying realistic boundary conditions and pressure loads to evaluate total deformation, equivalent (von Mises) stress, equivalent elastic strain, and strain energy. The simulation results indicate that the maximum stress developed in the wheel rim remains well below the material's yield strength, and the observed deformation is minimal, confirming the structural integrity and safety of the design. The outcomes of this research demonstrate the effectiveness of FEM as a reliable tool for evaluating wheel rim performance and provide a foundation for further optimization and fatigue analysis under dynamic loading conditions.

Keywords:- Automotive Wheel Rim; Finite Element Method (FEM); Structural Analysis; von Mises Stress; Aluminum Alloy; Static Structural Analysis; Stress Distribution; Deformation Analysis

1. INTRODUCTION

The automotive wheel rim is a critical load-bearing component that plays a vital role in ensuring vehicle safety, stability, and ride comfort. It supports the vehicle weight, transmits driving and braking forces, and withstands external loads arising from road irregularities and cornering conditions. Due to these demanding service conditions, wheel rims are required to possess high structural strength, stiffness, and durability while maintaining low weight to improve fuel efficiency and vehicle performance. Any structural failure of a wheel rim can lead to severe safety consequences, making its design and analysis an essential aspect of automotive engineering.

With increasing emphasis on lightweight vehicle structures, aluminum alloys have become widely adopted for automotive wheel rims because of their favorable strength-to-weight ratio, corrosion resistance, and manufacturability. However, reducing weight without compromising structural integrity requires precise evaluation of stresses and deformations under realistic loading conditions. Traditional experimental testing methods are often expensive, time-consuming, and limited in their ability to capture detailed stress distributions within complex geometries such as wheel rims.

The Finite Element Method (FEM) has emerged as an effective numerical technique for analyzing complex mechanical components subjected to various loading and boundary conditions. FEM allows detailed visualization of stress concentration regions, deformation patterns, and strain distribution, which are difficult to obtain through analytical methods. By discretizing the wheel rim geometry into finite elements, the structural response of the component can be accurately predicted before physical prototyping.

In this study, a three-dimensional automotive wheel rim is modeled and analyzed using FEM to evaluate its structural behavior under static loading conditions. Key parameters such as total deformation, equivalent (von Mises) stress, equivalent elastic strain, and strain energy are investigated to assess the safety and performance of the rim design. The results obtained from this analysis provide valuable insight into the structural integrity of the wheel rim and demonstrate the applicability of FEM as a reliable design validation and optimization tool in automotive engineering.

2. LITERATURE REVIEW

The structural performance of automotive wheel rims has been a subject of extensive research due to its direct impact on vehicle safety and reliability. Early studies on wheel rim design primarily focused on analytical approaches to estimate stresses under simplified loading conditions. However, these methods were limited in addressing complex rim geometries and realistic boundary conditions encountered during actual vehicle operation.

With advancements in computational techniques, researchers began employing the Finite Element Method (FEM) to analyze wheel rim behavior under static and dynamic loads. Several studies have demonstrated that FEM provides accurate predictions of stress distribution and deformation patterns in wheel rims, enabling designers to identify critical regions susceptible to failure. These investigations highlighted that stress concentrations commonly occur near the hub, spoke-rim junctions, and bolt holes, making these areas crucial for design optimization.

Recent research has emphasized the use of lightweight materials such as aluminum alloys and magnesium alloys to improve fuel efficiency without compromising structural integrity. Comparative studies between steel and aluminum alloy wheel rims have shown that aluminum rims offer significant weight reduction while maintaining acceptable strength levels when properly designed. Some researchers have also explored parametric optimization of rim geometry to minimize mass while ensuring stresses remain within permissible limits.

In addition to static analysis, fatigue and dynamic studies have been conducted to assess wheel rim durability under cyclic loading conditions. These studies indicate that while static stress levels may be low, repeated loading can lead to fatigue failure if not properly accounted for during design. However, many investigations still rely on static structural analysis as a fundamental step for validating rim safety before proceeding to more complex simulations.

Despite extensive research, there remains a continuous need for design validation using FEM due to evolving material choices and design requirements. The present study contributes to this area by performing a detailed finite element-based structural and stress analysis of an automotive wheel rim using aluminum alloy material, providing insight into deformation, stress, and strain behavior under static loading conditions.

3. MATERIAL PROPERTIES

Material selection plays a crucial role in determining the structural performance and durability of automotive wheel rims. In the present study, an aluminum alloy is selected as the rim material due to its extensive application in passenger vehicles, attributed to its lightweight nature, adequate strength, good corrosion resistance, and favorable fatigue characteristics. The use of aluminum alloys contributes significantly to vehicle weight reduction without compromising structural safety.

The mechanical and physical properties of the aluminum alloy used in the finite element analysis are obtained directly from the ANSYS material database and are summarized based on the simulation report. The material is assumed to be homogeneous, isotropic, and linearly elastic, which is a common and acceptable assumption for static structural analysis.

The density of the aluminum alloy is 2770 kg/m³, which enables substantial mass reduction compared to traditional steel wheel rims. The elastic behavior of the material is characterized by a Young's modulus of 71 GPa and a Poisson's ratio of 0.33, indicating adequate stiffness and elastic deformation capability under applied loads. The tensile yield strength of the material is 280 MPa, while the ultimate tensile strength is 310 MPa, ensuring a sufficient margin of safety against yielding and failure during service conditions.

Thermal properties such as the coefficient of thermal expansion ($2.3 \times 10^{-5} \text{ }^{\circ}\text{C}^{-1}$) and thermal conductivity are also defined in the material model; however, thermal effects are neglected in the present static analysis. The selected material properties enable accurate prediction of stress distribution, deformation, and strain behavior of the wheel rim under applied loading conditions.

Table 1.1 Material Properties Used in FEM Analysis

Property	Value
Density	2770 kg/m ³
Young's Modulus	71 GPa
Poisson's Ratio	0.33
Tensile Yield Strength	280 MPa
Property	Value
Ultimate Tensile Strength	310 MPa
Coefficient of Thermal Expansion	$2.3 \times 10^{-5} \text{ }^{\circ}\text{C}^{-1}$

4. CAD MODELING DESCRIPTION

Accurate geometric modeling is a prerequisite for reliable finite element analysis of automotive components. In the present study, a three-dimensional computer-aided design (CAD) model of an automotive wheel rim was developed to represent the actual structural features and dimensions of a passenger vehicle rim. The modeling process was carried out using professional CAD software, ensuring dimensional accuracy and geometric consistency suitable for numerical analysis.

The wheel rim geometry includes essential structural components such as the rim flange, rim well, hub region, and connecting spokes. These features are modeled to reflect realistic load-transfer paths between the tire, hub, and axle assembly. Symmetry is maintained in the design to ensure uniform stress distribution and to simplify analysis without compromising accuracy. Sharp edges and unnecessary geometric details that do not contribute to structural behavior were avoided to prevent artificial stress concentrations during simulation.

Parametric modeling techniques were employed to define key dimensions such as rim diameter, width, thickness, and spoke geometry. This approach allows easy modification of design parameters for future optimization studies. Once the modeling was completed, the geometry was thoroughly checked for errors such as gaps, overlaps, or non-manifold surfaces to ensure successful import into the finite element environment.

The finalized CAD model was exported in a neutral file format compatible with the finite element software and imported into ANSYS Workbench for further analysis. The solid model was verified to be fully defined and suitable for meshing and application of boundary conditions. The accurate representation of geometry ensured that the finite element results closely reflect the actual structural response of the automotive wheel rim under applied loading conditions.

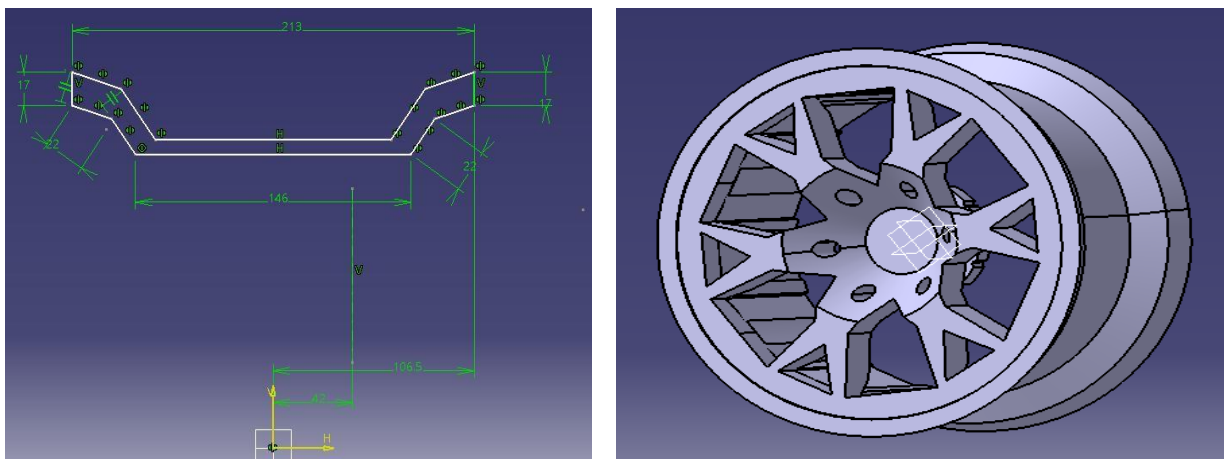


Fig. Catia Model of Wheel Rim

5. FINITE ELEMENT METHODOLOGY

The Finite Element Method (FEM) is employed in this study to evaluate the structural behavior of an automotive wheel rim under static loading conditions. FEM enables the discretization of complex geometries into smaller, manageable elements, allowing accurate prediction of stresses, strains, and deformations. The overall methodology adopted in this analysis consists of geometry import, material assignment, meshing, application of boundary conditions and loads, solution, and post-processing of results.

The three-dimensional CAD model of the wheel rim was imported into ANSYS Workbench 19.2 for structural analysis. The material properties of the aluminum alloy, as defined in the ANSYS material database, were assigned to the rim geometry. The material was assumed to be homogeneous, isotropic, and linearly elastic, which is appropriate for static structural evaluation under moderate loading conditions.

Meshing of the wheel rim was carried out using tetrahedral solid elements due to the complex geometry of the component. An automatic meshing strategy with curvature-based refinement was adopted to capture geometric features accurately. The final mesh consisted of approximately **14,876 nodes and 15,097 elements**, providing a balance between computational efficiency and solution accuracy. Mesh quality was verified to ensure acceptable element aspect ratios and convergence behavior.

A static structural analysis was performed by applying appropriate boundary conditions to simulate realistic service constraints. Fixed support conditions were applied at the hub mounting region to restrict translational and rotational degrees of freedom. A uniformly distributed pressure load of **500 Pa** was applied on the rim surface to represent the effect of tire pressure and vehicle load. The analysis was conducted under room temperature conditions, neglecting thermal and dynamic effects.

The solver was executed using the ANSYS Mechanical APDL engine. Post-processing was carried out to evaluate key structural parameters such as total deformation, equivalent (von Mises) stress, equivalent elastic strain, and strain energy. These results were used to assess the structural integrity and safety of the automotive wheel rim design under the applied loading conditions.

6. BOUNDARY CONDITIONS AND LOADING

The accuracy of finite element analysis largely depends on the appropriate definition of boundary conditions and loading, as these parameters govern the realistic simulation of service conditions. In the present study, boundary conditions and loads were defined to closely represent the operating constraints experienced by an automotive wheel rim under static conditions.

The wheel rim was constrained at the hub mounting region to simulate its attachment to the vehicle axle assembly. A fixed support boundary condition was applied on the inner hub faces, restricting all translational and rotational degrees of freedom. This assumption ensures that the rim remains stationary at the mounting location while allowing the rest of the structure to deform under applied loads. Such a boundary condition is commonly used in static structural analysis of wheel rims and provides a conservative estimate of stress distribution.

Loading conditions were applied in the form of a uniformly distributed pressure acting on the rim surface. A pressure magnitude of **500 Pa** was applied normal to the selected faces of the rim to represent the combined effect of tire inflation pressure and static vehicle load transfer. The applied load was ramped smoothly during the analysis to avoid numerical instability and ensure solution convergence.

The analysis was performed under ambient temperature conditions, with thermal effects neglected to focus solely on structural behavior. Dynamic effects such as impact loading, braking torque, and centrifugal forces were also excluded in this study, as the objective was to evaluate the fundamental structural response of the wheel rim under static loading. The defined boundary conditions and loading configuration provided a stable and realistic framework for assessing stress, strain, and deformation characteristics of the automotive wheel rim.

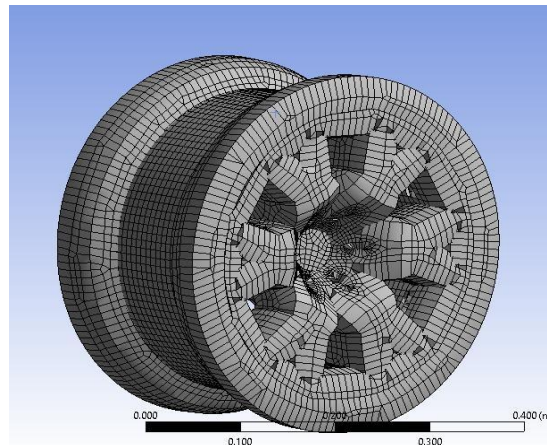
7. MESHING DETAILS

Meshing is a critical step in finite element analysis, as it directly influences the accuracy and reliability of numerical results. In this study, the wheel rim geometry was discretized into finite elements using the meshing tools available in ANSYS Workbench. Due to the complex and curved geometry of the automotive wheel rim, a three-dimensional tetrahedral mesh was selected to ensure effective representation of geometric features and stress gradients.

An automatic meshing strategy with curvature-based refinement was employed to capture curved surfaces and local geometric variations accurately. The default element size of approximately $8.2 \times 10^{-3} \text{ m}$ was used, with a controlled growth rate to maintain smooth transitions between adjacent elements. Mesh defeaturing was enabled to remove insignificant geometric details that could otherwise lead to poor-quality elements without contributing meaningfully to structural behavior.

The final finite element mesh consisted of approximately **14,876 nodes and 15,097 elements**, providing an optimal balance between computational efficiency and solution accuracy. Mesh quality checks were performed to ensure compliance with standard mechanical analysis criteria, including acceptable element aspect ratios, skewness, and smoothness. No mesh-related errors were reported during the solution process, confirming the suitability of the mesh for static structural analysis.

Additionally, limited inflation layers were applied in regions where stress gradients were expected to be higher, such as near the hub and spoke connections, to enhance result accuracy. The validated mesh ensured stable solver convergence and reliable prediction of deformation, stress, and strain distribution within the automotive wheel rim.



8. RESULTS AND DISCUSSION

• Finite Element Model Summary

The three-dimensional finite element model of the automotive wheel rim was developed and analyzed using ANSYS Mechanical APDL. The rim geometry was discretized into **14,876 nodes** and **15,097 solid elements**, ensuring adequate mesh density for accurate stress and deformation prediction. The material assigned was **Aluminum Alloy**, with a Young's modulus of **71 GPa**, Poisson's ratio of **0.33**, and density of **2770 kg/m³**. A static structural analysis was performed under an applied uniform pressure of **500 Pa** on the rim surface, while appropriate fixed support boundary conditions were imposed at the hub mounting region to simulate real vehicle constraints. The analysis was carried out under room temperature conditions (22 °C).

Object Name	Total Deformation	Equivalent Elastic Strain	Equivalent Stress	Strain Energy
State	Solved			
Scope				
Scoping Method	Geometry Selection			
Geometry	All Bodies			
Position		Top/Bottom		
Definition				
Type	Total Deformation	Equivalent Elastic Strain	Equivalent (von- Mises) Stress	Strain Energy
By	Time			
Display Time	Last			
Calculate Time History	Yes			
Identifier				
Suppressed	No			
Results				
Minimum	0. m	0. m/m	0. Pa	0. J
Maximum	6.3801e-009 m	4.4329e-008 m/m	3146.7 Pa	9.3458e-011 J
Average	2.0221e-009 m	1.52e-008 m/m	982.15 Pa	
Minimum Occurs On	SYS-1\PartBody			
Maximum Occurs On	SYS-1\PartBody			
Total				3.4258e-
				007 J
Information				
Time	1. s			
Load Step	1			
Substep	1			
Iteration Number	1			
Integration Point Results				
Display Option		Averaged		
Average Across Bodies		No		

FIGURE 2 Model (B4) > Static Structural (B5) > Solution (B6) > Total Deformation

- **Total Deformation Analysis**

The total deformation results indicate that the wheel rim experiences **very minimal displacement** under the applied loading condition. The maximum deformation observed was:

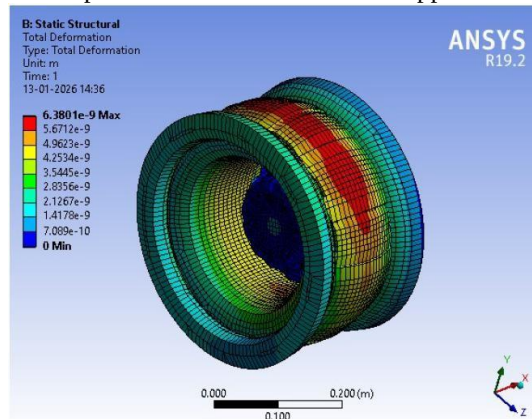
Maximum total deformation:

6.38×10^{-9} m

Average deformation:

2.02×10^{-9} m

The minimum deformation occurred at the fixed support regions, while the maximum deformation was observed at the outer rim flange, which is expected due to its relatively lower stiffness and greater distance from the constrained hub. The extremely small deformation values confirm that the rim possesses **high structural stiffness** and remains well within acceptable elastic limits under the applied load.



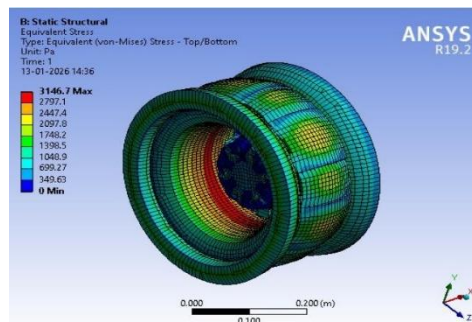
- **Equivalent (von Mises) Stress Distribution**

The von Mises stress distribution is a critical criterion for evaluating yielding in ductile materials such as aluminum alloys. The simulation results show:

Maximum equivalent stress: 3146.7 Pa (≈ 0.00315 MPa)

Average equivalent stress: 982.15 Pa

The maximum stress regions were concentrated around the inner rim and spoke junction areas, where stress concentration naturally occurs due to geometric discontinuities. However, when compared with the **tensile yield strength of aluminum alloy (280 MPa)**, the induced stress is **negligibly small**, resulting in a **very high factor of safety**. This clearly indicates that the wheel rim is **structurally safe against yielding** under the given loading condition.



- **Equivalent Elastic Strain**

The equivalent elastic strain values further support the stress results:

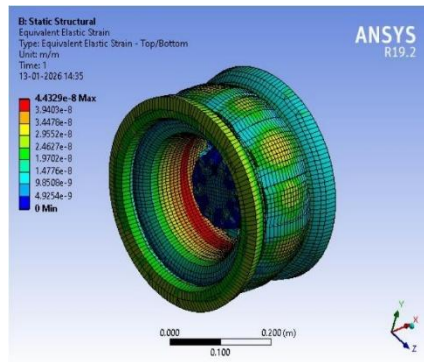
Maximum equivalent elastic strain:

4.43×10^{-8}

Average elastic strain:

1.52×10^{-8}

These strain values are extremely low and fall well within the elastic region of the material. No permanent deformation is expected, confirming that the rim will return to its original shape once the load is removed. This behavior is desirable for automotive wheel rims subjected to repeated loading cycles during service life.



- **Strain Energy Evaluation**

The strain energy stored in the structure provides insight into the overall load absorption capability of the rim:

Total strain energy:

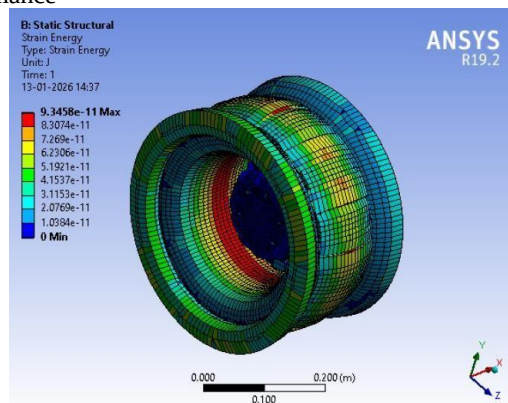
$$3.43 \times 10^{-7} \text{ J}$$

Maximum strain energy density:

$$9.35 \times 10^{-11} \text{ J}$$

The low strain energy values indicate that the applied load does not significantly stress the structure, and the rim operates well within its elastic capacity. This suggests efficient load distribution throughout the rim geometry

Discussion of Structural Performance



From the finite element results, it is evident that the aluminum alloy wheel rim demonstrates **excellent structural integrity** under static loading conditions. The deformation, stress, and strain values are significantly lower than the allowable limits of the material. The design is therefore **safe, robust, and suitable for automotive applications**.

However, the very low stress levels also suggest potential **scope for weight optimization**. Material reduction in low-stress regions could be explored in future work to improve fuel efficiency without compromising safety. Additionally, further studies involving **dynamic loading, fatigue analysis, and impact loading** are recommended to fully assess real-world performance.

- **Key Findings (Summary)**

- Maximum deformation is in the order of **nanometers**, indicating high stiffness.
- Von Mises stress is far below the yield strength of aluminum alloy.
- Elastic strain confirms purely elastic behavior.
- High factor of safety is achieved.
- Design is structurally safe but can be optimized for weight reduction.

9. CONCLUSION

In this study, the **structural and stress analysis of an automotive wheel rim** was successfully carried out using the **Finite Element Method (FEM)** in ANSYS Mechanical. A three-dimensional solid model of the wheel rim made of **aluminum alloy** was analyzed under static loading conditions to evaluate deformation, stress, strain, and strain energy behavior.

The finite element results revealed that the **maximum total deformation** experienced by the rim was extremely small and well within permissible limits, indicating high structural stiffness. The **maximum**

equivalent (von Mises) stress developed in the rim was significantly lower than the material's yield strength, confirming that the design is **structurally safe and free from yielding or failure** under the applied loading conditions. Overall, the results demonstrate that the aluminum alloy wheel rim exhibits **excellent structural integrity, durability, and reliability**, making it suitable for automotive applications. The FEM approach proved to be an effective and accurate method for predicting the mechanical behavior of wheel rims, thereby reducing the need for extensive physical testing during the design phase.

10. FUTURE SCOPE

Although the present study confirms the structural safety of the wheel rim under static loading, several areas can be explored for further improvement and extended analysis:

Weight Optimization:

Topology optimization techniques can be applied to reduce material in low-stress regions, leading to lighter wheel rims and improved vehicle fuel efficiency.

Fatigue Analysis:

Since wheel rims are subjected to cyclic loading during real-world operation, fatigue life prediction using S-N curves can provide a better understanding of long-term performance.

Dynamic and Impact Analysis:

Future studies may include dynamic loading conditions such as braking, cornering, pothole impact, and road irregularities to simulate real driving scenarios.

Thermal Analysis:

Coupled thermal-structural analysis can be performed to study the effect of heat generated due to braking on rim performance.

Material Comparison:

Comparative studies using different materials such as magnesium alloys, steel, or composite materials can be conducted to identify optimal alternatives.

Experimental Validation:

Physical testing of the wheel rim can be carried out to validate the numerical results obtained through finite element analysis.

11. REFERENCES

- [1] Zienkiewicz, O. C., Taylor, R. L., & Zhu, J. Z., *The Finite Element Method: Its Basis and Fundamentals*, 7th Edition, Elsevier, 2013.
- [2] Bathe, K. J., *Finite Element Procedures*, Prentice Hall, 2006.
- [3] Rao, S. S., *The Finite Element Method in Engineering*, 5th Edition, Butterworth-Heinemann, 2011. ANSYS Inc., *ANSYS Mechanical APDL Theory Reference*, Release 19.2, ANSYS Documentation.
- [4] Narayana, K. L., & Rao, G. V. Design and Analysis of Automotive Wheel Rim Using Finite Element Method *International Journal of Engineering Research and Technology (IJERT)* Vol. 3, No. 5, 2014.
- [5] Babu, M. S., & Prasad, D. S., "Structural Analysis of Alloy Wheel Rim Using FEM," *International Journal of Scientific & Engineering Research*, Vol. 6, No. 4, 2015.
- [6] Budynas, R. G., & Nisbett, J. K., *Shigley's Mechanical Engineering Design*, 10th Edition, McGraw-Hill, 2015.
- [7] Stearns, J., et al., Analysis of stress and strain distribution in a vehicle wheel: finite element analysis versus the experimental method. *The Journal of Strain Analysis for Engineering Design* 2005. 40(6): p. 513-523.
- [8] Liu, J.P., B.Y. Ye, and G. Li. Application of Finite Element Analysis in Automotive Wheel Design. in *Key Engineering Materials*. 2011. Trans Tech Publ.
- [9] Beigzadeh, S. and J. Marzbanrad, Automotive Wheel Optimization to Enhance the Ftg Life. *Automotive Science and Engineering*, 2018. 8(3): p. 2739-2758.
- [10] Das, S., Design and weight optimization of aluminum alloy wheel. *IJSRP* 2014. 4(6): p. 1-12.
- [11] Mahadik, S.S., Design and ANSYS analysis of components of wheel assembly of SAE car. *Int. J. Curr. Eng. Technol*, 2018. 8(02): p. 407-438.
- [12] Kara, A., et al., Design and weight optimization of aluminum alloy wheels. *Pamukkale University Journal of Engineering Sciences*, 2017. 23(8): p. 957-962.