IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 5, Issue 9, June 2025
Impact Factor: 7.67

Finite Element Analysis of Two Wheeler Rim By Using Composite Material

Dr J. Kingston Barnabas[1], Saran S[2], Dhivan M[3], Santhosh P[4], Vignesh S[5]
Department of Mechanical Engineering[1-5]
Anjalai Ammal Mahalingam Engineering College Kovilvenni, Tamilnadu, India

Abstract: *The cylindrical cam and follower mechanism is highly used in the packaging industry to obtain the sequential operation of material filling in the packaging machine. The occurrence of frictional wear between the cam and follower involves the reduction of cam life that eventually affects the quality of packing in package industry. In this work, an attempt is made to simulate the dynamic analysis with different material. Dynamic simulations are made for the cylindrical cam follower arrangement using a commercially available finite element code ANSYS. The simulation of analysis is carried out to predict the deformation of cam. The simulation of dynamic analysis is also conducted to investigate the effect of vibration on the performance of cam follower mechanism during a packing operation.*

Keywords*:* Rim wheel, Finite Element Analysis, ANSYS, Dynamic Analysis

I. INTRODUCTION

In every modern vehicle it has part which is fully integral from the bearing. It is known as wheel development. In vehicle running condition, wheels are encountered by the dynamic loads and static loads by the help of the suspension system. In the vehicles, wheels are used for move the vehicle while it supporting a mass. A two-wheeler wheel rim is shown in the figure 1. It is rotates in their own axis by the circular shape. The rolling of the axle friction which overcomes from the facilitating motion is known as wheel. The external force or gravity needs for moment of the wheel its own axis. The flywheel, steering wheel and ships wheel are also works at the same mechanism. The important thing for wheel is economy and safety.

II. LITRATURE REVIEW

Design and Analysis of Aluminium Alloy Wheel using PEEK Material PEEK 90HMF20 was identified as a superior alternative to traditional aluminum alloy due to its lightweight nature, high strength, and durability.(2013)

Structural And Fatigue Analysis Of Two-Wheeler Lightweight Alloy Wheel.

The study reinforced the importance of fatigue analysis in wheel design, demonstrating that lighter, high-strength materials can significantly enhance vehicle efficiency.(2013)

Influence of Material and Spoke Pattern on the Performance of Automotive Wheels.

This paper supports your study on FEA of two-wheeler rims by reinforcing that composite materials provide better mechanical performance than conventional alloys.(2019)

Design and Material Optimization of an Automobile Wheel Rim by Finite Element Analysis

Design Optimization of Alloy Wheels Based on a Dynamic Cornering Fatigue Test Using Finite Element Analysis and Multi- Additional Sampling of Efficient Global Optimization: This study proposes a novel method for alloy wheel design under cornering fatigue conditions, achieving a 20.181% decrease in optimal thickness and a 3.176% reduction in width compared to the initial design.(2020)

Design and Analysis of Wheel Rim With Magnesium Alloys (Zk60a) By Using Solidworks and Finite Element Method

In summary, the journal outcome indicates that the design and analysis using SolidWorks and FEA provided a strong case for using magnesium alloy ZK60a in automotive wheel rims. The approach not only achieved a reduction in weight but also maintained the structural performance required for safe and effective use, paving the way for future experimental studies and potential industrial application.(2016)

III. METHODOLOGY

The methodology followed in this project was designed to systematically develop, analyze, and compare the performance of a motorcycle wheel rim made from aluminum alloy and a hybrid composite fiber material. The process began with data collection through reverse engineering, wherein an existing alloy wheel rim from a two-wheeler was disassembled and its critical dimensions were measured. These measurements were used as the foundation for creating a detailed 3D model using CATIA V5, a powerful CAD software. During the modeling phase, best practices such as proper sketch constraining, structured feature organization, and naming conventions were followed to ensure design clarity and ease of modification

Following the modeling phase, two different materials— aluminum alloy and hybrid fiber composite—were selected for analysis. Material properties such as Young's modulus, Poisson's ratio, and density were defined for both materials based on existing literature and material databases. The aluminum alloy was chosen for its well-established mechanical strength and durability, whereas the hybrid composite fiber was selected for its lightweight nature and good vibration damping characteristics.

Finite Element Analysis (FEA) was conducted using ANSYS to evaluate the structural performance of the wheel rim under operational conditions. The 3D CAD model was imported into ANSYS via the STEP format, and mesh generation was carried out using tetrahedral elements. Boundary conditions were applied by fixing specific faces and applying a rotational velocity of 250 rad/s along with a radial load of 2874 N. Static structural analysis was performed to obtain results such as total deformation, equivalent stress, shear stress, and strain for both materials. These analyses provided insights into how each material behaves under loading conditions typically experienced during vehicle operation.

In parallel with the simulation work, experimental fabrication of the composite rim was undertaken. Two layup trials were conducted using wet layup techniques. In the first layup, fiberglass sheets were used to evaluate the mould design and layup strategy. The second layup involved a hybrid approach using glass fiber and carbon tape to replicate the performance of a hybrid fiber rim. The moulds were prepared, cleaned, and waxed, followed by layup, vacuum bagging, and room- temperature curing. Post-curing, the composite parts were demoulded using a freeze-and-heat method to overcome issues of adhesion and part removal.

After fabrication, the composite rims were inspected for surface finish, fiber conformity, resin distribution, and defects. The simulation results were then compared with the expected performance of the fabricated parts to draw conclusions about material suitability. The overall methodology allowed for a comprehensive comparison between traditional aluminum alloy rims and advanced composite fiber rims in terms of structural behavior and practical manufacturability.

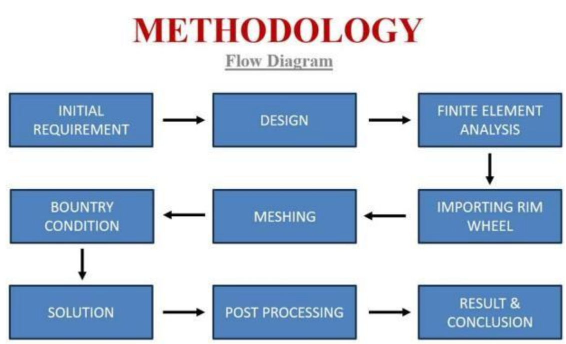

Fig 1. Methodology Flow Diagram

IV. MODELING OF RIM WHEEL

The modeling of the motorcycle wheel rim was carried out using CATIA V5, a widely used computer-aided design (CAD) software known for its powerful solid modeling capabilities. The reverse engineering process provided the dimensional data required to develop an accurate 3D representation of the existing alloy wheel rim. The modeling began with creating the base profile sketches, which were dimensioned and fully constrained to capture the geometry precisely. Features such as the rim contour, bolt holes, and bead seat areas were carefully constructed using a

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 5, Issue 9, June 2025
Impact Factor: 7.67

combination of pad, pocket, and revolve operations. Special attention was paid to the specification tree, where features were logically grouped, and meaningful names were assigned to make the design history easy to follow and modify. Planes and geometrical sets were organized in a way that allowed for efficient navigation and reference. Construction elements such as points, axes, and surfaces were created and stored in separate geometrical sets to ensure clarity and reusability. The entire model was developed with design intent in mind, allowing for future updates or variations. Once the model was finalized, it was exported in STEP format for structural analysis using ANSYS. This CAD model served as the basis for all subsequent simulations and helped bridge the gap between theoretical study and practical evaluation.

Fig 2. 3D Model of Rim Wheel

V. FINITE ELEMENT ANALYSIS OF RIM WHEEL

The analysis of the modeled motorcycle wheel rim was conducted using ANSYS, a comprehensive finite element analysis (FEA) software. After developing the 3D model in CATIA V5, it was exported in STEP format and imported into ANSYS Workbench for simulation. The first step in the analysis involved generating a high-quality mesh using tetrahedral elements to discretize the model into small finite elements. A finer mesh was applied to areas expected to experience high stress concentrations, such as bolt holes and the bead seat area, to improve result accuracy. Material properties were then defined for two materials: Aluminum Alloy and Hybrid Composite Fiber. These included Young's modulus, Poisson's ratio, and density, all derived from material data sheets and literature sources.

To simulate realistic working conditions, appropriate boundary conditions were applied. The rim was constrained on specific faces to represent its connection to the axle hub, and a radial load of 2874 N was applied to replicate operational loading. Additionally, a rotational velocity of 250 rad/s was applied to account for dynamic effects experienced during vehicle motion. The solver used a static structural analysis approach to evaluate how the rim deforms and responds under these loads.

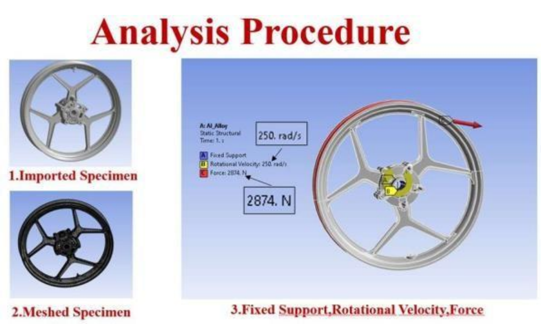

Fig 3. ANSYS Analysis Procedure

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 5, Issue 9, June 2025
Impact Factor: 7.67

Aluminium Alloy :

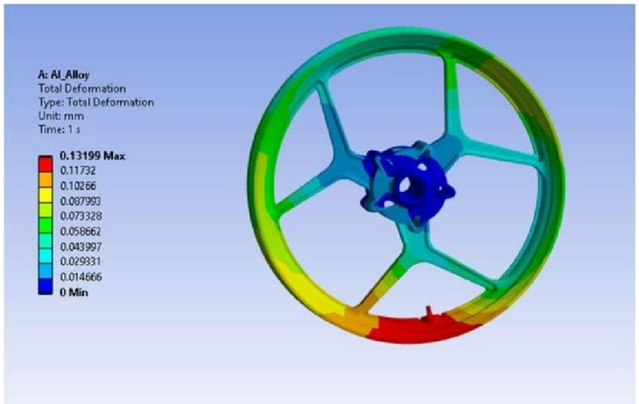

Fig 4. Total Deformation Of Al Alloy

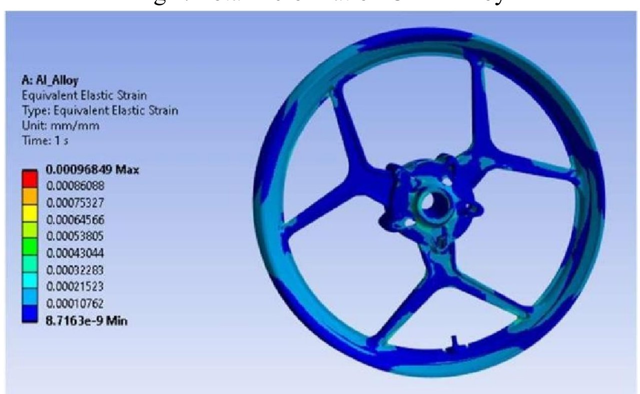

Fig 5. Elastic Strain Of Al Alloy

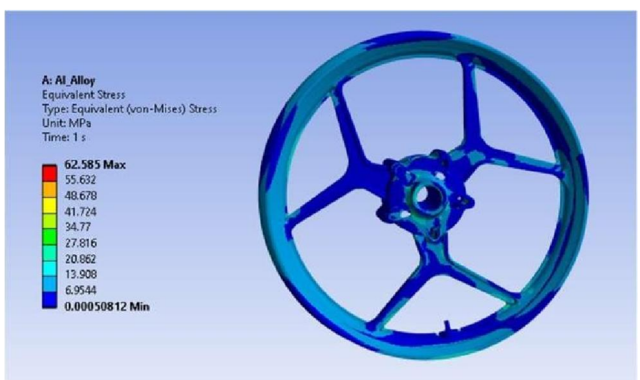

Fig 6. Equivalent Stress Of Al Alloy

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 5, Issue 9, June 2025
Impact Factor: 7.67

Fig 7. Shear Stress Of Al Alloy

Hybrid Composite Fiber :

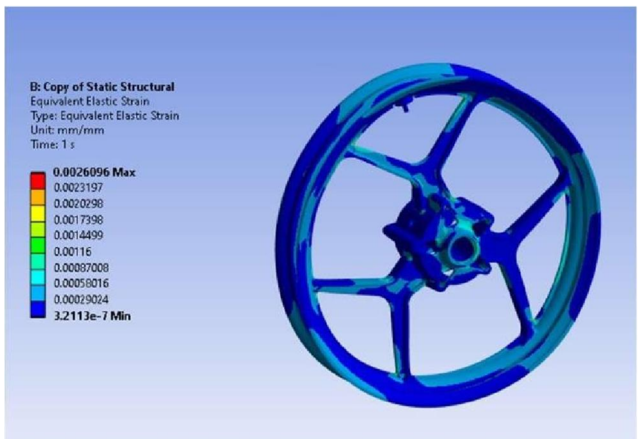

Fig 8. Elastic Strain Of Hybrid Composite

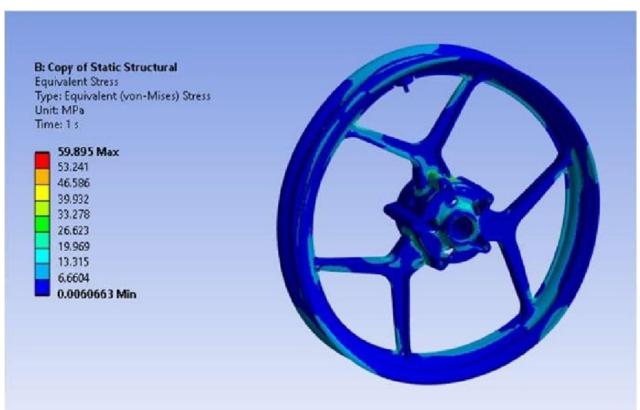

Fig 9. Equivalent Stress Of Hybrid Composite

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
Volume 5, Issue 9, June 2025
ISSN: 2581-9429
Impact Factor: 7.67

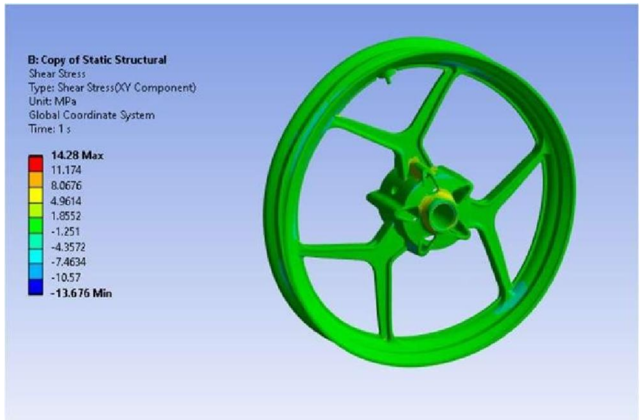

Fig 10. Shear Stress Of Hybrid Composite

Type	Total Deformation	Equivalent Elastic Strain	Equivalent (von-Mises) Stress	Shear Stress
Minimum	0. m	2.3794e-007 m/m	4609.9 Pa	-1.3288e+007 Pa
Maximum	2.4614e-004 m	2.248e-003 m/m	5.1244e+007 Pa	1.6512e+007 Pa
Average	9.9522e-005 m	2.3972e-004 m/m	4.6645e+006 Pa	12524 Pa

Fig 11. Result

Material	Deformation (mm)	Stress(MPa)	Shear stress(MPa)
Al-Alloy	0.13	62	24.9
Hybrid Fiber	0.28291	59	14

Fig 12. Comparative Result

VI. RESULTS

The structural analysis yielded critical insights into the mechanical behavior of the motorcycle wheel rim when subjected to defined loading and rotational conditions. Two materials—Aluminum Alloy and Hybrid Composite Fiber— were evaluated using finite element analysis in ANSYS. The simulation outputs revealed key differences in performance between the two materials. The aluminum alloy rim exhibited a maximum total deformation of 0.13 mm, with a maximum von Mises stress of 62 MPa and shear stress of 24.9 MPa. These values confirm the high structural stiffness and stress-bearing capacity of aluminum alloy, making it suitable for heavy-duty and high-load applications.

In contrast, the hybrid composite fiber rim demonstrated a higher total deformation of 0.28291 mm, indicating greater flexibility under load. However, the maximum stress of 59 MPa remained within safe operational limits, and the shear stress was considerably lower at 14 MPa, which can be advantageous in reducing fatigue-related failures. The equivalent elastic strain was also higher in the composite rim, reflecting its enhanced ductility and energy absorption capabilities. These results suggest that while aluminum alloy excels in terms of rigidity and stress resistance, the hybrid fiber material offers a lightweight alternative with acceptable strength characteristics and improved vibration damping.

Overall, the results highlight a trade-off between structural stiffness and weight savings. Aluminum alloy rims provide better resistance to deformation and higher stress tolerance, whereas hybrid composite rims show promise in applications where reduced weight, moderate strength, and energy absorption are prioritized.

IJARSCT

International Journal of Advanced Research in Science, Communication and Technology

International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal

ISSN: 2581-9429

Volume 5, Issue 9, June 2025

Impact Factor: 7.67

VII. CONCLUSION

A structural analysis of a wheel rim was conducted using two materials: Aluminum Alloy and a Hybrid Composite Fiber, focusing on deformation, stress, and shear stress characteristics under loading conditions.

The results showed that the Aluminum Alloy experienced lower deformation (0.13 mm) compared to the Hybrid Fiber (0.28291 mm), indicating better rigidity. Additionally, the aluminum alloy endured a slightly higher maximum stress of 62 MPa, while the hybrid fiber withstood 59 MPa, both within safe operating limits. However, in terms of shear stress, the aluminum alloy exhibited 24.9 MPa, significantly higher than the hybrid fiber's 14 MPa.

These findings suggest that Aluminum Alloy provides superior structural stiffness and higher shear resistance, making it suitable for high-load applications. On the other hand, the Hybrid Composite Fiber, with its lighter weight and acceptable stress performance, could be a viable alternative in applications where weight reduction and moderate strength are priorities.

Overall, material selection should depend on the design requirement — Al-Alloy for strength-critical designs and Hybrid Fiber for lightweight, cost-effective alternatives.

Comparing with these result Hybrid Fiber composite material crating minimum amount of failure by using this material instead of Al-alloy the life time cycle will be increasing

REFERENCES

[1.] Fatigue Analysis of Aluminium Alloy Wheel Under Radial Load, International Journal of Mechanical and Industrial Engineering (IJMIE), ISSN No. 2231 –6477, Vol-2, Issue-1, 2012.

[2.] "An analysis of stress and displacement distribution in a rotating rim subjected to pressure and radial loads" by P.C.Lam and T.S.Srivastam.

[3.] THE TIRE AND RIM ASSOCIATION, INC (1996), "50 Drop Centre Rim Contours", J (ISO) Contour for 14,15,16,18 and 20 diameter Designation, pp7.05.

[4.] Stress Anlasis Of Wheel Rim International Journal of Mechanical Engineering and Research Volume 1 Issue 1 (Page, 34-37), ISSN: 2277-8128.

[5.] CATIA V5 Fundamentals, www. http://handbook5.com/c/catia-v5-fundamentals-w3524.html.

[6]. "Finite Element Analysis of the Classic Bicycle Wheel" by Andrew D. Hartz at Rose-Hulman Institute of Technology ME522 Finite Element Analysis conference July 18, 2002. [7]. "Fatigue Analysis of an Automobile Wheel Rim" Case study

[8]. IS 10694-1 (2009): Automotive Vehicles - Rims – General Requirements, Part 1: Nomenclature, Designation, Marking and Measurement [TED 7: Automotive Tyres, Tubes and Rims. [9]. "Analyze the Effect of Camber Angle on Fatigue Life of Wheel Rim of Passenger Car by Using Radial Fatigue Testing" by Sunil N. Yadav and N.S Hanamapure at International Journal of Engineering Science and Innovative Technology (IJESIT) Volume 2, Issue 5, September 2013, pp-231-239, ISSN:2319-5967.

[10]. "Design and Analysis of Spiral Wheel Rim for Four Wheeler" by S.Ganesh and Dr. P. Periyasamy at the International Journal of Engineering and Science (IJES), Volume-3, Issue-4, Pages-29-37, 2014, ISSN (e): 2319 – 1813 ISSN (p): 2319 – 1805.

[11]. "Design and development of aluminium alloy wheels", by M. V. Prabha and Pendyala Veera Raju at the International Journal of Advanced Science, Engineering and Technology, ISSN 2319-5924, volume-1, Issue 2, 2012, pp 55-60.

[12]. "Simulation of dynamic cornering fatigue test of a steel passenger car wheel", by Xiaofeng Wang and Xiaoge Zhang at International Journal of Fatigue Volume-32, 2010 , pp 434– 442.

[13]. "Understanding the Influence of Pressure and Radial Loads On Stress and Displacement Response of a Rotating Body: The automobile wheel" , by J. Stearns, T. S. Srivatsan,
X. Gao and P. C. Lam, Hindawi Publishing Corporation International Journal of Rotating Machinery, Volume 2006, Article ID 60193, Pages 1–8.

[14]. "Finite element simulation of wheel impact test", by C.L. Chang, S.H. Yang , Journal of achievements in materials and manufacturing engineering, volume 28, Issue 2, pp-167-170, June 2008