© April 2026| IJIRT | Volume 12 Issue 11 | ISSN: 2349-6002

Static And Modal Analysis of Automotive Wheel Hubs

Mr. Shubham[1,] Mr. Rahul R. Kulkarni[2], Mr. Pranay A. Makasare[3], Mr. Laxman V. Kamble[4]
[1]*Pursuing Masters of Engineering (Design) at, Siddhant College of Engineering, Sudumbre – 412109, Pune, Maharashtra, Indian*
[2,3,4]*Department of Mechanical Engineering, Siddhant College of Engineering, Sudumbre – 412109, Pune, Maharashtra, Indian*

Abstract- This project investigates the structural and dynamic behavior of automotive wheel hubs through comprehensive static, modal, and topological analyses. Wheel hubs experience complex loading due to vehicle weight, braking forces, and road-induced vibrations, making their design critical for safety, durability, and overall vehicle performance. In this study, detailed 3D hub models are developed and evaluated using finite element analysis to determine stress distribution, deformation characteristics, and natural frequencies under realistic operating conditions. Multiple materials—including steel, aluminum alloys, and lightweight alternatives—are compared to identify options offering superior strength-to-weight performance and improved vibration resistance. Furthermore, topology optimization is applied to reduce mass while maintaining required stiffness and structural integrity. The results contribute to the development of optimized, lightweight wheel hub designs that enhance efficiency, reliability, and safety in modern automotive systems.

I. INTRODUCTION

The wheel hub is a critical load-bearing component in automotive systems, responsible for transmitting forces between the suspension, braking system, and rotating wheel assembly. Its structural integrity governs essential vehicle performance parameters such as durability, ride comfort, vibration characteristics, and overall dynamic stability. As modern vehicles continue to pursue higher safety standards, improved NVH (Noise, Vibration, Harshness) performance, and enhanced powertrain efficiency, the mechanical behavior of the wheel hub under different loading conditions has gained significant research interest. Static strength, fatigue resistance, and modal stability form the foundation of wheel hub design, as failure in any of these domains can result in catastrophic wheel detachment or long-term degradation under cyclic loads.

In the current automotive industry, the push for lightweighting has intensified due to increasing demands for fuel efficiency, extended EV driving range, and reduced carbon emissions. Manufacturers are progressively shifting toward advanced materials such as high-strength aluminum alloys, titanium alloys, and composite-metal hybrids to replace conventional steel hubs. Simultaneously, the adoption of digital engineering methods—Finite Element Analysis (FEA), multi-material optimization, advanced modal characterization, and topology optimization—has transformed the way structural components are designed. Instead of relying solely on empirical design iterations, engineers now leverage computational frameworks to achieve high stiffness-to-weight ratios while maintaining structural safety margins.

Wheel hubs exist in various configurations depending on vehicle type, functional architecture, and load distribution requirements. These include forged hubs for heavy-duty applications, cast aluminum hubs for passenger cars, integrated hub-bearing units for compact assemblies, and additive- manufactured variants for performance-focused systems. Regardless of type, the fundamental working mechanism involves transferring vertical, longitudinal, and lateral forces from the wheel to the suspension while also ensuring smooth rotation through the bearing interface. Under these conditions, the hub experiences complex stress patterns—bending loads during cornering, radial loads under acceleration and braking, and vibrational excitation from road irregularities—making static analysis and modal analysis essential for accurate performance prediction.

Fig. No. 1 Exploded Wheel hub

Despite advancements in computational tools, challenges remain in achieving optimal trade-offs between structural rigidity, lightweight performance, and manufacturability. Modal instability, resonance-induced fatigue failures, local stress concentrations, and material-dependent deformation behaviors continue to influence wheel hub design. Industry trends indicate growing interest in integrating multi-objective optimization techniques with advanced materials to overcome these limitations. Topology optimization, in particular, has emerged as a transformative approach, enabling engineers to generate organic, high-performance geometries that conventional design practices cannot achieve. With additive manufacturing becoming more feasible, these optimized geometries are now increasingly realizable in production.

Given this evolving landscape, a comprehensive investigation combining static and modal analysis with systematic material comparison and topology-based design improvement is necessary. Such a study provides insight into the influence of material selection, structural geometry, and vibrational characteristics on hub performance. By examining both traditional and lightweight materials and integrating optimization-driven redesign, the research aims to contribute meaningfully to modern automotive lightweighting strategies, structural reliability enhancement, and performance-driven component engineering.

II. LITERATURE REVIEW

1. Cang, P., Zhang, X., & Cui, W. (2023)

Cang et al. analyze an automotive wheel hub using combined static and modal FEA to study stress distribution, deformation, and vibration behavior. They show how geometric parameters like fillet radius and spoke thickness significantly influence stress concentrations and modal frequencies. The study compares aluminum and magnesium alloys, demonstrating 15– 35% mass reduction with acceptable stress and dynamic margins. Modal results indicate safe separation from road/engine excitation frequencies. Although load cases are simplified and experimental validation is limited, the paper offers a strong methodology for integrated static–modal evaluation. It also highlights key trade-offs between lightweighting and structural reliability.

2. Jia, J., & Sun, X. (2021)

Jia and Sun apply density-based topology optimization to an ATV wheel hub to minimize mass while maintaining strength under multiple load cases. Their TO model incorporates manufacturing filters and symmetry constraints to ensure practical, castable shapes. Optimized designs achieve 10–30% weight reduction while keeping stresses within allowable limits. The authors also refine the TO output into CAD and verify it through FEA, addressing common TO issues like stress raisers. Limitations include fatigue not being directly incorporated, and dependence on accurate load case definition.

The study provides a clear TO workflow suitable for lightweight hub development.

3. Almonti, D., et al. (2025)

Almonti et al. present a manufacturing-aware topology optimization process for a steering knuckle that is directly applicable to wheel hub design. Their workflow integrates multi-load-case TO, sustainability assessment, and additive- manufacturing-compatible constraints. Optimized designs show 20–40% mass reduction with improved stiffness-to-mass ratio. The study progresses from optimization to physical casting and experimental validation, proving feasibility for real applications. While long-term fatigue testing is limited, the work demonstrates how TO and manufacturing constraints can be merged in a practical design loop. It provides valuable lessons for ensuring TO results remain manufacturable and structurally sound.

4. Manasa, C. M., et al. (2025)

Manasa et al. use a reinforcement learning (RL) framework combined with topology optimization to generate optimized wheel hub geometries. Their RL agent iteratively removes material based on reward signals related to compliance, mass, and stress limits. The method produces manufacturable and competitive designs faster than classical TO alone. FEA validation shows that RL solutions maintain structural integrity while achieving significant mass reduction. Limitations include high training costs and sensitivity to reward design. The work demonstrates the potential of AI-assisted generative design for complex automotive components.

5. Long, M., et al. (2024)
Long and co-authors perform a combined numerical–experimental study on lightweight wheel hub and rim structures. Using parametric FE models and genetic algorithms, they generate optimized designs balancing weight, stiffness, and fatigue performance. Prototype testing validates simulation predictions and highlights real-world effects such as manufacturing tolerances and material variability. Results confirm that mass can be reduced significantly without compromising durability. The study also shows that rim–hub interaction strongly influences stress behavior. While material options are limited, the design-to-testing workflow is highly relevant for practical hub development.

6. Fang, Y., et al. (2024)
Fang et al. assess the structural reliability of wheel hubs manufactured from TB6 titanium alloy using additive manufacturing. They analyze AM-specific issues such as anisotropy, porosity, and residual stresses through both FEA and physical testing. Results show that TB6 hubs achieve high strength and good impact resistance but require post-processing like HIP and machining for fatigue reliability. Probabilistic analysis highlights variability in AM defects and their impact on safety margins. While large-scale production challenges remain, the study demonstrates the feasibility of AM titanium
hubs for high-performance applications. It also provides practical guidance for quality control and inspection.

7. Zhu, S., et al. (2025)
Zhu et al. introduce a sketch-guided topology optimization method that incorporates designer input directly into the TO process. Sketch-based constraints help steer the optimization toward manufacturable and stylistically acceptable designs while maintaining high structural performance. The method improves stiffness-to-mass ratios compared to unconstrained TO and avoids disconnected or unrealistic features. It also generates multiple valid design options, aiding collaborative engineering. Though sketch interpretation introduces subjective variability, the framework enhances TO practicality. The approach is well-suited for wheel hub designs requiring preserved interfaces and controlled load paths.

8. Qi, T., et al. (2024)
Qi et al. perform topology optimization on a ratchet compensation structure using periodic constraints relevant to rotating components like wheel hubs. Their use of cyclic symmetry prevents unbalanced material removal and produces dynamically stable designs. Optimization results show >10% weight reduction and significant stress improvements. They also emphasize postprocessing and smoothing steps to create manufacturable geometries. The comparison of single-period and multi-period constraints demonstrates how boundary conditions affect optimized layouts. The study highlights the importance of periodic boundary handling in rotating-part optimization.
Novelty of work: This work introduces a unique hybrid methodology that combines topology optimization, lightweight material selection, and real-world EV loading conditions to redesign a structurally efficient wheel hub. Unlike previous studies that focus only on geometry or material improvements, this research integrates static, modal, and fatigue analyses to ensure performance under dynamic service loads. The study also evaluates manufacturing-feasible optimized designs, bridging the gap between simulation and practical production. Overall, the novelty lies in delivering a lightweight, high- strength, and vibration-stable wheel hub design using a comprehensive and industry-oriented optimization workflow.

III. PROBLEM STATEMENT

- Wheel hubs in automobiles experience complex static and dynamic loads, requiring high strength, stiffness, and vibration stability.
- Traditional hub designs often use heavier materials and non-optimized geometries, increasing un-sprung mass and reducing vehicle efficiency.
- With the industry shifting toward lightweight and

high- performance components, there is a need to analyze and redesign wheel hubs using advanced simulation methods.

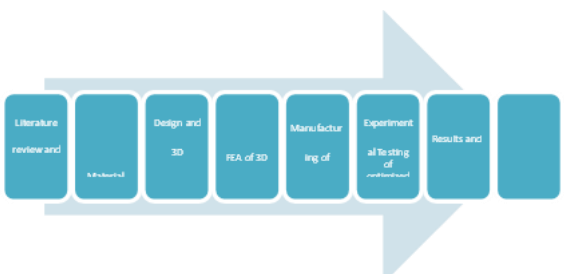
- Therefore, this project aims to evaluate the structural and modal behavior of wheel hubs, compare suitable materials, and apply topology optimization to develop a lighter yet structurally reliable hub design.

IV. OBJECTIVES

To analyze, compare, and optimize the design of an automotive wheel hub by performing static and modal analysis using finite element methods, evaluating different materials, and applying topology optimization to develop a lightweight, structurally efficient, and vibration-stable wheel hub suitable for modern automotive applications. The primary objectives are:

- Design and 3D modelling of wheel hubs.
- Analysis of wheel hubs of different materials.
- Topological Optimization of wheel hub
- Experimentation and validation of final optimized wheel hub.

V. RESEARCH METHODOLOGY



VI. FINITE ELEMENT ANALYSIS

The finite element method (FEM), is a numerical method for solving problems of engineering and mathematical physics. Typical problem areas of interest include structural analysis, heat transfer, fluid flow, mass transport, and electromagnetic potential. The analytical solution of these problems generally require the solution to boundary value problems for partial differential equations. The finite element method formulation of the problem results in a system of algebraic equations. The method yields approximate values of the unknowns at discrete number of points over the domain. To solve the problem, it subdivides a large problem into smaller, simpler parts that are called finite elements.

In the first step, the element equations are simple equations that locally approximate the original complex equations to be studied, where the original equations are often partial differential equations (PDE). The process, in mathematical language, is to construct an integral of the inner product of the residual and the weight functions and set the integral to zero. In simple terms, it is a procedure that minimizes the error of approximation by fitting trial functions into the PDE. The residual is the error caused by the trial functions, and the weight functions are polynomial approximation functions that project the residual. The process eliminates all the spatial derivatives from the PDE, thus approximating the PDE locally with

- A set of algebraic equations for steady state problems,
- A set of ordinary differential equations for transient problems.

These equation sets are the element equations. They are linear if the underlying PDE is linear, and vice versa. Algebraic equation sets that arise in the steady state problems are solved using numerical linear algebra methods, while ordinary differential equation sets that arise in the transient problems are solved by numerical integration using standard techniques such as Euler's method or the Runge-Kutta method.

FEM is best understood from its practical application, known as finite element analysis (FEA). FEA as applied in engineering is a computational tool for performing engineering analysis. It includes the use of mesh generation techniques for dividing a complex problem into small elements, as well as the use of software program coded with FEM algorithm. In applying FEA, the complex problem is usually a physical system with the underlying physics such as the Euler-Bernoulli beam equation, the heat equation, or the Navier-Stokes equations expressed in either PDE or integral equations, while the divided small elements of the complex problem represent different areas in the physical system.

In present research for analysis ANSYS (Analysis System) software is used. Basically, its present FEM method to solve any problem. Following are steps in detail

1. Geometry

2. Discretization (Meshing)
3. Boundary condition
4. Solve (Solution)
5. Interpretation of results

Workbench contain analysis of different types namely static, modal, harmonic, explicit dynamics, CFD, ACP tool post, CFX, topology optimization etc. as per problem defined.

Step 1: Details of material namely copper, steel, grey cast iron, composite material, fluid domain material is defined in engineering data. i.e. ANSYS default material is structural steel.

Step 2: Import of geometry created in any CAD software namely CATIA, PRO E, SOLIDWORK, INVENTOR etc. in geometry section. If any correction is to be made it can be created in geometry section in Design modeller or space claim. Step 3: In model section after import of component

- Material is assigned to component as per existing material
- Connection is checked in contact region i.e. bonded, frictionless, frictional, no separation etc. for multi body components.
- Meshing or discretization is performed i.e. to break components in small pieces (elements) as per size i.e. preferably tetra mesh and hexahedral mesh for 3D geometry and for 2 D quad or tria are generally preferred.

Step 4: Boundary condition are applied as per analysis namely in fixed support, pressure, force, displacement, velocity as per condition.

Step 5: Now problem is well defined and solve option is selected to obtain the solution in the form of equivalent stress, strain, energy, reaction force etc.

Static Structural Analysis of Wheel Hub (MS):

Geometry:

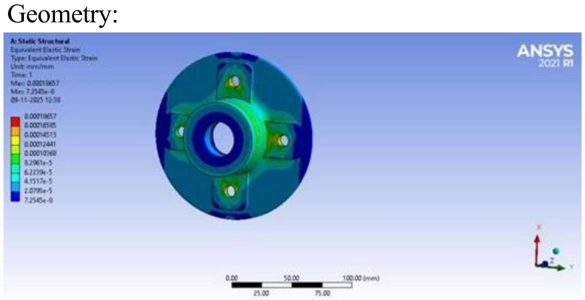


Fig. No. 2 Geometry of the MS wheel hub.

Material Properties:

Properties of Outline Row 3: Structural Steel

	A	B	C
1	Property	Value	Unit
2	Material Field Variables	Table	
3	Density	7850	kg m^-3
4	Isotropic Secant Coefficient of Thermal Expansion		
5	Coefficient of Thermal Expansion	1.2E-05	C^-1
6	Isotropic Elasticity		
7	Derive from	Young's Modulus and Poiss...	
8	Young's Modulus	2E+11	Pa
9	Poisson's Ratio	0.3	
10	Bulk Modulus	1.6667E+11	Pa
11	Shear Modulus	7.6923E+10	Pa

Table No. 1 Material Properties of structural steel.

Meshing:

Meshing is an integral part of the engineering simulation process where complex geometries are divided into simple elements that can be used as discrete local approximations of the larger domain. The mesh influences the accuracy, convergence and speed of the simulation. Furthermore, since meshing typically consumes a significant portion of the time it takes to get simulation results, the better and more automated the meshing tools, the faster and more accurate the solution. ANSYS provides general purpose, high-performance, automated, intelligent meshing software which produces the most appropriate mesh for accurate, efficient multiphysics solutions — from easy, automatic meshing to highly crafted mesh. Methods available cover the meshing spectrum of high- order to linear elements and fast tetrahedral and polyhedral to high-quality hexahedral and Mosaic.

Equivalent Elastic strain:

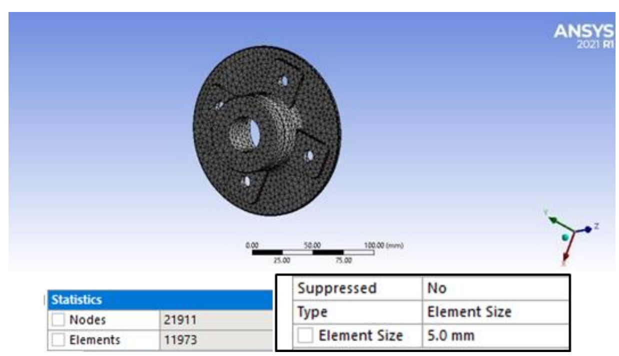


Fig. No. 3 Meshing details of the MS wheel hub.

Boundary Conditions:

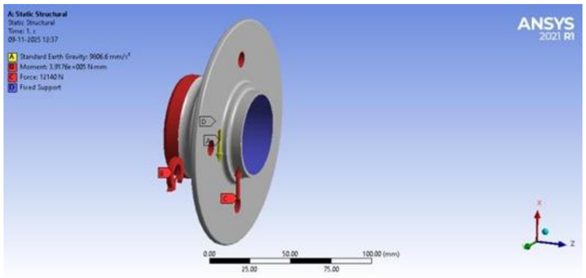


Fig. No. 4 Boundary conditions for the structural analysis of the MS wheel hub.

Results:

Total Deformation:

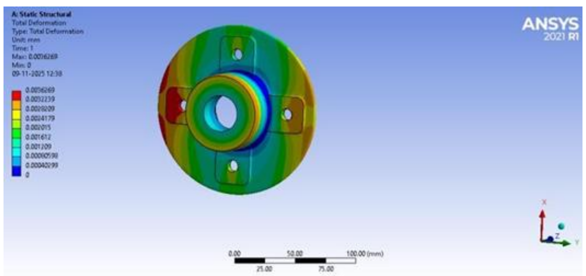


Fig. No. 5 Total deformation of the MS Wheel Hub.

Equivalent Stress:

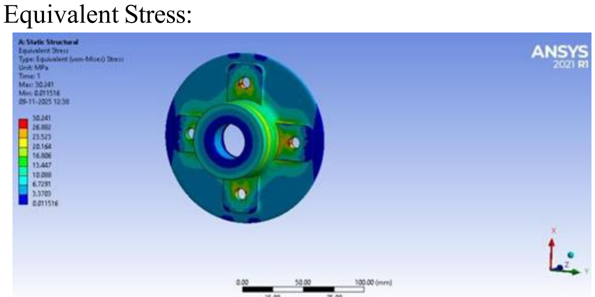


Fig. No. 6 Equivalent stress of Structural steel wheel hub.

Fig. No.7 Equivalent elastic strain of Structural steel wheel hub.

Life:

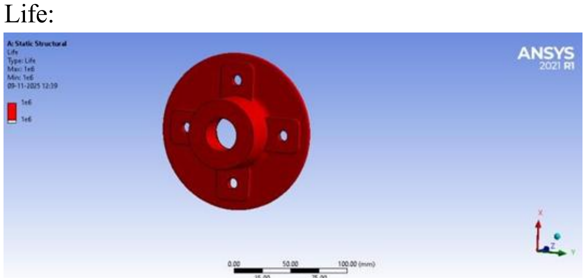


Fig. No. 8 Life of structural steel wheel hub.

Static Structural Analysis of Wheel Hub (Aluminium Alloy):

Geometry:

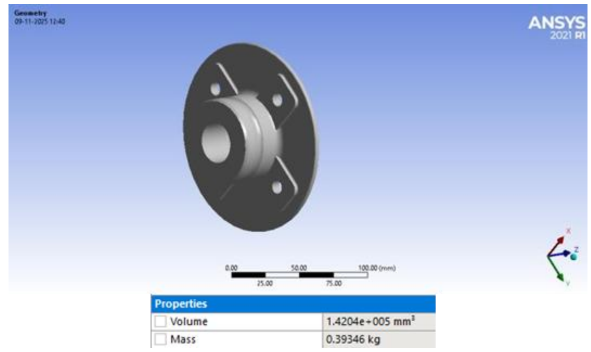


Fig. No. 9 Geometry of the aluminium alloy wheel hub.

Material Properties:

Properties of Outline Row 3: Aluminum Alloy

	A	B	C
1	Property	Value	Unit
2	Material Field Variables	Table	
3	Density	2770	kg m^-3
4	Isotropic Secant Coefficient of Thermal Expansion		
5	Coefficient of Thermal Expansion	2.3E-05	C^-1
6	Isotropic Elasticity		
7	Derive from	Young's Modulu...	
8	Young's Modulus	7.1E+10	Pa
9	Poisson's Ratio	0.33	
10	Bulk Modulus	6.9608E+10	Pa
11	Shear Modulus	2.6692E+10	Pa
12	S-N Curve	Tabular	
16	Tensile Yield Strength	2.8E+08	Pa
17	Compressive Yield Strength	2.8E+08	Pa
18	Tensile Ultimate Strength	3.1E+08	Pa

Table No. 2 Material Properties of aluminium alloy.

Meshing:

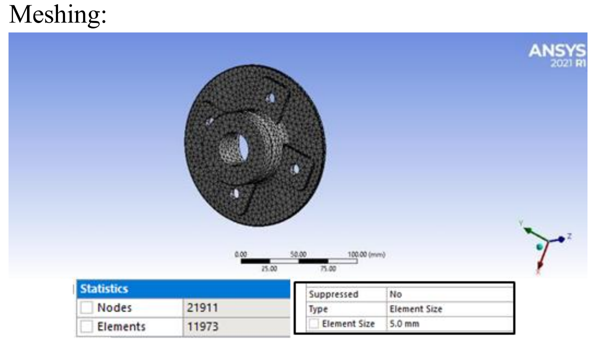


Fig. No. 10 Meshing details of the aluminium alloy wheel hub.

Boundary Conditions:

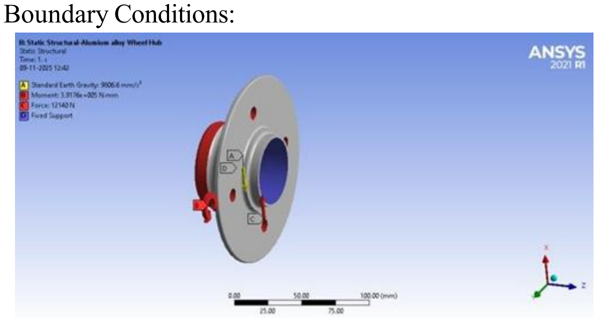


Fig. No. 11 Boundary conditions for the structural analysis of the Aluminium alloy wheel hub.

Results:

Total Deformation:

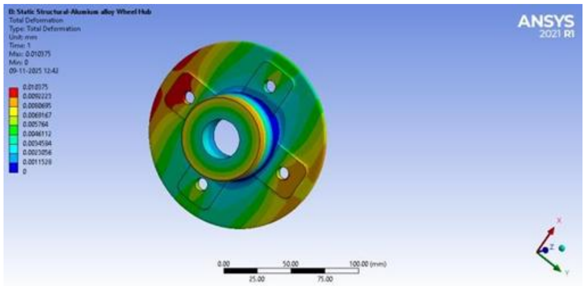


Fig. No. 12Total deformation of the aluminium alloy Wheel Hub.

Equivalent Elastic strain:

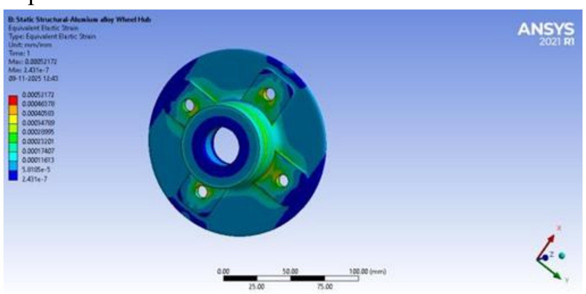


Fig. No. 14 Equivalent elastic strain of aluminium alloy wheel hub.

Life:

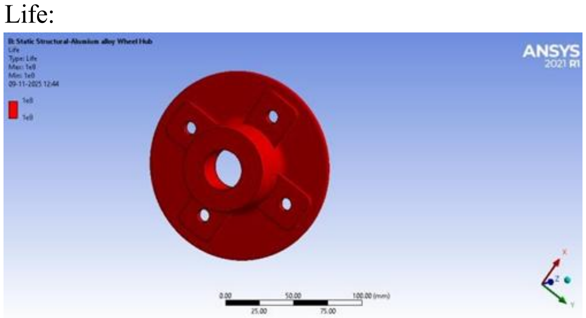


Fig. No. 15 Life of aluminium alloy wheel hub.

Static Structural Analysis of Wheel Hub (AISI 1035):

Geometry:

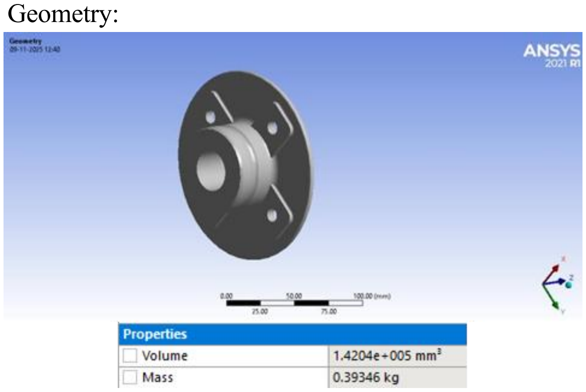


Fig. No. 16 Geometry of the wheel hub.

Equivalent Stress:

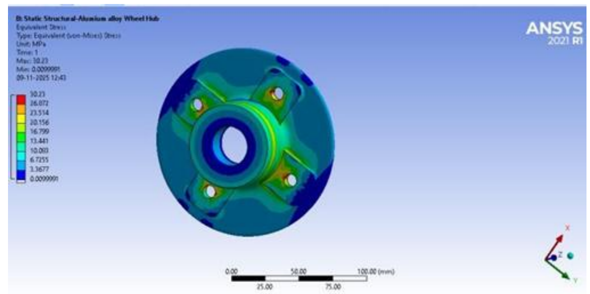


Fig. No. 13 Equivalent stress of aluminium alloy wheel hub.

Material Properties:

Properties of Outline Row 3: steel AISI 1035

	A	B	C
1	Property	Value	Unit
2	Material Field Variables	Table	
3	Density	7950	kg m^-3
4	Isotropic Secant Coefficient of Thermal Expansion		
5	Coefficient of Thermal Expansion	1.2E-05	C^-1
6	Isotropic Elasticity		
7	Derive from	Young's Modu...	
8	Young's Modulus	2E+05	MPa
9	Poisson's Ratio	0.29	
10	Bulk Modulus	1.5873E+11	Pa
11	Shear Modulus	7.7519E+10	Pa
12	Strain-Life Parameters		
20	S-N Curve	Tabular	
24	Tensile Yield Strength	2.5E+08	Pa
25	Compressive Yield Strength	2.5E+08	Pa
26	Tensile Ultimate Strength	4.7E+08	Pa

Table No. 3 Material Properties of AISI 1035.

Meshing:

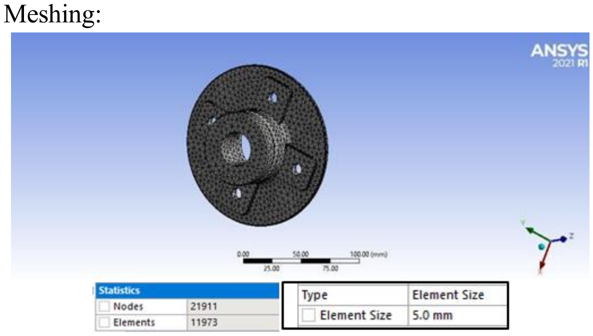


Fig. No. 17 Meshing details of the wheel hub.

Boundary Conditions:

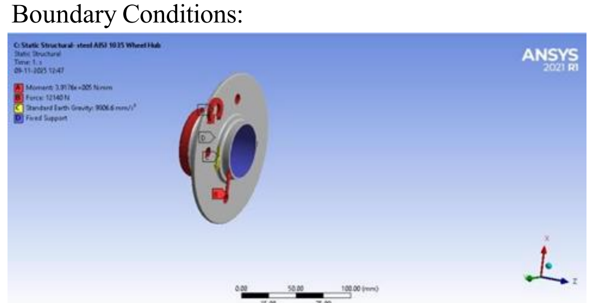


Fig. No. 18 Boundary conditions for the structural analysis of the wheel hub.

Results:

Total Deformation:

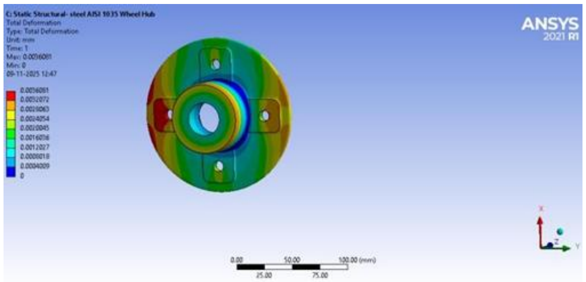


Fig. No. 19 Total deformation of the AISI 1035 Wheel Hub.

Equivalent Stress:

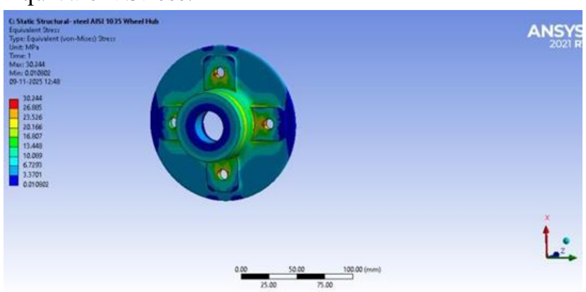


Fig. No. 20 Equivalent stress of AISI 1035 wheel hub.

Equivalent Elastic strain:

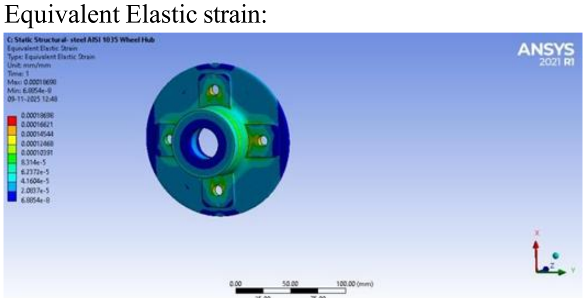


Fig. No. 21 Equivalent elastic strain of AISI 1035 wheel hub.

Life:

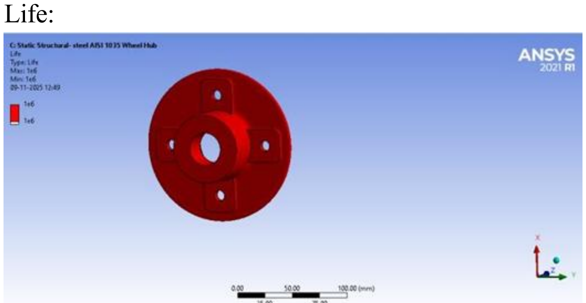
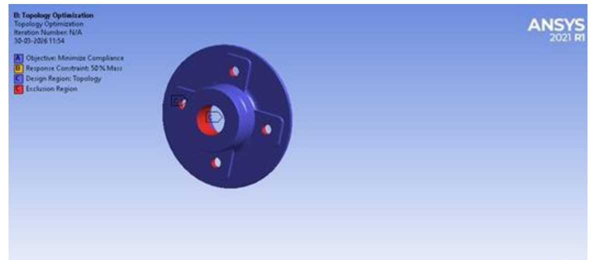
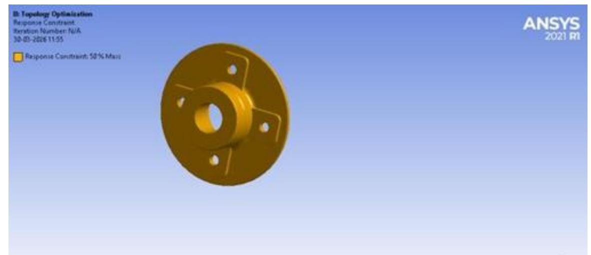


Fig. No. 22 Life of AISI 1035 alloy wheel hub.

Topology Optimization: Boundary Conditions:





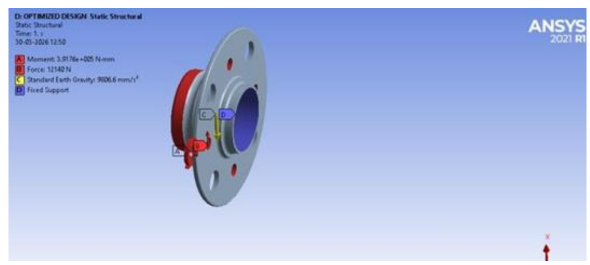
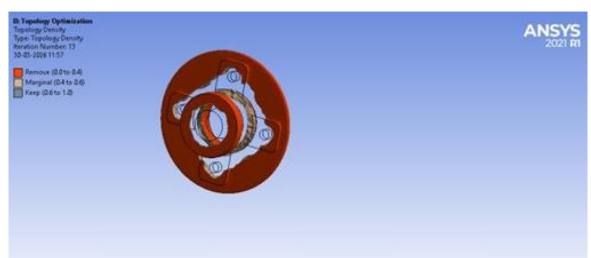


Fig. No. 26 Boundary Conditions of the new optimized wheel hub.



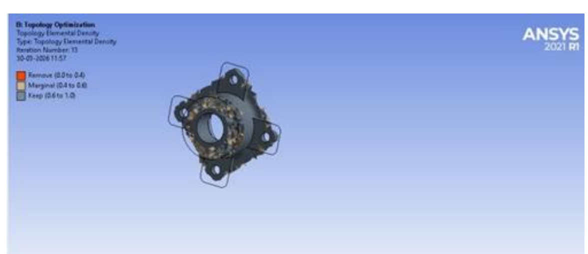


Fig. No. 23 Boundary Conditions for topology optimization.

Optimized Design:

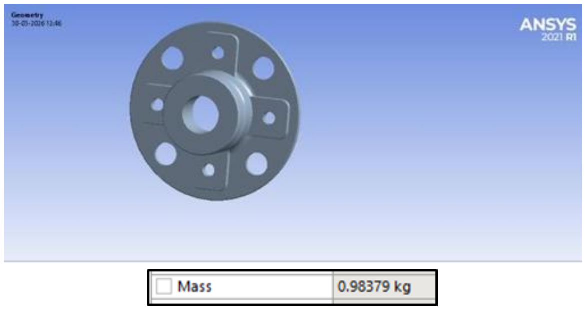


Fig. No. 24 New optimized wheel hub design.

Static Structural Analysis of the new optimized Wheel Hub: Meshing:

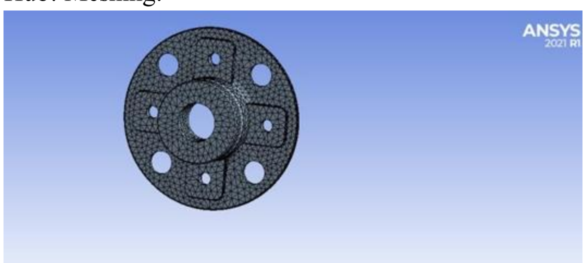


Fig. No. 25 Meshing details of the new optimized wheel hub.

Boundary Conditions:

Results:

Total Deformation:

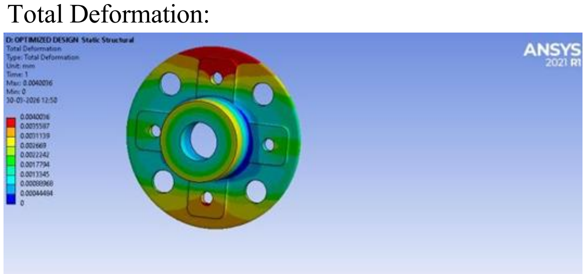


Fig. No. 27 Total deformation of the new optimized wheel hub

Equivalent Stress:

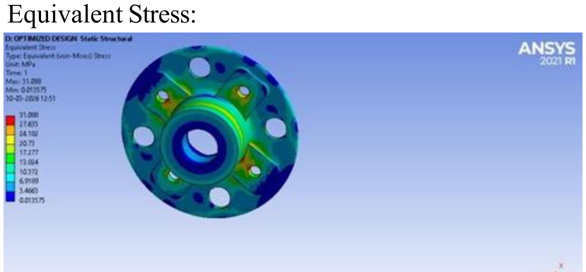


Fig. No. 28 Equivalent Stress of the new optimized wheel hub.

Equivalent Elastic Strain:

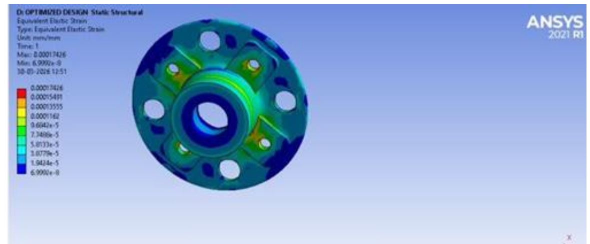


Fig. No. 29 Equivalent Elastic Strain of the new optimized wheel hub.

Life:

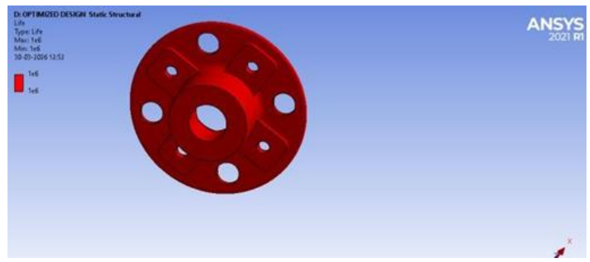


Fig. No. 30 Life of the new optimized wheel hub.

Safety factor:

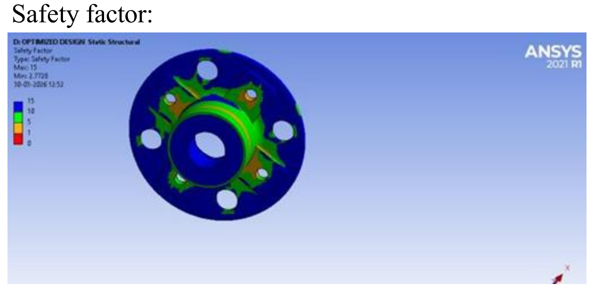


Fig. No. 31 Safety factor of the new optimized wheel hub.

FEA for experimental testing:

Boundary Conditions:

Fig. No. 32 Boundary conditions for experimental testing.

Results:

Total Deformation:

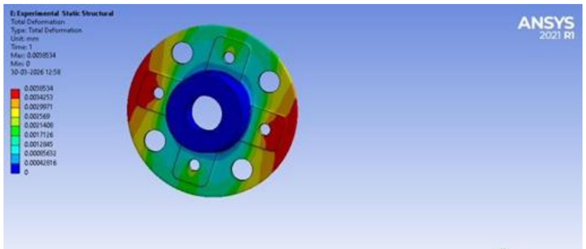


Fig. No. 33 Total Deformation of the hub.

Equivalent Stress:

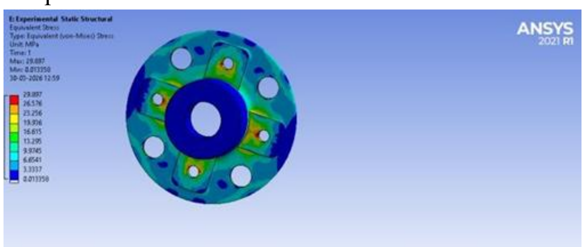


Fig. No. 34 Equivalent Stress of the hub.

Equivalent Elastic Strain:

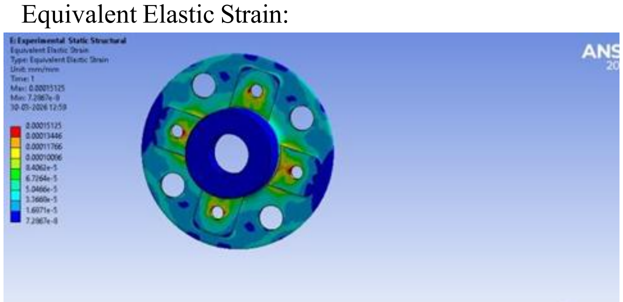


Fig. No. 35 Equivalent Elastic Strain.

FEA RESULTS

Parameters	MS wheel hub	Aluminium Wheel hub	AISI 1035 Wheel Hub
Total Deformation	0.0036269 mm	0.010375 mm	0.0036081 mm
Equivalent Stress	30.241 MPa	30.23 MPa	30.244 MPa
Equivalent Elastic Strain	0.00018657	0.00052172	0.00018698

FEA Results of New Optimized Wheel Hub:
Max. Total Deformation – 0.0040036 mm.
Equivalent Stress – 31.088 MPa.
Equivalent Elastic Strain – 0.00017426

V. MANUFACTURING OF THE COMPONENT

The optimized wheel hub was manufactured by modifying an existing hub geometry based on topology optimization results. A subtractive manufacturing approach was adopted to achieve weight reduction by removing material from identified low- stress regions without compromising structural integrity.

The hub was initially inspected and prepared for machining. The optimized design was translated into practical features, mainly in the form of drilling locations, and accurate marking was carried out to ensure proper positioning.

Drilling operations were performed using a vertical drilling machine with suitable cutting parameters such as spindle speed, feed rate, and tool selection. Material was removed in the form of multiple holes at predefined regions, ensuring adequate spacing to avoid stress concentration.

After machining, deburring and surface finishing were carried out to eliminate sharp edges and burrs. The component was then inspected dimensionally to verify conformity with the optimized design.
This method demonstrates a cost-effective and practical implementation of topology optimization using conventional machining. The resulting hub achieves reduced weight while maintaining strength and stiffness, making it suitable for further analysis and testing.

Fig. 36 Optimized specimen after drilling operation.

VI. EXPERIMENTAL TESTING

A Universal Testing Machine (UTM) is a type of mechanical testing equipment that is utilized for determining the mechanical properties of various materials, such as tensile strength, compressive strength, bending strength, and shear strength. The

UTM works by applying a controlled tensile or compressive load to the specimen being tested and measuring its response.

Testing Photograph

Fig.37 Specimen under compression test using UTM.

Testing Graph

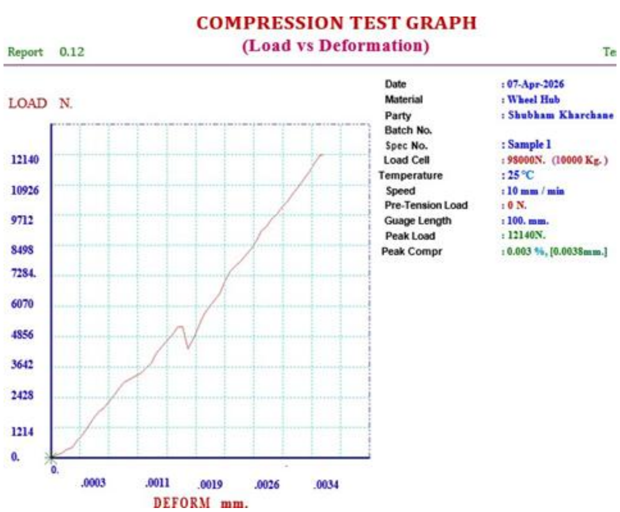


Fig. 38 Graph shows load vs displacement during Compression test

The above fig. shows the graph of displacement v/s load for automotive wheel hub. Displacement is taken on X-axis and load in N is taken on Y-axis.

Conclusion

This study presented a comprehensive evaluation of automotive wheel hubs through static structural analysis, material comparison, and topology optimization. The finite element results show that the maximum equivalent stress for Mild Steel (MS), Aluminum Alloy, and Steel AISI 1035 is approximately 30.24 MPa, which is significantly lower than their respective yield strengths. This confirms that all designs are structurally safe and operate within elastic limits, ensuring a high margin of safety against yielding.

A comparative analysis of deformation behavior reveals that the Aluminum Alloy hub exhibits nearly three times higher deformation than the steel-based hubs, indicating reduced stiffness despite its lightweight advantage. In contrast, Mild Steel and AISI 1035 demonstrate superior rigidity and dimensional stability, making them more suitable for load- bearing applications.

Topology optimization of the Mild Steel hub further enhanced performance by achieving weight reduction without compromising structural integrity. The optimized design resulted in a maximum deformation of 0.004026 mm, equivalent stress of 31.088 MPa, and equivalent strain of 0.00017246, indicating only a marginal increase in stress compared to the baseline model.

Experimental validation using a Universal Testing Machine (UTM) under compression loading conditions showed good agreement with the FEA results, thereby confirming the accuracy and reliability of the numerical analysis.

Overall, the study demonstrates that while all materials satisfy strength requirements, stiffness and weight-performance trade- offs are critical in material selection. Furthermore, topology optimization proves to be an effective tool for developing lightweight, structurally efficient, and reliable wheel hub designs, contributing to modern automotive engineering and lightweight design strategies.

REFERENCES

[1] Cang, P., Zhang, X., & Cui, W. (2023). Static and Modal Models of Automotive Wheel Hubs Based on Lightweight Design and Cost-Effectiveness. MDPI Applied Sciences, 13(18), 10164.
[2] Jia, J., & Sun, X. (2021). Topology Optimization of an ATV Wheel Hub. Journal of Physics: Conference Series, 2027(1), 012022.
[3] Almonti, D., et al. (2025). Lightweight and Sustainable Steering Knuckle via Topology Optimization. MDPI Applied Sciences, 9(8), 252.
[4] Manasa, C. M., et al. (2025). Reinforcement Learning- Based Topology Optimization for Lightweight Wheel Hubs.
[5] Long, M., et al. (2024). Research and Experimental Verification of Lightweight Wheel Hub Structures.
[6] Fang, Y., et al. (2024). Reliability Assessment of Additively Manufactured TB6 Titanium Alloy Wheel Hubs. ScienceDirect.This paper investigates the strength and reliability of additively manufactured titanium alloy wheel hubs. ScienceDirect
[7] Zhu, S., et al. (2025). Sketch-Guided Topology Optimization with Enhanced Structural Performance. MDPI Applied Sciences, 15(5), 2753.
[8] Qi, T., et al. (2024). Topology Optimization of a Ratchet Compensation Structure.
[9] Ma Long, Dong Zhikui, Liu Chunjiang, Gao Peng, Liang Ao, Hou Binfeng, Qi Xiangdong & Jiang Yunhong. (2024). Research and experimental verification of lightweight loader rims.
[10] Xinqiao Ding, Xia Liu (2024). Modal Analysis of Automobile Wheels based on Lightweight Technology.
[11] Liu, N. (2024). Bionic Optimization Design and Fatigue Life Prediction of Wheel Hub.
[12] Zhang, Y., et al. (2021). An Integrated Multi-Objective Topology Optimization Method for Automobile Wheels Made of Lightweight Materials. Structural and Multidisciplinary Optimization.