International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 04 , April - 2026

Design, Analysis and Fabrication of a Wheel Hub for a Formula SAE Vehicle

Kalyani Anumalla
Dept. of Mechanical Engineering
MCT's Rajiv Gandhi Institute of Technology
Mumbai, India

Bhoomi Gupta
Dept. of Mechanical Engineering
MCT's Rajiv Gandhi Institute of Technology
Mumbai, India

Mohd. Ajmal Javed
Dept. of Mechanical Engineering
MCT's Rajiv Gandhi Institute of Technology
Mumbai, India

Aditya Sarode
Dept. of Mechanical Engineering
MCT's Rajiv Gandhi Institute of Technology
Mumbai, India

Dr. N. J. Panaskar
Assistant Professor, Dept. of Mechanical Engineering
MCT's Rajiv Gandhi Institute of Technology
Mumbai, India

Abstract **- This work presents the complete development of a wheel hub intended for a Formula SAE vehicle, including load estimation, geometric design, numerical validation, and fabrication. The component is required to sustain combined loading arising from vehicle weight, braking, and cornering. Load cases were established using actual vehicle parameters and used to evaluate the design through finite element analysis.**

The hub geometry was created in CAD with attention to stress flow, assembly constraints, and manufacturability. Aluminium 7075-T6 was selected due to its high strength-to-weight ratio and suitability for high-load applications. The optimized design demonstrates reduced stress and deformation compared to a previously used hub.

The manufactured component confirms the feasibility of the design. The results show that the hub can safely withstand operating loads with a high factor of safety, making it suitable for Formula SAE applications.

The optimized design achieved approximately 30% reduction in stress and 78% reduction in deformation compared to the previous design.

Several studies have investigated wheel hub design and optimization for Formula SAE vehicles [1], [3], [4].

Keywords - Formula SAE, wheel hub, structural analysis, ANSYS, optimization

I. INTRODUCTION

In a Formula SAE vehicle, the wheel hub serves as the interface between the wheel assembly, braking system, and drivetrain. It is responsible for transmitting loads while maintaining alignment under varying operating conditions such as acceleration, braking, and cornering.

The design of such a component requires balancing strength, weight, and manufacturability. While several studies focus on numerical optimization of wheel hubs, many lack validation through fabrication and comparison with existing designs.

The present work addresses this by combining load-based design, structural validation, and physical manufacturing, along with a direct comparison to a previously developed hub.

II. PROBLEM STATEMENT

The wheel hub must withstand combined loading without excessive deformation or failure. The challenge lies in achieving sufficient strength while minimizing weight and ensuring compatibility with adjacent components. Additionally, the design must be manufacturable using available machining processes.

III. OBJECTIVES

- Determine realistic loading conditions
- Design a lightweight hub geometry
- Select an appropriate material
- Validate the design using finite element analysis
- Compare performance with an existing hub
- Manufacture the final component

IV. METHODOLOGY

The development process follows:

Vehicle Data → Load Calculation → CAD Design → Material Selection → FEA → Optimization → Fabrication

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 04 , April - 2026

Vehicle parameters used include mass (300 kg), weight distribution (55% front), wheelbase (1.525 m), center of gravity height (0.321 m), and track width (1.25 m).

V. MATERIAL SELECTION

The hub material must withstand high stresses while contributing minimal weight. Aluminium 7075-T6 was selected due to its high yield strength and excellent strength-to-weight ratio. This alloy is widely used in aerospace and high-performance automotive applications where structural efficiency is critical.

Compared to lower-strength aluminium alloys, 7075-T6 provides superior resistance to deformation under load. Although it requires careful machining, its performance advantages make it suitable for the present application.

Aluminium 7075-T6 is widely used in high-performance automotive applications due to its superior strength-to-weight ratio [1], [4].

The selected material ensures that the hub maintains structural integrity under severe loading conditions while minimizing mass.

VI. LOAD CALCULATIONS

Total vehicle weight is given by:

$$W = m \times g = 300 \times 9.81 = 2943 \text{ N}$$

Front axle load (55%): 1618.65 N → per wheel: 809.3 N

Braking Load Transfer:

$$\Delta W = m \times a \times h / L$$

$$\Delta W \approx 929.8 \text{ N}$$

Dynamic front wheel load ≈ 1274 N

Braking Force:

$$F = m \times a = 4414.5 \text{ N}$$

Per front wheel ≈ 1545 N

Braking Torque:

$$T = F \times R \approx 383 \text{ N·m}$$

Lateral Load Transfer:

$$\Delta W \approx 1134.8 \text{ N}$$

Outer wheel load ≈ 1377 N

- **Worst-Case Loads:**

Vertical: 1300 – 1400 N

Longitudinal: ~ 1500 N

Lateral: ~ 1300 N

Torque: ~ 380 – 400 N·m

VII. DESIGN AND CAD MODEL

The hub geometry includes a mounting flange, bearing seats, threaded section, and CV joint interface for connection with the half shaft. Fillets were incorporated at transitions to reduce stress concentration. Material was removed from low-stress regions to achieve weight reduction.

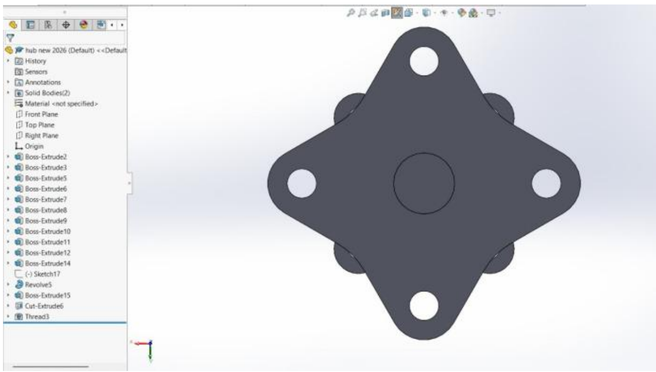

Fig. 1. Front View CAD image

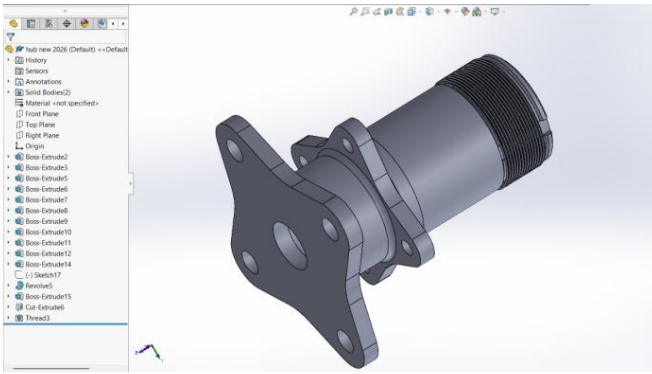

Fig. 2. Isometric View CAD image

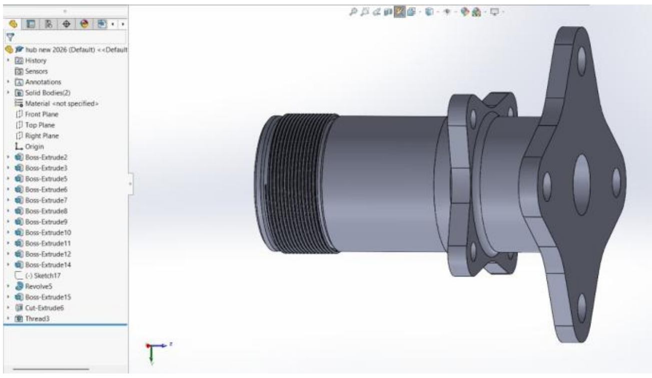

Fig. 3. Side View CAD image

VIII. STRUCTURAL ANALYSIS

Finite element analysis was performed using ANSYS under static loading conditions.

Finite element analysis is commonly used to evaluate structural performance of automotive components under realistic loading conditions [5].

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 04 , April - 2026

Results:

Parameter	Value
Maximum Deformation	0.60778 mm
Maximum Stress	23.907 MPa
Maximum Strain	1.4157×10^{-4}
Maximum Normal Stress	11.809 MPa
Factor of Safety	10.45

Table 1. Summary of structural analysis results

The analysis results indicate that the maximum deformation occurs near the outer region of the hub, while the fixed regions remain stable. The equivalent stress is concentrated near the transition between the flange and threaded section due to geometric discontinuity. However, the maximum stress value of 23.907 MPa is significantly lower than the yield strength of Aluminium 7075-T6, ensuring safe operation under the applied loading conditions.

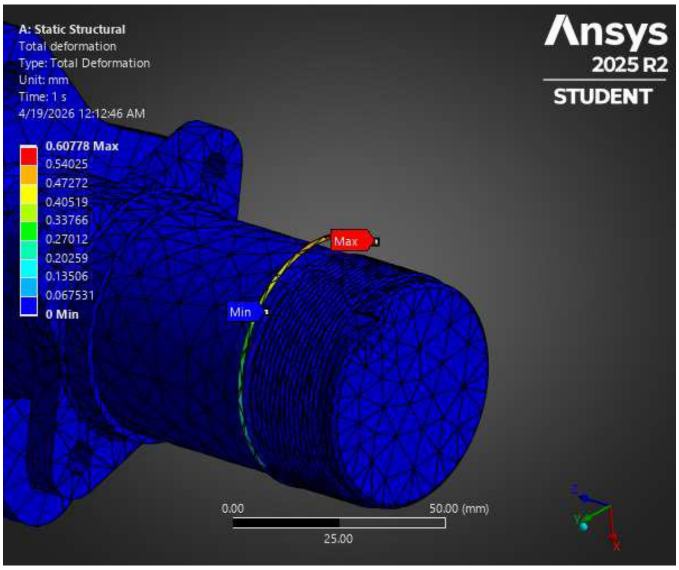

Fig. 4. Total deformation of optimized wheel hub

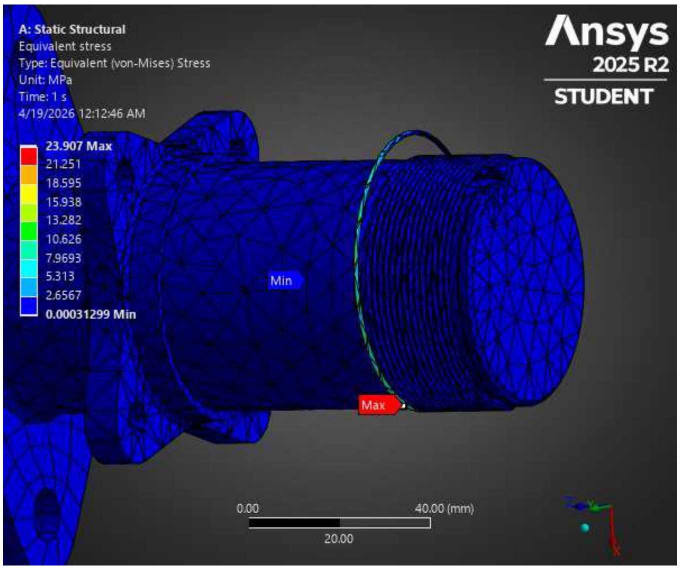

Fig. 5. Equivalent (von-Mises) stress distribution

IX. COMPARISON WITH PREVIOUS DESIGN

Parameter	Previous Hub	New Hub	Improvement
Stress	34.263 MPa	23.907 MPa	~30% reduction
Deformation	2.75 mm	0.607 mm	~78% reduction
Factor of Safety	Lower	10.45	Increased

Table 2. Comparison Table

The improved design shows better stiffness and more uniform stress distribution.

X. DESIGN IMPROVEMENTS

Performance improvements are achieved through optimized geometry and better load distribution. The introduction of fillets reduces stress concentration, while redistribution of material enhances stiffness. The CV joint interface enables smooth torque transmission while accommodating angular misalignment during suspension and steering movement.

The CV joint interface is designed to accommodate angular variation while maintaining constant velocity transmission, which is essential for Formula Student vehicle dynamics.

XI. FABRICATION

The hub was manufactured using turning, drilling, and milling operations. The threaded section was produced using lathe machining, while flange holes were drilled with precision. The internal CV joint interface profile was machined to ensure proper seating and alignment of the CV joint for efficient torque transmission.

Similar fabrication approaches have been reported in previous studies on Formula vehicle wheel assemblies [3].

Minor machining marks were observed but do not affect performance.

Fig. 6. Flange of the wheel hub

Fig. 7. Side view of the wheel hub

Fig. 8. Internal CV joint interface of the wheel hub

XII. RESULTS AND DISCUSSION

The results indicate that the hub performs within allowable limits under combined loading. Reduced deformation improves structural stability, while the high safety factor ensures reliability during operation.

XIII. CONCLUSION

The developed hub demonstrates improved performance compared to the previous design. Reduction in stress and deformation confirms the effectiveness of the design approach. The use of Aluminium 7075-T6 enhances structural capability, and fabrication confirms practical feasibility.

The results demonstrate that the optimized design successfully balances strength, weight, and manufacturability for Formula Student applications.

The integration of design, simulation, and fabrication provides practical validation of the proposed methodology, enhancing its applicability in real-world Formula SAE vehicle development.

XIV. FUTURE SCOPE

- Fatigue analysis under cyclic loading
- Experimental validation
- Advanced optimization techniques
- Material alternatives

XV. ACKNOWLEDGEMENT

The authors would like to thank the Department of Mechanical Engineering, MCT's Rajiv Gandhi Institute of Technology, for providing the necessary facilities and support for this work.

REFERENCES

[1] Rohitraj N., Arun Sundar S., "Wheel hub design for formula electric vehicle," IJERT, 2020.

[2] I.S. Jacobs, C.P. Bean, "Fine particles, thin films and exchange anisotropy," Magnetism, Vol. III, 1963.

[3] H. Bhural et al., "Wheel assembly for formula type automotive," 2021.

[4] A. Dabb et al., "Wheel hub optimization for formula race car," IRJET, 2019.

[5] D. J. Johnson et al., "Dynamic analysis of formula car," IJSRET, 2025.

[6] Ponshanmugakumar, R. Rajavel, "Wheel hub for ATV," IJMPERD, 2018.