Published by :
https://www.ijert.org/
An International Peer-Reviewed Journal

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 05 , May - 2026

# Lightweight Automotive Wheel Rim Design via Finite Element Analysis and Density-Based Topology Optimization: Comparative Assessment of AA6061-T6 and Al-Sc Alloys

Ednilson Jacinto Mutemba[1]
Student, Department of Mechanical Engineering
Hubei University of Technology
Wuhan, China

Xue Huan[2*]
Professor, Department of Mechanical Engineering
Hubei University of Technology
Wuhan, China

*Abstract* - Lightweight wheels can significantly improve vehicle efficiency, handling, and ride comfort. However, automotive rims must maintain structural safety under complex service loads, including cornering forces, tire inflation pressure, vertical loads, and rotational effects. This study may well combines finite element analysis (FEA) and density-based topology optimization to improve and evaluate the stiffness to weight performance of two wheel rims: a Conventional design made of AA6061 T6 and New Design made of Al–Sc alloy. The baseline CAD geometries were analysed using validated static FEA under combined cornering, radial, inflation, and rotational loads to quantify deformation, von Mises stress, and safety factor. A simp-based topology optimization was then applied to the spoke and rim band regions with mass minimization as the aim. The optimised designs were remeshed and re-evaluated under identical load cases to verify structural consistency. For the conventional design, a 20 % mass reduction was achieved, reducing the mass from 24.64 kg to 19.6kg, while maintaining low deformation of 0.081 mm, a peak von ises stress of 29.48 MPa, and a safety factor above 2.8. For the New design, the optimization improved stress distribution, reducing the maximum von Mises stress from 130.08 MPa to 91.51 MPa, while increasing the safety factor from 1.15 to 1.64 and reducing the mass to 14.925kg. These results show that topology optimization can improve the stiffness-to-weight performance of wheel rims, but the benefits depend strongly on the initial geometry and material

Keywords - *Structural-performance*; Finite Element Analysis; *Topology Optimization*; *Lightweighting*, *Alloy wheel*

## I. INTRODUCTION

Lightweight automotive component play a significant role in automotive designs in the present day because sprung mass is directly proportional to handling, brake response, ride comfort, and fuel or energy efficiency. Between these components, wheel rims particularly play an important role since they have to fulfill structural, and enduringness requirements while operating under continuous service loads, including cornering loads, radial loads, tire inflation pressure, and rotational inertia. One of the greatest engineering challenges is to reduce mass and ensure structural integrity. Based on this, finite element analysis (FEA) has become a an essential technique used to estimate components stiffness, stress distribution and deformation under realistic load conditions. Expanding on this, FEA can give accurate results of tension concentrations and safety when these are checked by mesh convergence and test-based boundary conditions. At the same time, topology optimization has become an effective computation tool to eliminate non-load-bearing material without compromising performance requirements and to enable designers to seed lightweight and efficient geometries across all cases. However it is also worth mentioning that despite this, most of the studies report optimized designing with no clear demonstration of how geometry redistribution affects the stress way, stiffness and the common margins to the baseline models, which diminishes clarity. According to the evidence, another crucial factor of optimization is material choice. It is evidenced that, AA6061-T6 aluminum corpse is commonly used because of its desirable strength-to-weight ratio and manufacturing capability, however, alloys such as aluminum-scandium alloy (Al–Sc) are better yielding, with better fatigue impedance and enhanced aggressive lightweighting capabilities. It has to be mentioned that, disgusts of such benefits, comparative studies with topology optimisation and various alloys in wheel blueprint are not numerous. This paper studies the structural behaviour and mass reduction capabilities of two automotive wheel rims, one casted with AA6061-T6 and the other with Al–Sc. A standard FEA-based workflow and constrained topology optimization approach were used to evaluate both designs before and after optimization. Total deformation, von Mises stress distribution, factor of safety, and mass reduction are compared to quantify the effects of material selection and optimized geometry on stiffness-to-weight performance. The study provides a systematic and transparent framework for developing lightweight automotive wheel rims while considering structural safety and manufacturability.

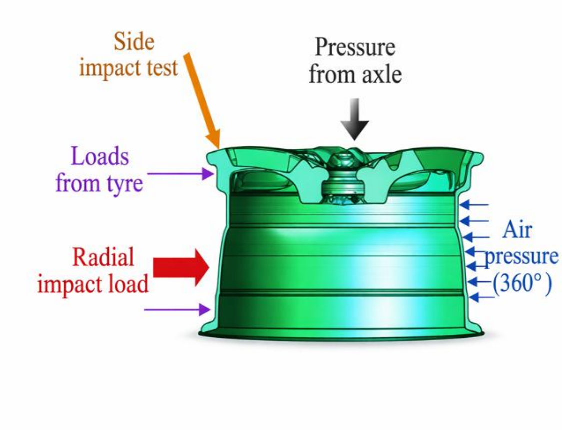

Fig 1. Typical wheel rim loads and test concepts

### A. *Problem Definition*

The key engineering challenge addressed by the paper is to reduce the mass of automotive wheel rims without compromising safety or performance. Lightweight wheels improve vehicle efficiency, handling and ride comfort, but rims must still withstand complex loads such as cornering forces, inflation pressure, vertical loads and rotational effects. Finite-element analysis (FEA) is a proven tool for estimating stiffness, stress distribution and fatigue life under realistic conditions; topology optimization can remove non-load-bearing material to achieve lightweight yet efficient geometries. However, many studies do not demonstrate how geometry redistribution alters stress paths and safety margins, and they rarely compare different alloys. To fill this gap, the paper formulates a problem where the objective is to minimize the mass of two wheel-rim designs (conventional six-spoke and new multi-spoke) while maintaining adequate structural stiffness and safety factors under combined cornering, radial, inflation and rotational loads.

### B. *Methodology*

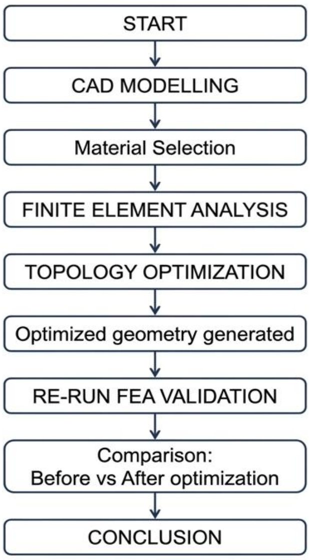

*Fig 2. Flow chart of methodology*

## II. MODELING OF THE WHEEL RIM

Fusion 360 software was used for the design and modification of the rim., chosen for its ability to generate precise parametric geometries and smooth surface passage suitable for finite element analysis. These models were exported in STEP format to assure compatibility with the finite element solver. This study potentially considered two wheel rim designing: a conventional rim and a new rim. This observation, both design were crafted to meet standard passenger-car fitment requirements. Key geometric parameters admit the boilersuit rim diameter, rim width, hub diameter, bolt pitch circle diameter (PCD), stud-hole diameter, and center bore. These interface feature are critical for vehicle climbing and were therefore kept as non-design region during topology optimization. The conventional in many cases rim has an overall diameter of 545.0 mm and a rim width of 205.0 mm. The lightweight rim features a bead-seat rim width of 204.05 mm and an overall outer cross-sectional width, include rim flanges, of 255.91 mm. These results, the hub could be and bolt-circle feature match in both designs to assure compatibility with the same vehicle chopine. This approach, detailed dimensional information for both rims is shown in figure 3 and 4. All dimensions were chosen based on typical passenger-car wheel specifications and kept consistent between the cad models and the finite constituent meshes. It is worth noting that, these baseline geometries function as reference configurations for subsequent finite element analysis and topology optimisation, ensuring that all performance variations are due solely to material redistribution rather than modification in fitment geometry.

*Fig 3. Design and dimensions (conventional rim)*

*Fig 4. Design and dimensions (New rim)*

## III. ANALYSYS OF WHEEL RIMS

The pre-optimization phase is to establish the initial mechanical performance of the conventional and new rim designs under the defined loading and boundary conditions. After topology optimization, the same loading conditions is reapplied to the optimized geometries so that a direct and consistent comparison could be made between the both on before-optimization and after-optimization cases. This comparison made it possible to assess how topology optimization influenced deformation response, stress distribution, stiffness retention, and overall structural performance. Material Selection: AA6061-T6 and Al–Sc alloy were selected as the candidate materials for evaluation. Material specifications are given in table 1.

| Property | AA6061-T6 (Conventional) | Al–Sc-Alloy (New Design) |
|---|---|---|
| **Elastic modulus, *E*** | 70 GPa | 74 GPa |
| **Poisson's ratio, *ν*** | 0.33 | 0.33 |
| **Density, *ρ*** | 2700 kg·m$^{-3}$ | 2750 kg·m$^{-3}$* |
| **Yield strength, *σy*** | 275 MPa | 450 MPa |
| **Ultimate strength, *σu*** | 310 MPa | 450 MPa |

*Table 1. Mechanical properties used in FEM*

### A. *Meshing*

The two generally speaking geometries were engage exploitation parabolic tetrahedral elements to effectively capture stress gradients at curved interfaces. This approach, the boiler suit constituent size was set to 7 mm, with local refinement to 2 mm at bolt holes, junctions of spokes and rim, and valve hole area. It is worth noting that, the interlocking approaching is illustrated in figure 3, showing the refined elements where high stress concentrations were expected. A mesh convergence study was do by halving the element sizes until the maximum stress changed by less than 3 percent, ensuring numerical accuracy..

*Figure 5. Finite-element mesh models of both rims*

### B. *Boundary Conditions and Load Aplications*

Both rim models were analyzed under the same constraint and load conditions to ensure a fair comparison. The bolt-hole regions were assigned fixed supports in all translational directions to represent the clamped connection between the rim and the vehicle hub. The models were then subjected to combined service , including cornering force, vertical load, tire inflation pressure, and rotational velocity.

The vertical wheel load was calculated based on an assumed vehicle mass of **1640 kg**, evenly distributed across four wheels. Therefore, the load supported by one wheel was:

$$m\omega = \frac{1640}{4} = 410\,kg \tag{1}$$

The vehicle speed used in the analysis was **50 km/h**, which was converted to meters per second as follows:

$$v = \frac{50 \times 1000}{3600} = 13.89\,m/s \tag{2}$$

Based on this speed and an assumed turning radius of **10 m**, the concerning force was calculated as:

$$Fc = \frac{410 \times 13.89^2}{10} = 7910.2\ \mathrm{N} \tag{3}$$

A vertical load of **4022.1 N** was applied at the hub center, calculated from the single-wheel supported mass:

$$Fv = 420 \times 9.81 = 4022.1\mathrm{N} \tag{4}$$

The tire inflation pressure was applied as 0.24 MPa, equivalent to approximately 35 PSI. Rotational velocity was calculated from the vehicle speed and the rolling radius of each rim model:

$$\omega = \frac{v}{r} \tag{5}$$

For the conventional wheel rim, the rolling radius obtained from the wheel geometry was 0.27343 m, resulting in:

$$\omega = \frac{13.89}{0.27343} = 50.8\,rad/s \tag{6}$$

For the new wheel rim design, the rolling radius was 0.21502 m, resulting in:

$$\omega = \frac{13.89}{0.21502} = 64.59\,rad/s \tag{7}$$

Thus, the conventional rim was analysed using a rotational velocity of 50.8 rad/s, while the new design was analysed using 64.59 rad/s. All remaining loading conditions were kept constant for both models to ensure a consistent comparative evaluation.

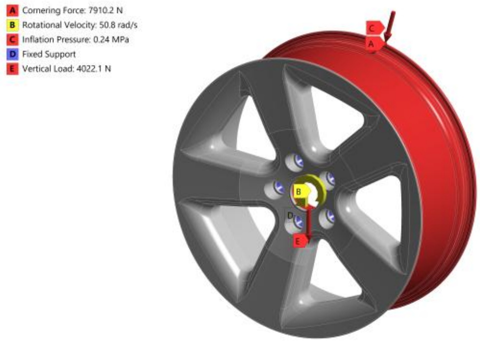

*Fig 6. Boundary conditions of conventional design*

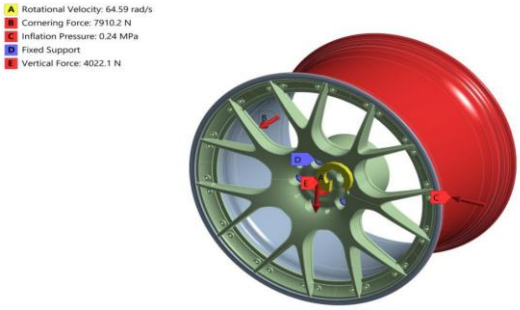

*Fig 7. Boundary conditions of the New design*

## C. *Static Analysis*

Ansys performs static analysis after applying constraints and forces to both rims. Total deformation, stress, and safety factor are assessed below:

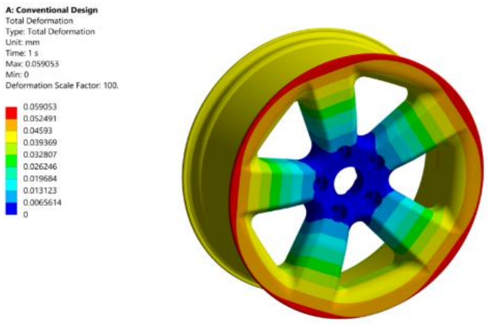

*Fig 8. Conventional model — total deformation before TO*

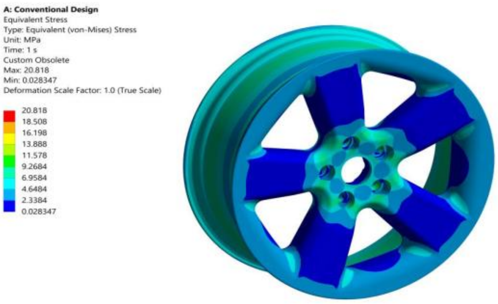

*Fig 9. Conventional model — equivalent stress before T0*

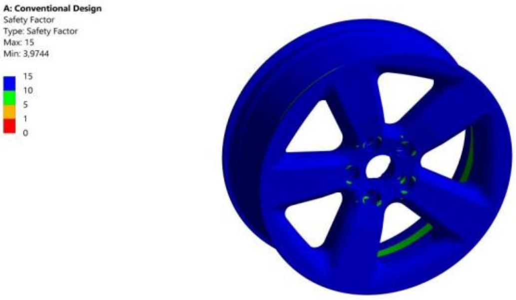

*Figure 10. Conventional design — Safety factor before TO*

- Total deformation on the conventional rim before Topology Optimization is 0.059 mm
- Equivalent Stress on the conventional rim before Topology Optimization is 20.818 MPa
- The minimum Safety Factor on the conventional rim before Topology Optimization is 3.97

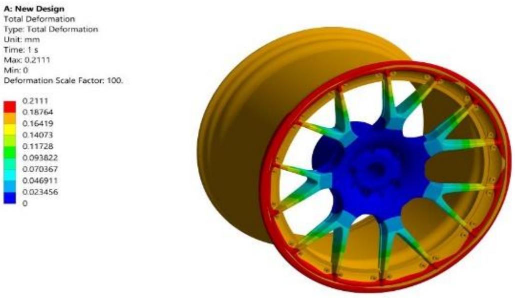

*Fig 11. New design — Total deformation before TO*

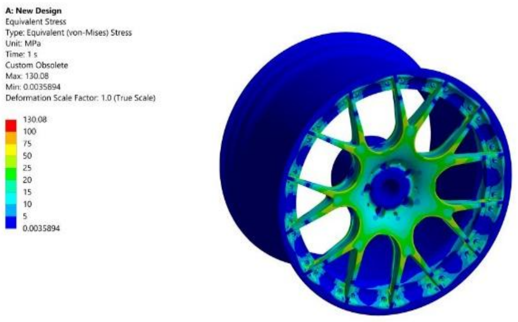

*Fig 12. New design — equivalent stress before TO*

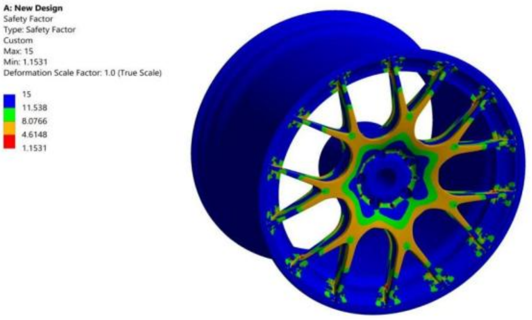

*Fig 13. New design — Safety factor before TO*

- Total deformation on the New rim before Topology Optimization is 0.211 mm
- Equivalent Stress on the New rim before Topology Optimization is 130 MPa
- The minimum Safety Factor on the New rim before Topology Optimization is 1.15

### D. *Baseline Comparison and Design Implications (No Topology Optimization Applied)*

**Stiffness and deflection.** The Conventional rim is less flexible than the new one because of its increased thickness and mass. In terms of ride and handling, the two deformations are minor, but the traditional wheel has a larger stiffness margin, which could be balanced with weight reduction.

**Stress and safety factor.** Significant unused capacity is suggested by the stress of the wheel and minimum safety factors. The new design shows a more efficient use of material, thanks to the excellent strength of Al- Sc, which allows for maintaining margins with less mass. Both design methods are safe based on static analysis.

**Mass efficiency and optimization potential.** The conventional rim is a good candidate for extensive material removal because it is light and has a comfortable safety factor. The new design is lightweight at 15.08 Kg and should be focused on smoothing the spoke roots and smoothing the material to prevent margins from erosion.

## IV. TOPOLOGY OPTIMIATION PROCESS

In this study, the topology optimization problem was defined with the primary objective of minimizing the mass of the wheel rim while maintaining adequate structural stiffness under the prescribed loading conditions. The optimization strategy was therefore based on a stiffness weight balancing approach, where material was removed only from the allowable design space in order to improve mass efficiency without causing unacceptable structural deterioration. The design region for topology optimization was mainly assigned to the spokes and selected rim sections, where material redistribution could be permitted. In contrast, the bolt-hole region and hub mounting area were treated as protected or non-design regions because these areas are functionally essential for assembly, clamping, and load transfer, and therefore could not be modified during optimization.

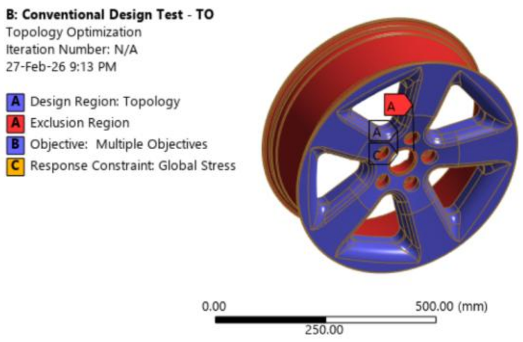

*Fig 14. Conventional design — Exclusion Region*

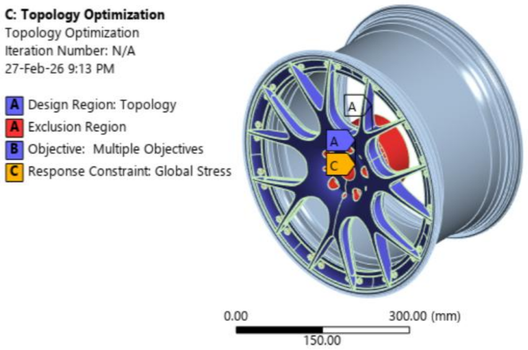

*Fig 15. New Design — Exclusion Region*

### A. *Topology Optimization Results*

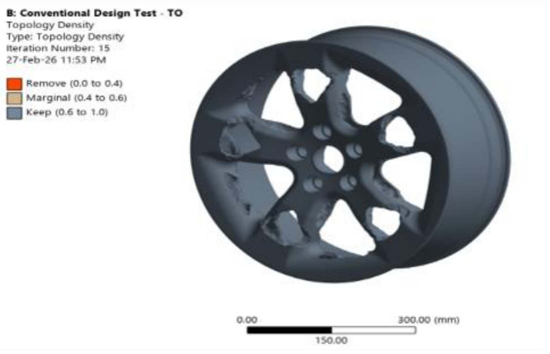

*Fig 16. Topology Optimization Result (Conventional Rim)*

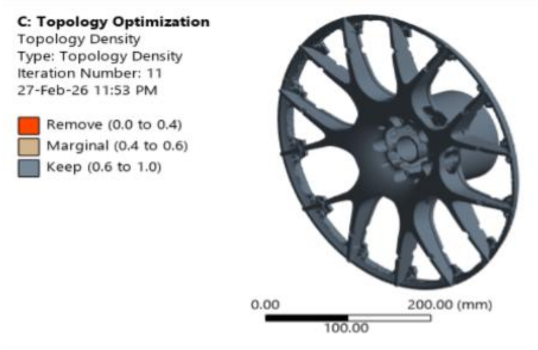

*Fig 17. Topology Optimization— New Rim*

The topology optimization results differed significantly between the two rim concepts because the baseline structural reserve of each design was different. The Conventional rim exhibited greater excess material in the initial configuration and therefore showed a higher potential for mass reduction with limited structural penalty. In contrast, the New rim design was already lightweight, meaning that further material removal affected regions that were more structurally active, making the design more sensitive to stiffness reduction and fatigue deterioration.For the conventional rim, topology optimization reduced the mass from 24.641 kg to 19.598 kg, corresponding to a mass reduction of 5.043 kg or 20.5%. In contrast, the new design showed only a small mass reduction, from 15.08 kg to 14.925 kg, corresponding to a decrease of 0.155 kg or approximately 1.03%. These results indicate that the conventional rim contained a larger amount of structurally underutilized material, whereas the new design was already close to a lightweight configuration before optimization.

### B. *Static structural results: Conventional and New Design (After TO)*

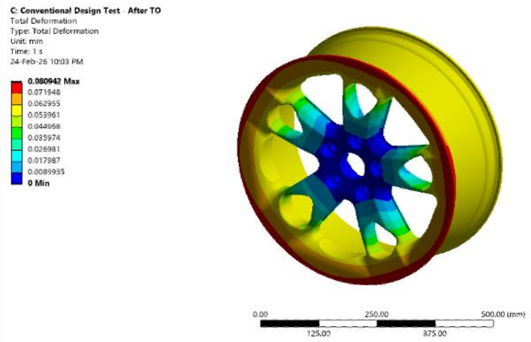

*Fig 18. Total Deformation Conventional — After TO*

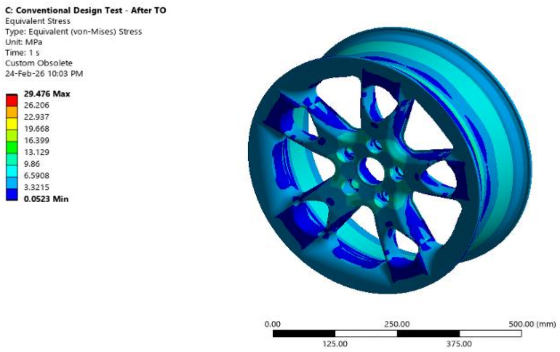

*Fig 19. Von-Mises Conventional — After TO*

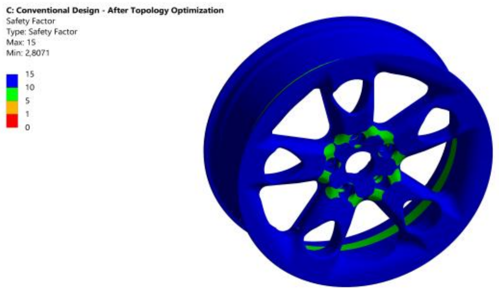

*Figure 20. Safety Factor Conventional — After TO*

- After topology optimization, the Conventional Model showed a slight increase in maximum deformation from 0.059 mm to 0.080942 mm, indicating reduced stiffness.
- The maximum von Mises stress increased from about 20.8 MPa to 29.476 MPa, showing higher stress concentration after material removal, but still within the safe elastic range.
- After topology optimization, the safety factor decreased from 3.97 to 2.8071, but the optimized structure still satisfied the required safety margin and remained structurally safe.

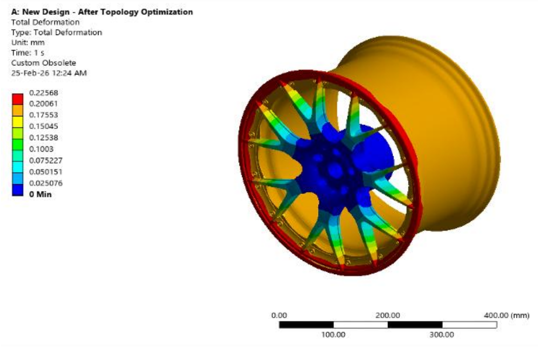

*Fig 21. Total Deformation New Design — After TO*

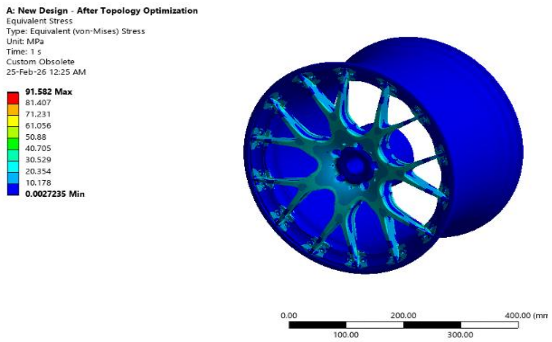

*Fig 22. Von-Mises New Design — After TO*

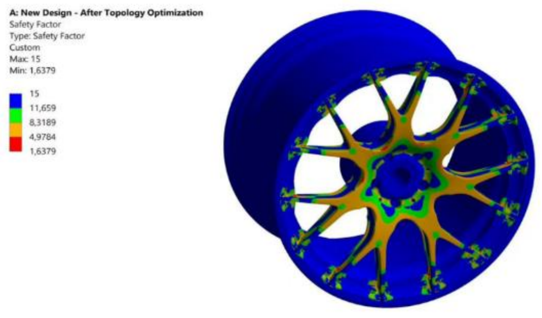

*Figure 23 Safety Factor New Design — After TO*

- The New Design showed only a slight increase in maximum deformation after topology optimization, from 0.2111 mm to 0.22568 mm, indicating that global stiffness was largely preserved.

- the maximum von Mises stress decreased significantly from 130.08 MPa to 91.582 MPa, showing an improved stress distribution after optimization.

- The minimum safety factor increased from 1.15 to 1.64, confirming that the optimized design achieved a better safety margin under the same loading conditions.

## C. *Comparative and Interpretation*

a)

| **Model** | Max stress before TO (MPa) | Max stress after TO (MPa) | Safety Factor before TO | Safety Factor after TO |
|---|---|---|---|---|
| Conventional Design | 20.818 | 29.476 | 3.97 | 2.8 |
| New Design | 130.08 | 91.582 | 1.15 | 1.64 |

b)

| **Model** | Weight before TO (kg) | Weight after TO (kg) | Max deformation before TO (mm) | Max deformation after TO (mm) |
|---|---|---|---|---|
| Conventional Design | 24.641 | 19.598 | 0.059053 | 0.080942 |
| New Design | 15.08 | 14.925 | 0.2111 | 0.22568 |

*Table 2. (a & b) Static structural comparison for Conventional Design and New Design (before vs after TO)*

**Weight efficiency of topology optimization :** The Conventional Model achieved a much larger weight reduction (~20.5%) than the New Design (~1.0%). This supports the report's explanation that the conventional geometry initially contained more excess material.

**Stiffness trade-offs :** The deformation increased after topology optimization in both models, indicating a reduction in structural stiffness due to material removal. However, this outcome is expected in lightweight design and should be interpreted as a stiffness–weight trade-off rather than a negative result. The Conventional Model experienced a larger stiffness penalty because it underwent much greater mass reduction, whereas the New Design preserved most of its global stiffness with only a slight increase in deformation.

**Stress evolution and what it implies :** The Conventional Model experienced a stress increase after TO (20.818 MPa → 29.476 MPa), while the New Design experienced a stress decrease (130.08 MPa → 91.582 MPa). Here is the key contrast your thesis narrative should have:

- TO in the Conventional Model was primarily material removal, moving the stress increasing upwards but still below the yield.
- TO has been more difficult to interpret as stress rebalancing in the New Design, and has reduced peak stress with the same deformation remaining near

baseline which is indicative of better load paths.

**Safety margin outcomes:** The reported safety factors were 3.97 before topology optimization and 2.8 after topology optimization for the Conventional Model. Although the safety factor decreased after optimization, the structure remained within a safe range, indicating that the optimized conventional rim was still structurally secure under the applied loading conditions. For the New Design, the safety factor increased from 1.15 before topology optimization to 1.64 after optimization, showing that the optimized configuration achieved an improved safety margin. Overall, the Conventional Model remained highly safe, while the New Design demonstrated enhanced structural robustness after topology optimization.

## V. CONCLUSION

This study investigated the structural performance and lightweighting capability of two automotive wheel rim designs using static finite element analysis and topology optimization. The results showed that both the conventional AA6061-T6 rim and the new Al-Sc alloy rim remained structurally safe under the applied loading conditions before and after optimization. The conventional rim achieved the largest mass reduction, decreasing from 24.641 kg to 19.598 kg, although this was accompanied by a moderate increase in deformation and stress and a reduction in safety factor from 3.97 to 2.8. In contrast, the new rim showed only a small mass reduction, from 15.08 kg to 14.925 kg, but exhibited improved stress distribution, with von Mises stress decreasing from 130.08 MPa to 91.582 MPa and safety factor increasing from 1.15 to 1.64. These findings show that topology optimization can serve different purposes depending on the initial design condition: for heavier conventional rims, it is highly effective for weight reduction, while for already lightweight rims, it is more valuable for improving structural efficiency and load distribution. Overall, the study confirms that combining FEA with topology optimization provides an effective framework for developing lighter and safer automotive wheel rims.

## ACKNOWLEDGMENT

The authors gratefully acknowledge the Hubei University of Technology for providing access to computational facilities and software used in this research. The authors also thank colleagues from the School of Mechanical Engineering for their technical support and constructive discussions throughout the study

## REFERENCES

[1] W. Zhang and J. Xu, “Advanced lightweight materials for automobiles: A review,” Materials & Design, vol. 221, Art. no. 110994, 2022, doi: 10.1016/j.matdes.2022.110994.

[2] S. Shahbazian, A. Pasharavesh and A. A. Khayyat, “Geometrical optimization of aluminum alloy wheels for high fatigue and impact strength,” Journal of Materials Science: Materials in Engineering, vol. 21, Art. no. 47, 2026, doi: 10.1186/s40712-026-00418-9.

[3] J. Lockett, M. Fahad, A. W. Awan and S. Islam, “A Digital Testing Framework for Design Improvements of Three-Piece Alloy Wheels Through Finite Element Analysis,” Applied Sciences, vol. 15, no. 21, Art. no. 11654, 2025, doi: 10.3390/app152111654.

[4] G. Zhang, S. Tao, Y. Zhou, Z. Ye, J. Lu, R. Li, S. Li, L. Zhou and T. Deng, “Shape optimisation of rim structure of aluminium alloy car wheels based on 90° impact test,” Scientific Reports, vol. 15, Art. no. 24623, 2025, doi: 10.1038/s41598-025-07802-z.

[5] V. R. I. Santos, F. Scinocca and F. L. Santos, “Virtual Development of Automotive Wheels: Modal, Vertical Impact and Fatigue Analysis and Simulation, Using the Finite Element Method,” Theoretical and Applied Engineering, vol. 9, no. 1, pp. 1–20, 2025, doi: 10.31422/taae.v9i1.63

[6] X. Meng, X. Feng, P. Liu and X. Sun, “Topology Analysis and Structural Optimization of Air Suspension Mechanical-Vibration-Reduction Wheels,” Machines, vol. 12, no. 7, Art. no. 488, 2024, doi: 10.3390/machines12070488

[7] M. Long, J. Guo, Y. Li, and X. Zhang, “Research and experimental verification of lightweight design of aluminum alloy wheels,” Scientific Reports, vol. 14, Art. no. 12266, 2024, doi: 10.1038/s41598-024-62667-y.

[8] G. Zhang, X. Cui, Y. Zang, Y. Zhou, J. Lu, Z. Li, H. Yang, J. Wang, Z. Ye, R. Li and L. Zhou, “Topology optimization of wheel spoke cavities for lightweight design under bending fatigue and impact load cases,” Scientific Reports, vol. 16, Art. no. 10817, 2026, doi: 10.1038/s415