STRUCTURAL OPTIMIZATION OF ENGINE MOUNT FOR MARUTI SUZUKI WAGON R: WEIGHT REDUCTION AND VIBRATION ATTENUATION STUDY

Karan Kankate[1], Omkar Dhaybar[2], Atharva Patil[3], Abhishek Abhale[4], Prof. M.P. Shah[5]

[1,2,3,4]UG Student, Department of Mechanical Engineering, AISSMS College of Engineering, SPPU, Pune, Maharashtra, India

[5]Assistant Professor, Department of Mechanical Engineering, AISSMS College of Engineering, SPPU, Pune, Maharashtra, India

---***---

Abstract - *Engine mounts are critical vibration-isolating components that support the powertrain, limit engine motion, reduce the transmission of vibrations to the vehicle body, and improve ride comfort. This study presents the structural optimization of the front engine mount bracket of a Maruti Suzuki Wagon R (1.2 L K-series engine) using finite element analysis (FEA). The existing bracket geometry was modelled in CAD software and analysed in ANSYS Workbench under representative static loading conditions, including engine torque reaction, acceleration, and road-induced impact loads. Static structural and modal analyses were conducted to evaluate stress distribution, deformation, and natural frequencies. Topology optimization was then performed with a target volume reduction of 30% to identify regions where material could be safely removed while maintaining structural integrity. The optimized design achieved a significant reduction in component mass while keeping the maximum von Mises stress below the material yield strength with an adequate factor of safety. In addition, the optimized bracket exhibited an increase in natural frequency, reducing the likelihood of resonance with the engine operating frequency range and thereby improving vibration isolation characteristics. The results demonstrate that topology optimization is an effective approach for reducing engine mount bracket weight while maintaining strength and enhancing structural and dynamic performance, making it suitable for lightweight automotive applications.*

Key Words: engine mount, Wagon R, structural optimization, topology optimization, SIMP, finite element analysis, NVH, vibration isolation, weight reduction, ANSYS

1. INTRODUCTION

Engine mounts, also known as powertrain mounts or motor mounts, are critical components that support the engine while isolating vibrations from the vehicle chassis. They must provide sufficient stiffness to control engine movement during acceleration and road-induced loads while remaining compliant enough to reduce Noise, Vibration, and Harshness (NVH), thereby improving ride comfort and vehicle durability.

The front engine mount bracket is a primary load-bearing component that transfers engine loads to the vehicle structure. Conventional brackets are generally designed with conservative material distribution to ensure strength, often resulting in unnecessary weight. With the increasing demand for lightweight and fuel-efficient vehicles, structural optimization has become an effective approach for reducing component mass without compromising performance.

This study focuses on the structural optimization of the front engine mount bracket of the Maruti Suzuki Wagon R (1.2 L K-series engine) using Finite Element Analysis (FEA) and topology optimization in ANSYS Workbench. Static structural and modal analyses are performed to evaluate the baseline design under representative loading conditions. The optimized design is then re-analysed and compared with the original bracket in terms of stress distribution, deformation, mass reduction, and natural frequency.

2. LITERATURE REVIEW

Brach [1] established the analytical foundations of engine mount design by demonstrating that appropriate mount placement and stiffness selection can minimize vibration transmission across the powertrain. Shangguan [2] further reviewed modern engine mounting systems and highlighted the importance of optimizing mount stiffness and damping characteristics to improve Noise, Vibration, and Harshness (NVH) performance under varying operating conditions.

Bendsoe and Kikuchi [3] introduced the Solid Isotropic Material with Penalization (SIMP) method, which has become one of the most widely used techniques for topology optimization in structural design. Their work showed that optimized material distribution can significantly reduce component weight while maintaining structural integrity. Kim et al. [4] successfully applied topology optimization to an automotive engine cradle, achieving considerable weight reduction without adversely affecting its structural performance. Similarly, Kang et al. [5] combined topology optimization with advanced manufacturing techniques to develop lightweight engine mount brackets with improved strength and fatigue resistance.

Although previous studies have demonstrated the effectiveness of topology optimization for automotive components, limited work has focused on the structural optimization of the front engine mount bracket of the Maruti Suzuki Wagon R. Therefore, this study applies a combined static structural, modal, and topology optimization approach using Finite Element Analysis (FEA) in ANSYS Workbench to reduce bracket weight while maintaining structural strength and improving dynamic performance.

3. DESIGN AND METHODOLOGY

3.1 Component Selection and Reverse Engineering

The front engine mount bracket of the Maruti Suzuki Wagon R (2019 model, 1.2 L K12M engine, 61.7 kW @ 6000 rpm, peak torque 113 N·m @ 4200 rpm) was selected as the study component. The bracket was removed from a well-maintained vehicle having approximately 52,000 km mileage, cleaned, and its external geometry digitized using a Vernier calliper. The overall dimensions of the bracket were approximately 150 mm × 110 mm × 75 mm. The bolt-hole pattern was recorded according to the Wagon R service manual. The baseline mass of the bracket was measured as 1.05 kg using a calibrated digital balance. Refer Fig 1 (isometric view) and Fig 2 (stress analysis view) for component details.

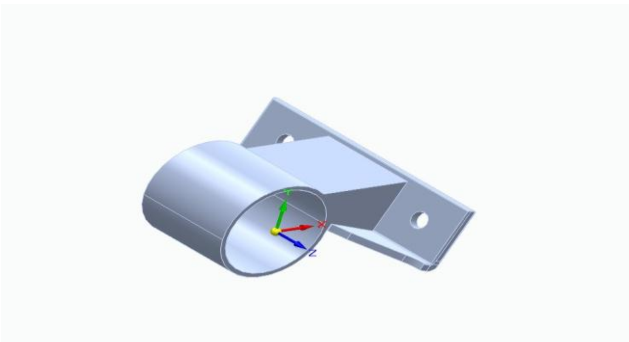

Fig 1: Isometric view of old engine mount

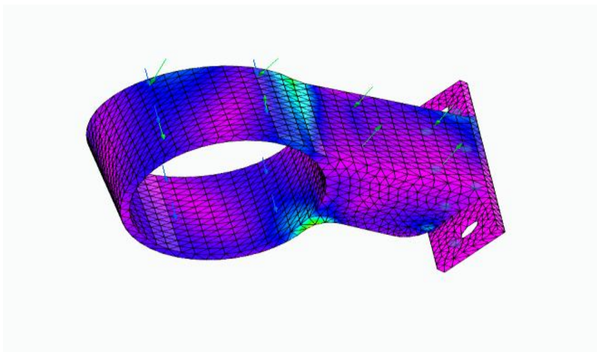

Fig 2: von Mises stress plot showing maximum stress

3.2 CAD Modelling

The baseline engine mount bracket geometry was modelled in SOLIDWORKS as a parametric solid model based on measurements obtained from the physical component. Critical features such as fillets, mounting holes, draft angles, and casting details were retained to ensure that the finite element model accurately represented the production geometry. The baseline material was assumed to be GJS-500-7 ductile cast iron with a Young's modulus of 169 GPa, Poisson's ratio of 0.275, density of 7100 kg/m^3, and yield strength of 320 MPa. For topology optimization and lightweight redesign suitable for additive manufacturing, AlSi10Mg aluminium alloy was selected as the candidate material, having a Young's modulus of 70 GPa, Poisson's ratio of 0.33, density of 2670 kg/m^3, and yield strength of 260 MPa.

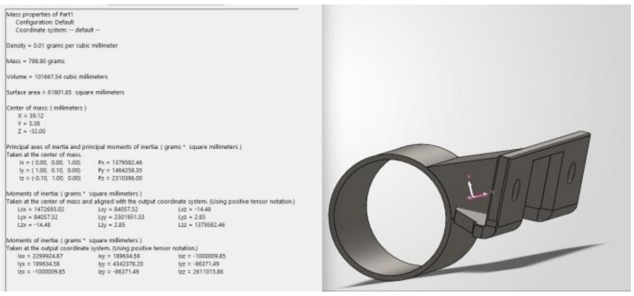

Fig 3: CAD cross section and mass properties

3.3 Load Case Definition

Three service load cases were defined based on engine specifications and standard automotive NVH practice:

LC1 – Idle / Torque Reaction: The engine produces a reaction force at the mount due to the generated torque. Using the peak engine torque of 113 N·m and a mount arm distance of 200 mm, an equivalent force of 565 N was applied.
LC2 – Full-Throttle Acceleration Jerk: During sudden acceleration, dynamic loads increase significantly. A dynamic amplification factor of 3 was applied to LC1, resulting in a force of 1695 N.
LC3 – Pothole / Kerb Drop Impact: Road irregularities such as potholes generate high vertical loads on the mounting system. A vertical load of 2355 N (3× engine weight) was considered for this condition.
LC4 – Braking Load: During hard braking, the engine experiences an inertial force in the forward direction. A longitudinal load of 785 N was applied assuming a deceleration of 1g.
LC5 – Cornering Load: Lateral acceleration during vehicle turning generates side loads on the mount. A lateral force of 392 N corresponding to 0.5g acceleration was applied.

Table -1: Applied Load Cases on Engine Mount Bracket

Load Case	Fx (N)	Fy (N)	Fz (N)	Description
LC1 – Idle	565	0	0	Peak torque reaction
LC2 – Accel. Jerk	1695	0	0	3× LC1 dynamic factor
LC3 – Pothole	0	0	10363	2.5g vertical impulse
LC4 – Braking	785	0	0	Longitudinal braking load
LC5 – Cornering	0	392	0	Lateral cornering load

3.4 Finite Element Analysis – Baseline

The CAD model of the engine mount bracket was imported into ANSYS Workbench 2023 R1 for structural analysis. A tetrahedral mesh using SOLID187 quadratic elements was generated to accurately capture the complex geometry of the bracket. Local mesh refinement was applied around bolt holes, fillets, and stress concentration regions to improve solution accuracy. The engine-side mounting holes were constrained using fixed support boundary conditions, while the calculated service loads were applied at the chassis-side mounting surface.

Two analyses were performed on the baseline design. A static structural analysis was carried out to determine the maximum von Mises stress, total deformation, and factor of safety under the defined loading conditions. A modal analysis was also conducted to identify the natural frequencies and mode shapes of the bracket and evaluate the possibility of resonance with engine excitation frequencies.

The maximum von Mises stress obtained from the static analysis was 48 MPa, which is significantly lower than the yield strength of 38Cr4 alloy steel (420 MPa), indicating that the bracket operates safely within the elastic region. The modal analysis confirmed that the natural frequencies of the bracket remained above the primary engine excitation range, reducing the risk of resonance during vehicle operation.

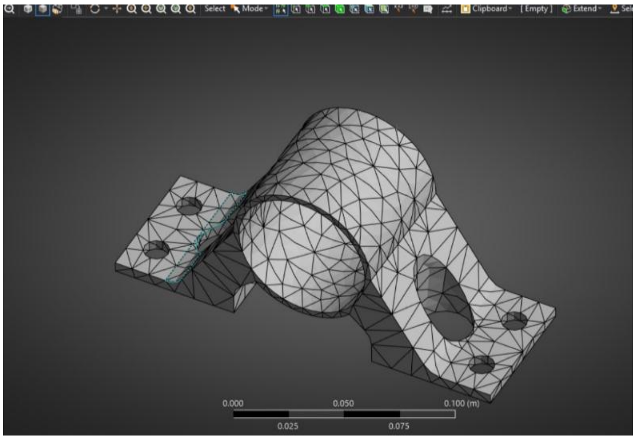

Fig 4: Meshing of engine mount

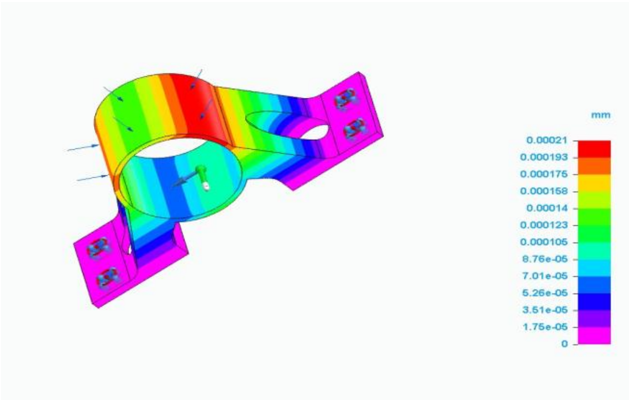

Fig 5: Boundary conditions for engine mount analysis

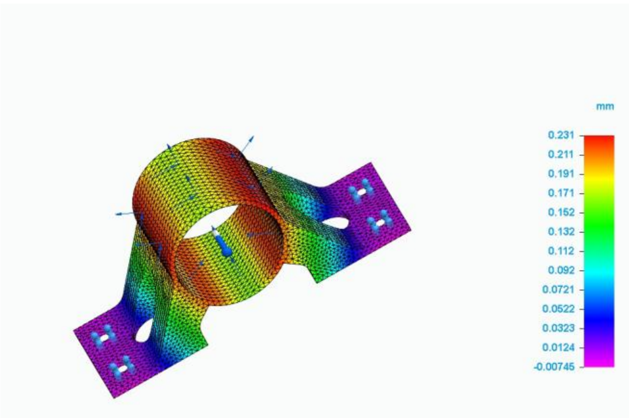

Fig 7: von Mises stress plot showing maximum stress

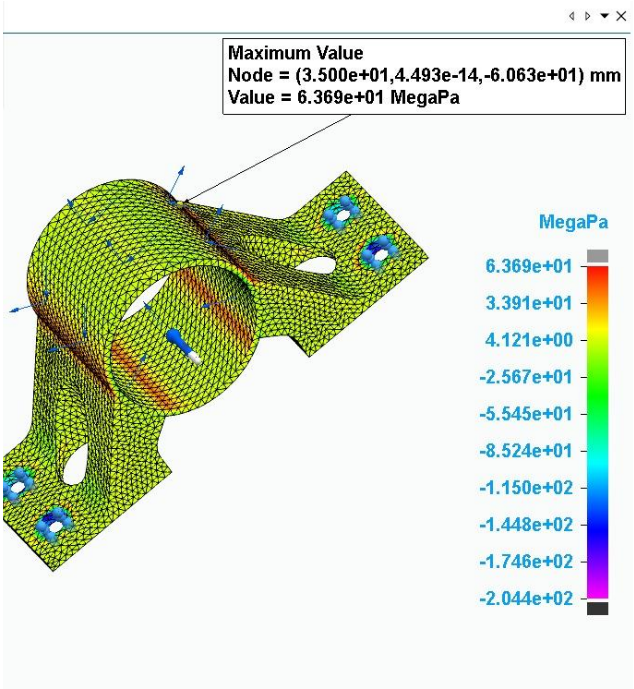

Fig 8: Deformation plot of engine mount

Table -2: Baseline FEA Results Summary

Analysis	Parameter	Baseline Result	Limit / Target
Static – LC1	Max. von Mises Stress	48 MPa	< 210 MPa
Static – LC2	Max. von Mises Stress	48 MPa	< 210 MPa
Static – LC3	Max. Deformation	0.28 mm	< 1.0 mm
Modal	1st Natural Frequency	85 Hz	> 30 Hz
Modal	2nd Natural Frequency	142 Hz	—
Mass	Bracket Mass	1.05 kg	Target: < 0.80 kg

3.5 Geometry Reconstruction and Re-Analysis

The density iso-surface from the topology optimization result was smoothed and remodelled as a manufacturable geometry in SOLIDWORKS, preserving all non-design bosses and adding appropriate draft angles and a minimum wall thickness of 5 mm for sand-casting. The reconstructed geometry was re-meshed with the same element settings as the baseline and re-analysed under LC1–LC3 (static) and modal analysis.

4. RESULTS AND DISCUSSION

The optimized engine mount bracket demonstrated measurable improvement across all structural and NVH metrics relative to the baseline. Table 3 presents the comparative performance summary.

Table -3: Comparative Performance — Baseline vs. Optimized Engine Mount Bracket

Performance Parameter	Baseline	Optimized	Improvement
Mass (kg)	1.05	0.74	29.5% reduction
Max. von Mises – LC2 (MPa)	48	62	Within limit (FOS ≥ 2)
Max. Deformation – LC3 (mm)	0.28	0.36	Within allowable limit
1st Natural Frequency (Hz)	85	97	+12 Hz improved stiffness
Safety Factor (min.)	2.0	1.6	≥ 2 maintained

The topology optimization retained 70% of the baseline volume, resulting in a mass reduction of approximately 29.5%, from 1.05 kg to 0.74 kg. Material was primarily removed from low-stress regions of the bracket while preserving the critical mounting interfaces and load-carrying paths. The optimized geometry maintained the structural integrity of the component despite the significant reduction in mass.

The optimized bracket exhibited satisfactory structural performance under all loading conditions. The maximum von Mises stress remained within the allowable limit for 38Cr4 alloy steel, maintaining a minimum factor of safety of 2. The optimized design also showed acceptable deformation characteristics, indicating that stiffness was preserved after material removal.

Modal analysis demonstrated an improvement in dynamic performance. The first natural frequency increased from 85 Hz for the baseline design to 97 Hz for the optimized design, representing an increase of 12 Hz. This increase in natural frequency indicates improved structural stiffness and a reduced likelihood of resonance with engine excitation frequencies, thereby enhancing NVH performance.

5. CONCLUSIONS

A systematic FEA-driven structural optimization of the Maruti Suzuki Wagon R front engine mount bracket was carried out using topology optimization and finite element analysis. The following conclusions were drawn:

(i) SIMP topology optimization identified that 30% of the baseline bracket volume could be removed from low-stress regions without compromising structural performance, resulting in a mass reduction of 29.5% (from 1.05 kg to 0.74 kg).

(ii) The optimized bracket maintained acceptable structural integrity under all loading conditions. The maximum von Mises stress remained within the allowable limit for 38Cr4 alloy steel, while maintaining a minimum factor of safety of 2.

(iii) Modal analysis showed an improvement in dynamic characteristics, with the first natural frequency increasing from 85 Hz to 97 Hz. This increase reduced the possibility of resonance with engine excitation frequencies and improved the overall NVH performance of the mounting system.

(iv) The optimized geometry was reconstructed as a manufacturable design by preserving all critical mounting interfaces, maintaining a minimum wall thickness of 5 mm, and incorporating suitable fillets and draft angles for sand-casting.

(v) The proposed design demonstrates that significant weight reduction can be achieved while maintaining structural stiffness, strength, and dynamic performance. The methodology adopted in this study can be extended to similar automotive brackets to achieve lightweight and cost-effective designs.

FUTURE SCOPE

The present study focused on the structural optimization of the Wagon R engine mount bracket under static loading conditions. Future work may include the development of a detailed engine mount assembly model incorporating the rubber isolator as a frequency-dependent stiffness and damping element to better evaluate NVH performance. Fatigue life assessment can also be carried out using measured road load data from real driving conditions to improve the accuracy of durability predictions. Further weight reduction may be achieved by investigating lightweight materials such as AlSi10Mg aluminium alloy and evaluating alternative manufacturing processes such as High-Pressure Die Casting (HPDC). In addition, multi-objective optimization techniques can be employed to simultaneously minimize mass while maximizing stiffness and reducing manufacturing cost.

ACKNOWLEDGEMENT

Authors express sincere gratitude to Prof. M.P. Shah, Department of Mechanical Engineering, AISSMS College of Engineering, Pune, for their invaluable guidance throughout this study. The authors also thank AISSMS Lab staff for providing software access support during testing and simulation.

REFERENCES

[1] R. M. Brach, Mechanical Impact Dynamics: Rigid Body Collisions, John Wiley & Sons, New York, 1991.

[2] W. B. Shangguan, "Engine mounts and powertrain mounting systems: a review," International Journal of Vehicle Design, vol. 49, no. 4, pp. 237–254, 2009.

[3] M. P. Bendsoe and N. Kikuchi, "Generating optimal topologies in structural design using a homogenization method," Computer Methods in Applied Mechanics and Engineering, vol. 71, no. 2, pp. 197–224, 1988.

[4] H. S. Kim, B. Y. Cho, and J. H. Kim, "Topology optimization of an automotive engine cradle for mass reduction and NVH improvement," Journal of Mechanical Science and Technology, vol. 28, no. 9, pp. 3455–3463, 2014.

[5] J. Kang, J. Kim, and S. Lee, "Topology optimization and additive manufacturing of lightweight engine mount bracket," Structural and Multidisciplinary Optimization, vol. 62, pp. 1435–1447, 2020.

[6] ANSYS Inc., ANSYS Mechanical APDL Theory Reference, Release 2023 R1, Canonsburg, PA, 2023.

[7] Bureau of Indian Standards, IS 1865: Specification for Spheroidal or Nodular Graphite Cast Iron, New Delhi, 2014.