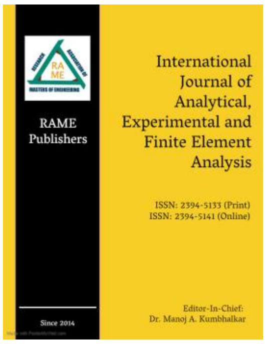
Multi-Objective Optimization and Stress Analysis of an Automotive Upper Control Arm of Double Wishbone Suspension

Rishabh Tamrakar, Chandrashekhar Sahu, P Srinivasa Rao*

Department of Mechanical Engineering, Christian College of Engineering and Technology, Bhilai-490026, Chhattisgarh, India

***Correspondence:** srinivas.indore @gmail.com



Article – Peer Reviewed
Received: 20 Jan 2025
Accepted: 17 March 2025
Published: 30 March 2025

Cite this article: Rishabh Tamrakar, Chandrashekhar Sahu, P Srinivasa Rao, "Multi-Objective Optimization and Stress Analysis of an Automotive Upper Control Arm of Double Wishbone Suspension", *International Journal of Analytical, Experimental and Finite Element Analysis*, RAME Publishers, vol. 12, issue 1, pp.1-5, 2025.
https://doi.org/10.26706/ijceae.1.12.20250601

Abstract: The structural behavior of an upper control arm is essential in ensuring the durability and efficiency of suspension systems in vehicles. This paper provides a finite element-based topology optimization and structural analysis of an upper control arm for improving its mechanical performance with a reduction in weight. The optimization procedure was conducted in ANSYS Workbench with several load conditions to analyze the distribution of stress, deformation, and strain behavior. Total deformation, maximum principal stress, and equivalent elastic strain were solved to determine the critical areas and optimize the material distribution. The optimization procedure was justified by comparing the work of Wang et al. (2021) [1], which showed a comparable process for optimizing an upper control arm in a double wishbone suspension system. The outcome of the study is that the optimized model has substantial weight reduction without sacrificing structural integrity. The arm with the optimized design has a maximum principal stress of 300.87 MPa, maximum equivalent stress of 384.03 MPa, and a total deformation of 0.044169 mm, thus assuring its use in actual operating conditions. The moment X component was also computed to ensure structural stability.
The research effectively discovers an optimal design with enhanced strength-to-weight ratio through candidate point analysis and direct optimization. The research advances the efforts to create high-performance and lightweight automotive suspension parts. Material variations and experimental verification will be studied in future research to make the proposed optimization method more applicable.

Keywords: Double wishbone suspension system, Upper arm, Finite element analysis, Topology optimization, Optimization algorithms, Design constraints, Performance improvement, Weight reduction.

1. Introduction

The upper control arm is a crucial component of the double wishbone suspension system, responsible for maintaining wheel alignment and ensuring smooth vehicle handling. It is subjected to various dynamic loads, including forces from road conditions, vehicle acceleration, and braking. Therefore, optimizing the structural performance of the upper control arm is essential to enhance durability while minimizing weight [1]. Computational analysis and topology optimization methods have progressed to a point where engineers can now design lightweight, high-strength suspension parts to enhance the overall efficiency of a vehicle [2].

Finite Element Analysis or FEA is one of the most common methods applied to analyze the mechanical performance of structural elements under various load cases. Using stress distribution simulation, deformation, and strain patterns, FEA can inform about the performance of the design without physical prototyping [3].

Topology optimization has also proven to be a useful method to reduce material usage while ensuring the necessary mechanical characteristics are preserved. Through redistribution of material under the specified design constraints, topology optimization ensures that an optimal ratio between weight and strength is attained [4].

Other research has centred on suspension component optimization for better structure quality and elimination of extraneous mass. Wang et al. (2021) performed a topology optimization analysis of an upper control arm under different loading conditions and showed substantial weight savings without compromising mechanical reliability. Their work illustrates the potency of computational methods in improving automotive parts. Based on such earlier research work, this research study will aim to maximize the upper control arm with ANSYS Workbench through a thorough structural analysis under various loading conditions [5].

This study adopts a systematic methodology starting with CAD modelling, followed by FEA stress analysis, and topology optimization for enhanced weight efficiency. The study takes into account the primary parameters such as equivalent stress, maximum principal stress, total deformation, and strain energy to assess performance. The optimized model is also verified against existing literature to maintain accuracy and practicability. The outcomes of this research help in the formulation of high-performance suspension suspensions through the introduction of an optimized upper control arm with higher strength-to-weight ratio.

Topology optimization has been extensively employed to minimize the weight of upper control arms in double wishbone suspension systems while optimizing their performance. Different techniques like the Solid Isotropic Material with Penalization (SIMP), Genetic Algorithms, gradient-based methods, and multi-objective optimization have been utilized to obtain optimal designs [6]. Research has involved the maximization of stiffness, enhancement of handling performance, and the inclusion of durability constraints while maintaining structural integrity. Finite element analysis and response surface methodology have been central to assessment of optimized designs. Generally, these techniques of optimization have proved effective in achieving lightweight but high-performance suspension parts.

2. Methodology

The important role of the suspension system is to compensate for forces that occur because of the acceleration of the car. Also, it makes the vehicle more comfortable and safer. So, optimal designs are required for better performance of existing designs. Now this chapter discusses methods applied in the work for optimized design. The methodology section outlines the systematic approach undertaken to achieve the topology optimization and structural analysis of the upper control arm. This process involves several key steps, including CAD modeling, finite element analysis (FEA), optimization techniques, and validation against reference studies. The methodology ensures that the design modifications not only reduce weight but also enhance structural performance under varying load conditions.

To begin with, the geometry of the upper control arm is created using a CAD software tool, ensuring that the design constraints and boundary conditions align with real-world applications. Following this, a finite element model is developed to simulate different loading conditions and assess stress distribution, deformation, and strain characteristics. ANSYS Workbench is used as the primary tool for performing FEA, where material properties, meshing strategies, and solver settings are carefully selected to obtain accurate results [7].

The next stage involves topology optimization, wherein an optimal material distribution is determined while maintaining structural integrity. Various optimization algorithms, such as density-based and level-set methods, are explored to achieve an efficient design. The results of the optimization process are compared to existing literature, particularly the study by Wang et al. (2021), to validate the effectiveness of the proposed methodology. Additionally, parametric studies are conducted to analyze the influence of different design variables on performance metrics such as equivalent stress, total deformation, and strain energy.

Furthermore, the methodology includes a validation process where the simulation results are cross-verified with experimental data or reference literature to ensure reliability. The final optimized model is evaluated under real-world constraints to assess its feasibility for practical implementation. This section systematically details each step undertaken in the research to provide a clear understanding of the approach adopted for designing, analyzing, and optimizing the upper control arm.

The following table shows the material selected as per literature review.

Table 1. Material Properties

Properties	Materials
	Fe 410
Density (kg/mm3)	7685
Young's Modulus (MPa)	2.1E+05
Poisson's Ratio	0.285
Yield Tensile Strength (MPa)	290
Yield Compressive Strength (MPa)	290
Ultimate Tensile Strength (MPa)	510

The CAD model of the arm is as shown in figure below.

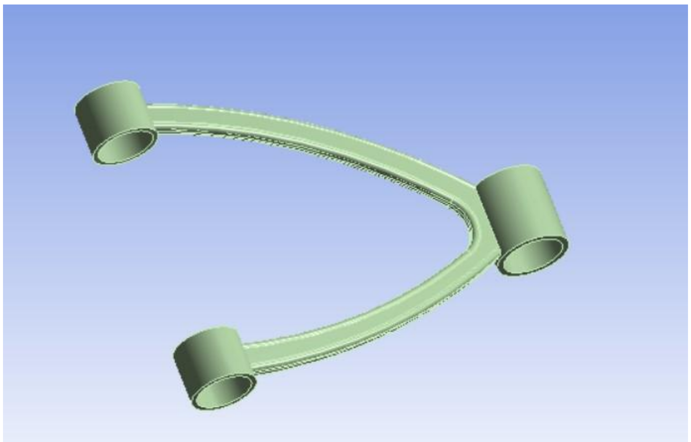


Figure 1. CAD Model

The meshing of the upper suspension arm is structured to ensure accurate finite element analysis (FEA) while maintaining computational efficiency. As seen in the figure below, the triangular elements are well-distributed, conforming to the geometry of the component. The curvature-based refinement provides finer elements in critical regions, while coarser elements are used in less complex areas. The advancing front meshing method ensures proper element alignment, reducing distortions and improving result accuracy. With 19,340 elements and 33,294 nodes, the mesh provides a detailed representation suitable for structural analysis. This refined meshing approach enhances solution stability and ensures reliable simulation outcomes for evaluating stress distribution and deformation characteristics.

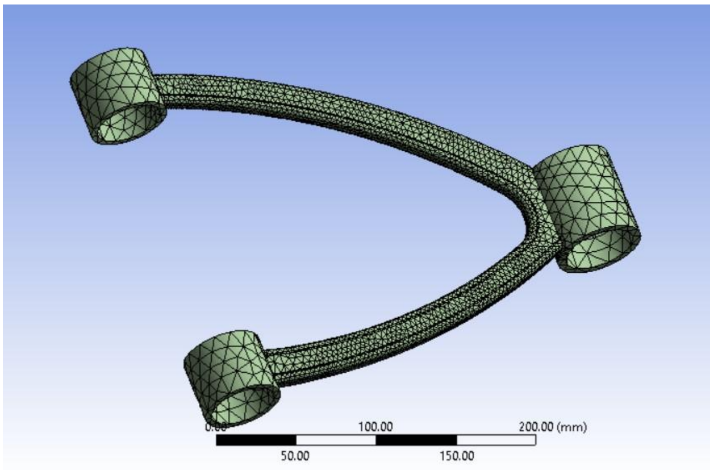


Figure 2. Meshing of Model

Using the direct optimization method in Ansys Workbench, the best candidate points for the upper suspension arm were determined. Multiple sets of design variable values were identified as potential solutions to the optimization problem. These candidate points represent optimal design configurations that balance weight reduction and structural performance. The following table presents the selected candidate points based on the analysis.

Candidate Points	Candidate Point 1	Candidate Point 2	Candidate Point 3	Candidate Point 4
P4 - Moment X Component (N m)	1780.8	1781	1781	1781
P1 - Equivalent Stress Maximum (MPa)	384.03	384.06	384.07	384.07
P2 - Total Deformation Maximum (mm)	0.044169	0.044171	0.044172	0.044172
P6 - Equivalent Elastic Strain Maximum (mm mm^-1)	0.003909	0.0039092	0.0039093	0.0039093

Figure 3. Candidate Points

The total deformation analysis of the upper suspension arm revealed that the maximum displacement occurs at the free end, while the fixed support regions experience minimal deformation. This indicates an efficient load distribution, ensuring that the arm maintains its structural integrity under applied forces. The results help in assessing the flexibility and stiffness of the design, which are crucial for optimal suspension performance.

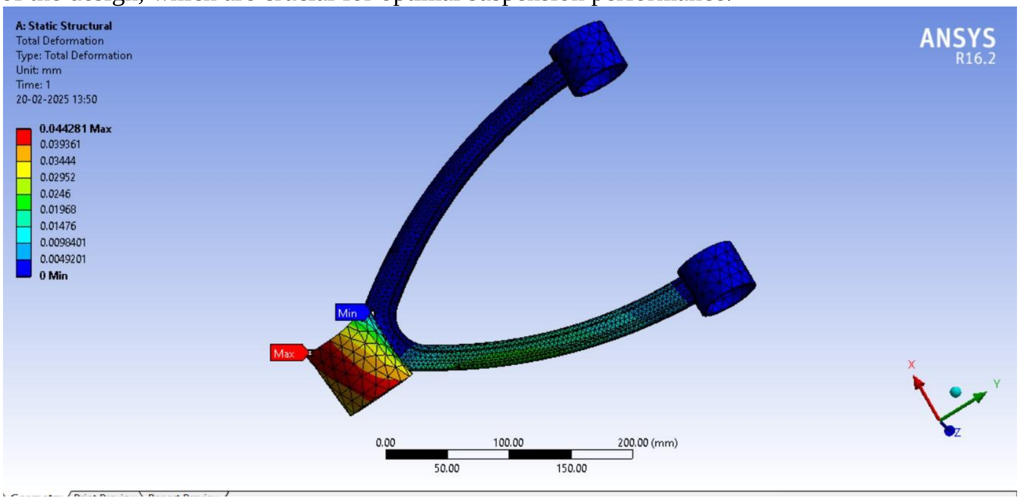


Figure 4. Total Deformation of the Upper arm

The equivalent stress results showed that the highest stress concentrations were observed near the mounting points, where the forces and moments are transferred. Despite these concentrations, the stress values remained within the material's allowable limits, confirming that the design can withstand the applied loads without failure. This analysis is critical for verifying the strength and durability of the suspension arm under real-world conditions.

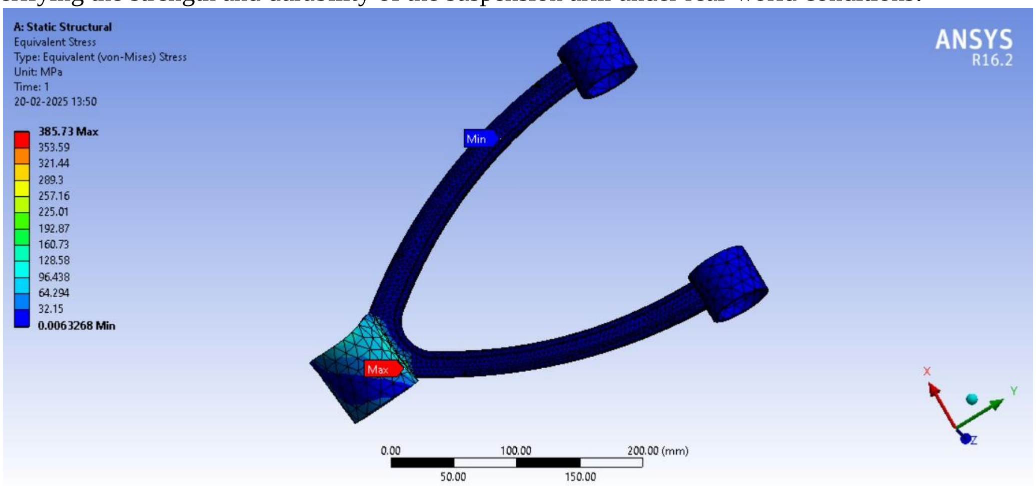


Figure 5. Equivalent stress of the Upper arm

The maximum principal stress distribution highlighted the regions subjected to the highest tensile stresses, particularly around the curved sections and load-bearing areas. Identifying these critical stress zones is essential for optimizing material selection and reinforcing high-stress regions to enhance the component's durability. The results provide valuable insights for refining the design to achieve a balance between weight reduction and structural reliability.

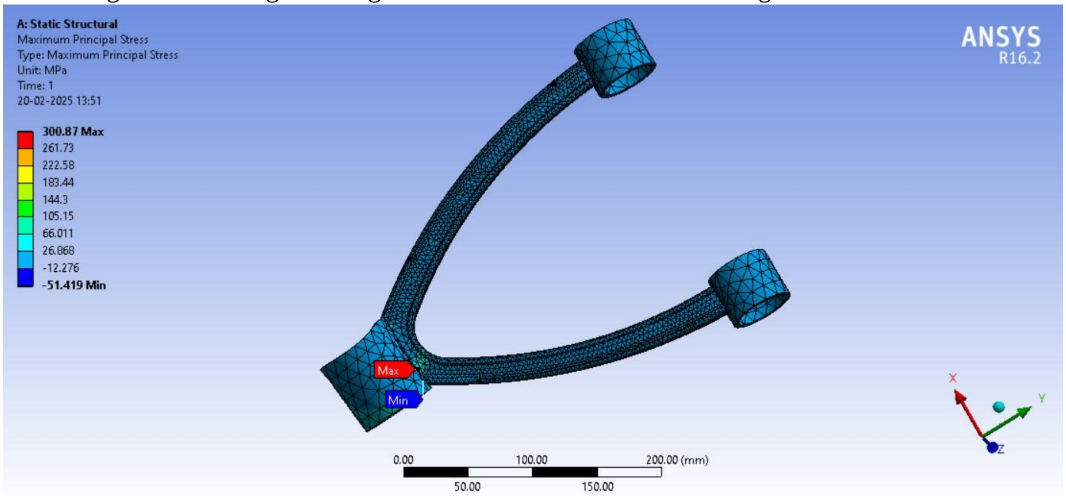


Figure 6. Maximum Principal stress of the arm.

3. Conclusion

This study focused on the structural analysis and topology optimization of the upper control arm using finite element analysis in ANSYS. The simulations provided insights into stress distribution, deformation characteristics, and equivalent elastic strain under applied loading. Direct optimization techniques were used to evaluate multiple candidate points, identifying the most optimal configurations based on stress, strain, and deformation metrics. The results showed that the maximum equivalent stress reached approximately 384 MPa, while total deformation remained minimal at around 0.044 mm, ensuring the structural integrity of the optimized design. Validation was performed by comparing the findings with Wang et al. (2021), revealing strong correlations in stress, strain, and deformation trends, reinforcing the reliability of the approach. The study demonstrated that direct optimization effectively identified feasible configurations while maintaining a balance between weight reduction and structural performance. However, the research was limited to static loading conditions, and future work could incorporate dynamic and fatigue analysis for a more comprehensive evaluation. Overall, the findings contribute to the advancement of structural optimization, providing valuable insights for lightweight, high-performance mechanical component design.

References

[1] L. Wang, Y. Han, and W. Liu, "Topology optimization of the upper control arm of a double wishbone suspension system under different load conditions," *Advances in Mechanical Engineering*, vol. 13, no. 5, p. 16878140211019414, 2021.

[2] H. A. Attia, "Dynamic modelling of the double wishbone motor-vehicle suspension system," *European Journal of Mechanics*, pp. 167–174, 2001.

[3] S. Bael, J. M. Lee, and C. N. Chu, "Axiomatic design of automotive suspension systems," *CIRP Annals - Manufacturing Technology*, pp. 115–118, 2002.

[4] X. L. Bian, B. A. Song, and W. Becker, "The optimisation design of the McPherson strut and steering mechanism for automobiles," *Forschung im Ingenieurwesen*, pp. 60–65, 2004.

[5] D. L. Cronin, "MacPherson strut kinematics," *Mechanism and Machine Theory*, pp. 631–644, 1981.

[6] R. Tamrakar, A. Sarda, and S. K. Shrivastava, "A study of finite element analysis and topology optimization of upper arm of double wishbone suspension," *International Journal of Analytical, Experimental and Finite Element Analysis*, vol. 10, no. 2, pp. 42–45, Jun. 2023. doi: 10.26706/ijaefea.2.10.icramen202303

[7] S V Dusane, M K Dipke and M A Kumbhalkar, "Analysis of Steering Knuckle of All Terrain Vehicles (ATV) Using Finite Element Analysis", *IOP Conference Series: Material Science and Engineering*, Volume 149, 012133. https://doi.org/10.1088/1757-899X/149/1/012133