International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 08 Issue: 07 | July - 2024 SJIF Rating: 8.448 ISSN: 2582-3930

A Review of Stress Analysis and Advanced FEA Techniques in Automotive Suspension Systems

Pradip Diwan Borase
Department of Mechanical Engineering
Orientatal Institute of Science and Technology, Bhopal
pradipborase32@gmail.com

Vivek Babele
Department of Mechanical Engineering
Orientatal Institute of Science and Technology, Bhopal
vivekbabele@oriental.ac.in

Girish Kumar Khare
Department of Mechanical Engineering
Orientatal Institute of Science and Technology, Bhopal
girishkhare@oriental.ac.in

Abstract- **The lower control arm (LCA) is an essential component of a vehicle's suspension system, responsible for maintaining stability, improving handling, and enhancing passenger comfort. This paper examines different approaches and findings regarding the stress analysis and optimisation of LCAs using CAD modelling and Finite Element Analysis (FEA). The analysis primarily aims to identify regions with high stress concentration, specifically in very near proximity to the top mounting point. It also examines various optimisation techniques, such as topology and shape optimisation, surrogate modelling, and multi-objective optimisation. Significant studies emphasise the efficacy of these techniques in decreasing weight, enhancing structural strength, and enhancing fatigue resistance. By combining dynamic analysis, material selection, and advanced optimisation methods, substantial enhancements in LCA performance and cost efficiency are attained. This review offers a thorough comprehension of current methodologies and upcoming approaches for enhancing life cycle assessments (LCAs) in automotive suspension systems.**

Keywords- **Lower Control Arm (LCA), Suspension System, Stress Analysis, Finite Element Analysis (FEA), Topology Optimization, Structural Integrity, CAD.**

I. INTRODUCTION

In an automobile, the front control lower arm, often referred to simply as the lower control arm (LCA), is a vital component of the vehicle's suspension system. This part connects the wheel hub and steering knuckles to the car's chassis, playing a key role in supporting the vehicle's weight and ensuring proper wheel alignment. The LCA allows the wheels to move up and down independently, absorbing shocks from road irregularities, which contributes to a smoother ride and better handling. Constructed typically from high-strength steel or aluminum, the lower control arm features bushings at the chassis end and a ball joint at the wheel end. The bushings provide flexible mounting, reducing vibrations and noise, while the ball joint enables the necessary pivoting action for steering and suspension movements.

By maintaining correct wheel alignment and enabling controlled movement, the front lower control arm significantly enhances the vehicle's stability, handling, and overall driving comfort, making it an indispensable part of modern automotive engineering. In the intricate system of a car's suspension, the front control lower arm stands out as a crucial element, intricately designed to connect the vehicle's wheel hub and steering knuckles to the frame. This component, often made from robust materials like high-strength steel or aluminum, serves multiple essential functions. Primarily, it supports the vehicle's weight and maintains precise wheel alignment, which is fundamental for optimal handling, stability, and tire longevity.

The lower control arm allows the wheels to move vertically, absorbing and dampening shocks from uneven road surfaces. This controlled movement not only enhances ride comfort by minimizing the impact felt within the cabin but also plays a significant role in maintaining traction and stability during driving. The front lower control arm is engineered with bushings at the chassis end and a ball joint at the wheel end. The bushings, typically made from rubber or polyurethane, provide a flexible yet secure connection, reducing the transmission of road vibrations and noise to the vehicle's body. The ball joint, on the other hand, allows for the necessary pivoting and articulation required during steering and suspension movements.

This dual functionality ensures that the vehicle can navigate turns and uneven surfaces with ease, maintaining a smooth and controlled ride. Additionally, the precise construction of the front control lower arm ensures that the wheels remain in the correct position relative to the road, which is crucial for the vehicle's directional stability and responsiveness. Thus, the front control lower arm is indispensable in modern automotive engineering, significantly contributing to the safety, comfort, and performance of the vehicle.

The suspension system of an automobile is a complex network designed to enhance stability, handling, and passenger comfort by absorbing and dissipating energy from road irregularities. It comprises various components, including springs (coil springs, leaf springs, and torsion bars), shock absorbers (dampers), control arms, sway bars, and bushings, all working in unison. Springs absorb the initial energy from bumps, while shock absorbers dampen the oscillations, converting kinetic energy into thermal energy to prevent excessive bouncing. Control arms, such as upper and lower control arms, maintain proper wheel alignment and allow controlled wheel movement, with the lower control arm (LCA) playing a crucial role in absorbing lateral forces. Sway bars improve stability during turns by distributing forces across the suspension, and bushings provide flexible mounting points, reducing noise and vibrations. Together, these components ensure continuous tire contact with the road, delivering a smoother ride, better handling, and improved vehicle safety [1].

Suspension systems can be categorised into two primary types: dependent (solid axle) and independent. A dependent suspension system consists of two wheels that are linked together by a trailing rod, resulting in any disturbances experienced by one wheel having an impact on the other. This design is highly durable; however, it may lead to a less comfortable riding experience. On the other hand, an independent suspension system enables individual wheels to move vertically without affecting the movement of other wheels, thereby reducing the impact on them. Independent suspension is frequently utilised in passenger cars and light trucks. It offers additional room for engine installation, permits greater wheel displacement, minimises steering vibrations, and improves passenger comfort [2].

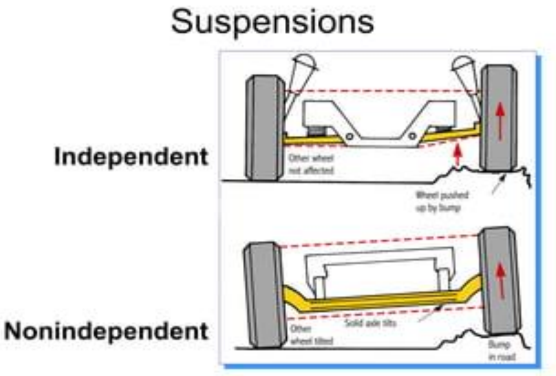

Figure 1 Automobile suspension system

II. LITERATURE REVIEW

A. Front Control Lower Arm Design Improvement

Lokesh Yadav and Imraan Siraj has focused on enhancing the design of front control lower arms, critical components in vehicle suspension systems that connect the wheel hub to the chassis. This improvement is achieved through a combination of experimental and numerical techniques. Experimental methods involve physical testing to validate theoretical models and assess real-world performance under various load conditions, including fatigue testing to ensure longevity. Numerical techniques, particularly Finite Element Analysis (FEA), are extensively used to simulate the behavior of these arms, predicting stress distribution, deformation patterns, and failure points. Optimization methods such as topology optimization and evolutionary structural optimization (ESO) further refine design parameters to improve performance metrics like durability and cost-effectiveness. Key considerations include material selection to optimize weight, strength, and corrosion resistance, crucial for enhancing structural efficiency and meeting safety standards [3].

B. Topology Optimization

Biglete et al., 2020 conducted a Weight Reduction and Performance Improvement: Topology optimization techniques are employed to redesign LCAs to reduce weight while maintaining or enhancing structural integrity. By altering the shape and material distribution, significant weight reductions can be achieved without compromising performance. For instance, a study on the MacPherson suspension system demonstrated a considerable weight reduction using topology optimization [4].

C. Unsprung Mass and Suspension System

Khode .et.al has explored the Unsprung mass, encompassing components like wheel axles, bearings, springs, shock absorbers, and lower control arms, directly impacts vehicle dynamics, ride quality, and noise levels. Optimization techniques, particularly Finite Element Analysis (FEA), are pivotal in evaluating and optimizing suspension components like the lower control arm. A case study focusing on the MacPherson suspension system illustrates this approach, using CAD software for solid modeling and ANSYS Workbench for detailed FEA analysis. By optimizing material selection and geometric configurations under static loading conditions, researchers aim to enhance stiffness-to-weight ratio and fatigue resistance, crucial for improving overall vehicle performance and efficiency [5].

D. Suspension System and Vehicle Dynamics

Santosh Kullur, S.N. Kurbet, Vishwanath Hugar, And Somappa Gudageri has conducted the suspension system which plays a critical role in ensuring vehicle comfort, safety, and handling characteristics. Research emphasizes the use of Finite Element Analysis (FEA) to analyze and optimize front lower control arms under static loading conditions. Using CAD tools for modeling and ABAQUS for simulation, researchers study stress distribution and deformation patterns to refine design parameters. Material optimization strategies are employed to reduce weight while maintaining structural integrity, essential for enhancing performance metrics such as stiffness and durability. This comprehensive approach contributes to advancing suspension system design in modern automobiles, addressing dynamic challenges and improving overall driving experience [6].

E. Fatigue Life Analysis for Lower Suspension Arm

Rashdan, Muhammad suggested Predicting fatigue life is crucial for ensuring the long-term reliability and durability of lower suspension arms in vehicles. Advanced computational techniques, including Finite Element Analysis (FEA) and strain-life methods, are employed for accurate fatigue life predictions. FEA simulations analyze stress distribution and deformation within the arm, while experimental strain data validate these predictions. Solidworks software aids in developing precise 3D models for simulation, integrating real-world conditions into computational analysis. Optimization efforts focus on improving fatigue resistance and extending operational lifespan through iterative design refinements and structural enhancements, ensuring robust performance under varying operational conditions [7].

F. Functional Characteristics of Control Arms in Automotive Suspension

Pachapuri concentrated on the Control arms which are integral components of automotive suspension systems, linking the chassis to the wheel hub and influencing vehicle dynamics, ride quality, and noise levels. These arms typically feature hinged suspension links that control radial distance and degrees of freedom, crucial for vehicle stability and maneuverability. Finite Element Analysis (FEA) plays a key role in analyzing dynamic characteristics under static and dynamic loads, identifying critical stress points and optimizing structural configurations to mitigate noise and vibration during vehicle operation. Case studies on McPherson suspension systems demonstrate modal and fatigue analyses, highlighting the importance of topological optimization to reduce weight while enhancing performance metrics like stiffness and durability. These advancements underscore the critical role of control arms in automotive engineering, continually improving vehicle performance and driving experience [8].

This literature review examines a range of studies that investigate the finite element analysis (FEA) of lower suspension arms and lower control arms in the automotive industry.

G. Analysis of the lower suspension arms using finite element methods.

In their study, Gadade and Todkar (2023) conducted a thorough examination of the stress analysis of an A-Type lower suspension arm using finite element methods. The primary goal was to ascertain the operational lifespan of the component when subjected to static loading conditions [9]. The lower suspension arm was designed and fabricated using CAD software, utilising AISI 1040 material. The process of finite element modelling and analysis was conducted utilising HYPERMESH software. The mesh was generated using 10-node tetrahedral elements. The study showcased the congruity between the theoretical outcomes derived from Finite Element Analysis (FEA) and the experimental outcomes obtained by subjecting the model to stress using a computerised universal testing machine (UTM). The wheel experienced a maximum static load of 5000N. Finite Element Analysis (FEA) revealed that the highest stress of 280 MPa occurred at the contact point between the wheel and the hub. However, actual experimental tests showed a slightly lower maximum stress value of 254 MPa. This confirmed the suitability of the designed model in real-world operating conditions.

H. Structural Analysis of Lower Control Arms

Kumar, Balaji, Balachandar, and Prem Kumar (2023) concentrated on the modelling and execution of structural analysis for a lower control arm employed in the front suspension system. The lower control arm, which is made of sheet metal, enables the vertical movement of the wheel and usually comprises a steel bracket that pivots on rubber bushings attached to the chassis [10]. Their study emphasised the structural soundness and efficiency of the lower control arm when subjected to different loading conditions.

I. Design Optimization Utilising Finite Element Analysis (FEA),

Liew, Hashim, and Rahman (2023) suggested a design optimisation approach for an aluminium cast lower control arm in the front suspension system [11]. The objective was to attain a 20% decrease in weight compared to the current steel component by utilising CATIA software for design and Hyperworks for structural analysis. Their findings demonstrated a 25% decrease in weight along with a fatigue life cycle of approximately 396,000 cycles. The new design successfully fulfilled the requirements for fatigue life and was considered appropriate for utilisation in C-segment passenger cars.

J. Structural Safety and Performance Evaluation

In their study, Singh and Bhushan (2023) performed a finite element analysis on the front lower control arm of a Macpherson type suspension system. The study employed Finite Element Analysis (FEA) techniques to forecast the structural efficacy of the design, a critical factor in ensuring the vehicle's stability and handling capabilities [12]. The lower control arm, made of hot-rolled steel (JSH 590B), was designed using CATIA software and then analysed using Hypermesh. The finite element model comprised approximately 14,561 elements, each with an average size of 2 mm. Following the process of meshing, a thorough quality check was conducted to ensure accuracy. Subsequently, the predetermined loading conditions and structural boundary constraints were implemented. The linear static structural analysis performed using Radioss Linear indicated that the stresses and displacements remained within acceptable thresholds, affirming the structure's ability to endure the applied load. The accuracy of these Finite Element Analysis (FEA) predictions was confirmed through comparison with empirical data.

The reviewed literature demonstrates the effectiveness of FEA in analyzing and optimizing automotive suspension components, particularly lower suspension arms and lower control arms. These studies highlight the importance of accurate modeling, material selection, and validation through experimental testing to ensure the reliability and durability of these critical automotive parts. The continuous advancements in FEA tools and techniques will further enhance the design and performance of suspension systems in the automotive industry.

III. OBJECTIVE

The primary objective of this research is to comprehensively analyze the structural integrity and performance characteristics of a front control lower arm within an automotive suspension system. This investigation utilizes advanced CAD modeling techniques and Finite Element Analysis (FEA) to evaluate how the control arm responds under various loading conditions. By focusing on structural analysis, the study aims to enhance the design's durability, optimize its performance metrics, and ensure compliance with safety standards. Through detailed simulation and analysis, this research seeks to provide valuable insights into improving the overall efficiency and reliability of front control lower arms in automotive applications.

IV. METHODOLOGY

The methodology begins with the initial design phase using Creo Parametric software, where 2D sketches define the fundamental geometry and dimensions of the front control lower arm. These sketches are developed into a detailed 3D model within Creo, ensuring accuracy in geometry representation. The finalized model is then exported in IGES format to maintain compatibility with ANSYS Workbench, a leading software platform for FEA.

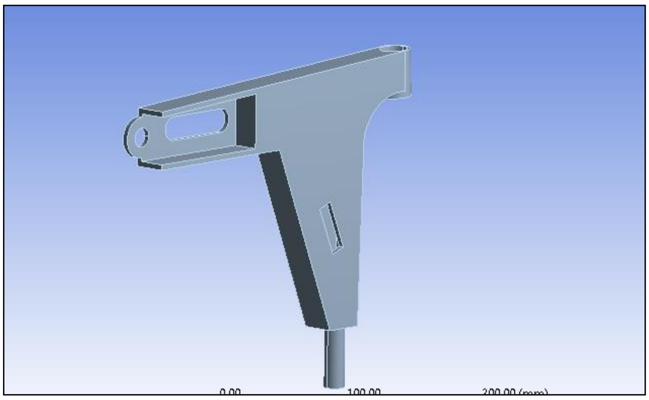

Figure 2 Design of front control lower arm

Upon import into ANSYS Workbench, the geometry undergoes rigorous checks for any inconsistencies or errors that could impact subsequent analyses. The next crucial step is meshing, where the model is divided into smaller, finite elements (tetrahedral elements in this case) to facilitate computational analysis.

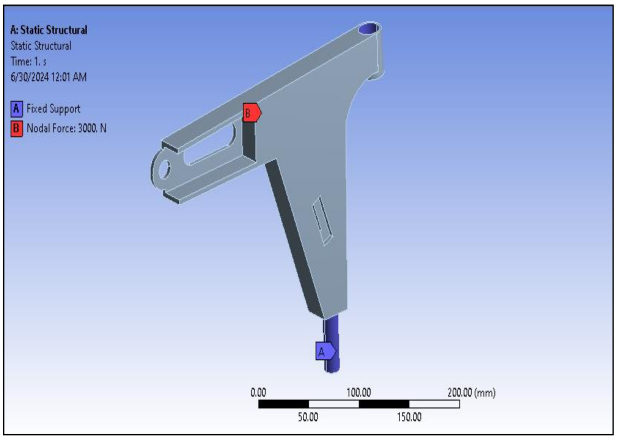

Figure 3 Meshed model of front control lower arm

Meshing is performed with careful consideration of element size and distribution to ensure accurate results. Areas with complex geometries or anticipated high stress gradients receive smaller mesh elements to capture detailed stress variations effectively. Techniques like adaptive meshing are employed to refine the mesh in critical regions, thereby enhancing analysis precision [6].

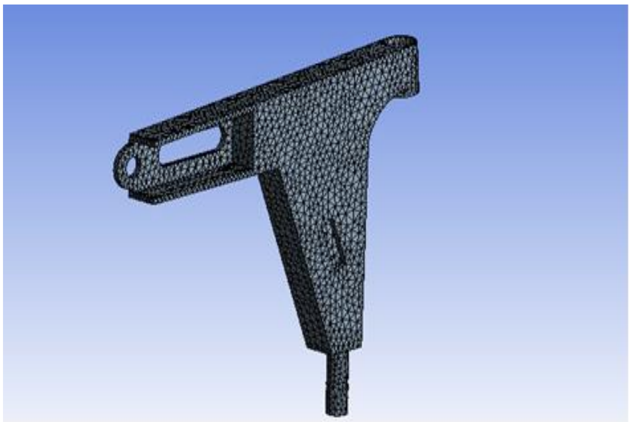

Figure 4 Structural boundary conditions

Structural boundary conditions are then applied to the model, simulating realistic operational scenarios. This includes fixing supports at the base of the control arm and applying nodal forces at specified points, as illustrated in Figure 4. ANSYS Workbench proceeds with the finite element analysis, solving equations that govern the control arm's mechanical behavior under the defined loads and boundary conditions. This iterative process calculates stress, strain, and displacement distributions throughout the model, providing comprehensive insights into its structural performance.

By systematically following this methodology, the study aims to validate the design of the front control lower arm, optimize its structural integrity, and contribute to advancing the field of automotive suspension system engineering through detailed simulation and analysis.

V. CONCLUSION

The front control lower arm (LCA) is an important component of a car's suspension system, serving an important role in maintaining wheel alignment, absorbing shocks, and ensuring overall vehicle stability and comfort. This review has highlighted the significance of the Life Cycle Assessment (LCA), specifically emphasising its role in enhancing vehicle performance by considering its intricate design and material composition. The LCA's capacity to bear the weight of the vehicle and regulate the movement of the wheel is essential in contemporary automotive engineering.

The utilisation of advanced analysis techniques, such as Finite Element Analysis (FEA), has demonstrated its immense value in improving the design and performance of LCAs. FEA enables accurate detection of potential failure locations and areas necessitating design enhancements by simulating stress distributions, deformation patterns, and fatigue life. By integrating CAD modelling and optimisation techniques, such as topology optimisation and evolutionary structural optimisation (ESO), these components are enhanced to enhance their durability and cost-effectiveness.

The analysis shows that the top mounting points of the LCA have high-stress regions that are crucial and may need design modifications, such as incorporating reinforcements or utilising materials with greater strength and fatigue resistance. The deformation analysis indicates that the overall deformation remains within acceptable thresholds. However, additional optimisation measures have the potential to improve performance and minimise deformation. Based on fatigue life predictions and safety factor assessments, it is evident that these specific areas are susceptible to fatigue failure. Therefore, it is necessary to implement targeted design enhancements in order to guarantee long-term reliability.

The literature review highlights the efficacy of Finite Element Analysis (FEA) in optimising automotive suspension components. Research has demonstrated that precise modelling, meticulous material selection, and confirmation through experimental testing are essential for guaranteeing the dependability and longevity of these vital components. Constant advancements in FEA tools and techniques will continue to improve the design and performance of suspension systems, resulting in safer, more efficient, and more comfortable vehicles.

Overall, the systematic utilisation of advanced computer-aided design (CAD) modelling and finite element analysis (FEA) in the development and evaluation of the front control lower arm has shown considerable promise in enhancing the strength and functionality of automotive suspension systems. The knowledge acquired from this research establishes a strong basis for forthcoming advancements, with the goal of enhancing the durability, performance, and adherence to safety regulations of these components. This study makes a valuable contribution to the continuous progress of automotive engineering by using thorough simulation and analysis. Its aim is to guarantee that vehicles maintain their safety, reliability, and efficiency.

REFERENCE

[1]. Ranshing Aishwarya Sanjeev, Amol B. Gaikwad, and Kharad B.N. "Stress Analysis & Optimization of a Lower Control Arm of Suspension System by Using OptiStruct." International Research Journal of Engineering and Technology (IRJET), vol. 10, no. 05, May 2023, pp. 1829. e-ISSN: 2395-0056, p-ISSN: 2395-0072.

[2]. Lajqi, Shpetim & Pehan, Stanislav & Lajqi, Naser & Gjelaj, Afrim & EMIN, Jože. (2013). DESIGN OF INDEPENDENT SUSPENSION MECHANISM FOR A TERRAIN VEHICLE WITH FOUR WHEELS DRIVE AND FOUR WHEELS STEERING.

[3]. Lokesh Yadav and Imraan Siraj. "Review on Control Arm of Vehicle." Journal of Advances and Scholarly Researches in Allied Education, vol. 20, no. 3, July 2023, ISSN 2230-7540.

[4]. E. R. Biglete et al., "Topology Optimization and Strength Performance Analysis of a Lower Control Arm Sedan Suspension Part," 2020 IEEE 12th International Conference on Humanoid, Nanotechnology, Information Technology, Communication and Control, Environment, and Management (HNICEM), Manila, Philippines, 2020, pp. 1-6, doi: 10.1109/HNICEM51456.2020.9400012.

[5]. Swapnil S. Khode, Amol N. Patil, and Amol B. Gaikwad. "Design Optimization of a Lower Control Arm of Suspension System in a LCV by using Topological Approach." International Journal of Innovative Research in Science, Engineering and Technology, vol. 6, no. 6, June 2017, pp. 11657. DOI:10.15680/IJIRSET.2017.0606084.

[6]. Santosh Kullur, S.N. Kurbet, Vishwanath Hugar, And Somappa Gudageri. "Structural Analysis Of Front Suspension Lower Control Arm For Material Optimization – FEM Approach." International Journal On Recent Technologies In Mechanical And Electrical Engineering (IJRMEE), Vol. 2, No. 6, June 2015, Pp. 91-95. ISSN: 2349-7947. Available At: Http://Www.Ijrmee.Org.

[7]. Rashdan, Muhammad. (2016). Prediction Model Of Fatigue Life On Lower Suspension Arm.

[8]. Pachapuri, Mohammed Sadiq & Lingannavar, Ravi & Kelageri, Nagaraj. (2021). Design and analysis of lower control arm of suspension system. Materials Today: Proceedings. 47. 10.1016/j.matpr.2021.05.035.

[9]. Balasaheb Gadade, R.G. Todkar, ―Design and Analysis of A-type Front Lower Suspension arm in Commercial vehicle‖ International Research Journal of engineering and Technology, vol. 2, Issue, 2015.

[10]. S. Arun Kumar, V. Balaji, K. Balachandar, and D. Prem Kumar, ―Analysis and optimization of lower control arm in front suspension system,‖ Int. J. Chem. Sci., vol. 14, no. 2, pp. 1092– 1098, 2016.

[11]. P. J. Liew, U. S. Hashim, and M. N. A. Rahman, ―A New Design Optimization of Light Weight Front Lower Control Arm,‖ J. Adv. Manuf. Technol., vol. 11, no. 1, pp. 61–68, 2017.

[12]. Singh, Hardial & Bhushan, Gian. (2012). Finite Element Analysis of a Front Lower Control Arm of LCV Using Radioss Linear.

[13]. Sagar Darge et al. "Finite Element Analysis and Topography Optimization of Lower Arm of Double Wishbone Suspension Using Abacus and Optistruct" International Journal of Engineering Research and Applications .ISSN : 2248-9622, Vol. 4, Issue 7(Version 6), July 2014, pp.112-117.

[14]. Jagwinder Singh et al. "Static Structural Analysis of suspension arm using Finite Element Method" IJRET: International Journal of Research in Engineering and Technology eISSN: 2319-1163,pISSN: 2321-7308 Volume: 04 Issue: 07 July-2015.

[15]. Sarvadnya Ajinkya Thakare et al. "Design and Analysis of Modified Front Double Wishbone Suspension for a Three Wheel Hybrid Vehicle" Proceedings of the World Congress on Engineering 2015 Vol II WCE 2015, July 1 - 3, 2015, London, U.K.

[16]. Nikita Gawai et al. "Design, Modelling& Analysis of Double Wishbone Suspension System "International Journal on Mechanical Engineering and Robotics (IJMER) ISSN (Print) : 2321-5747, Volume-4, Issue-1,2016.