Journal of Advances and Scholarly Researches in Allied Education
Vol. 20, Issue No. 3, July-2023, ISSN 2230-7540

Review on Control Arm of Vehicle

Lokesh Yadav[1]*, Imraan Siraj[2]

[1] Research Scholar, IITM Group of Institutions, Murthal (Haryana)

Email: yadavlokesh22@gmail.com

[2] Associate Professor, IITM Group of Institutions, Murthal (Haryana)

Abstract - Front control lower arm are used in suspension system of vehicles. The current research reviews the existing work conducted in improving design of lower control arm. These researches are conducted using both experimental and numerical techniques. The studies explore different techniques, including finite element analysis (FEA), topology optimization, and evolutionary structural optimization (ESO), to enhance the performance, durability, and cost-effectiveness of front control lower arm. The use of computational methods, such as FEA and CAD software, enables researchers to analyze and model the behavior of these components under various loading conditions and optimize their design parameters. The studies emphasize the importance of considering factors such as material selection, weight reduction, fatigue life, and structural efficiency in the design and optimization process.

Keywords - Front control lower arm, Design Optimization

- X -

1. INTRODUCTION

The control arms of an automobile serve to link the steering rack to the wheels and the wheels to the vehicle's frame. The wheels are capable of mobility, and they are accompanied by the control arms that facilitate their movement.

Figure 1: Suspension lower arm [20]

The wheels of a vehicle can adapt to variations in the road, such as bumps, hills, and other hazards, by rising and falling accordingly. Control arms aid in maintaining proper wheel alignment with the road surface while also facilitating ease of movement.

2. LITERATURE REVIEW

Bendsoe et. al. [1] In order to generate final designs for structural elements based on boundary variations that are topologically equivalent to the initial design selection, general, stable computational schemes for this approach often require some remeshing of the finite element approximation of the analysis problem. In order to get over these two limitations, our research suggests a new strategy for optimum shape design. In this process, which is related to contemporary manufacturing practices, an anisotropic material is created by inserting a finite number of tiny holes at regular intervals into an otherwise homogenous and isotropic material. The finished product must be strong enough to withstand the specified loads and conform to all other design specifications. To determine the true material properties of an anisotropic material, the homogenization technique is used. The results of calculations are compared to those of boundary adjustments.

Gaviglio et. al. [2] This research uses the FEA technique to examine a more horizontal suspension arm. A constrained component analysis in Ansys and a PRO-E lowlife model would have allowed for the generation of this report. Using element motors and a life cycle assessment (LCA), the evaluation's primary focus will be on establishing the structural quality. This paper's investigation may show how limiting access to certain components affects the overall product development process. Since the product development cycle for a single vehicle consists of three to four stages, the first stage involves a significant amount of cost recovery. Proto-I, Alpha-ii, Gamma-ii, and Beta-ii are all necessary at these stages. The study may have provided an indication that the findings of limited component inspection with actual physical example testing would be adopted. Results from both theoretical analysis and experimental verification of models were compared in this study.

Sridharan and Balamurugan (2007) [3] examined the sheet metal Lower Control Arm (LCA) used in front suspension systems in order to perform structural analysis. In accordance with the parameters you supply, Pro-E will simulate LCA. LCA analysis is performed with the help of CAE software. Because the stresses occurring on the control arm are dynamic, a buckling analysis is crucial. At first, finite element analysis is used to calculate the shear strength of a control arm. Solid Works is used as a stimulation software for FEA. The FEA results are compared after design iterations. The elements that have an impact on the outcome of the response have been isolated. After the finite element analysis was finished, the design of experiments technique was utilised to fine-tune the results. Taguchi's experimental design was used to minimise the total number of experiments. The production cost of the lower control arm is decreased by lowering the thickness of the sheet metal and advising a suitable material. Because of this, both the product's material quality and its price are enhanced.

Saleem et al (2008) [4] The industrial sector is experiencing a surge in cutting-edge design, weight reduction, and cost-effectiveness of its products, owing to the progress made in structural optimization techniques and computer simulations. Topology optimization has become a critical stage in the design process, particularly in the aerospace and automotive industries. The present investigation involves the optimization of the vertical stabilizer of a commercial aircraft utilizing nonparametric topology within the ANSYS software. In order to incorporate the actual stresses encountered by an aircraft during flight maneuvers, the design space of the component is augmented with relevant loads and constraints. The challenges encountered in this scenario are attributed to a combination of factors, including the influence of fin gust, rudder deflection, lateral gust, and various other aerodynamic forces. Furthermore, a comprehensive methodology has been developed to address the issue of impracticable topology optimization outcomes and validate the structural efficacy. Furthermore, subsequent to generating an initial residual stress field via sequential coupled field analysis, it is possible to simulate post-machining distortions by means of the element deactivation technique. Prior to commencing the virtual machining process, the optimized finite element model is converted into a CAD model based on geometry using CATIA. Subsequently, the fabricated component of the aircraft is evaluated against the topology-guided blueprint, culminating in the last stage of the process. The implementation of topology optimization is anticipated to yield a novel and superior product design, characterized by enhanced structural efficiency and stability.

Nawar and Asady (2009) [5] This approach has the potential to address challenges in diverse engineering domains, encompassing structural analysis, thermal management, mechanics, and other related areas. Finite element analysis (FEA) is a technique employed by engineers to simulate complex engineering problems and obtain distinct solutions for each component of a system. The mathematical approximation of the mechanical and physical characteristics of individual components within a system serves as the fundamental basis for Finite Element Analysis (FEA). This study centered on the fundamental finite element analysis (FEA) of truss structures utilizing rods.

Noor and Rahman (2009) [6] conducted a study. To optimize the potential benefits of FEA analysis in the future, it is advisable to validate its accuracy and reliability by cross-checking it against experimental data. Through the analysis of strain data from the finite element model, scholars were able to identify the optimal locations for the placement of strain sensors on the object. Despite the occurrence of fatigue damage at low cycle ranges, its justification lies in the significant number of cycles. Therefore, it is imperative to obtain experimental data on components to ensure their reliability.

Rahman et al (2009) [7] The fatigue life behavior of the lower suspension limb was investigated by the author through the utilization of the strain life method. The primary objectives of this study are to devise techniques for predicting fatigue endurance, identifying the pivotal juncture, and making knowledgeable decisions regarding the material selection for the suspension arm. Aluminium alloys are utilized as suspension arm materials in various applications. The projected endurance duration ascertained through the implementation of a finite element-driven fatigue analysis methodology. The suspension limb's structural model was generated utilizing Solid Works software. The finite element model and analysis were conducted using software for finite element analysis. The fatigue life was determined utilizing the strain life method under conditions of loading with varying amplitudes. This research considers the three distinct types of variable amplitude.

Das and Jones (2011) [8] have demonstrated the optimal topology design of an aeronautical component through a modified Evolutionary Structural Optimisation (ESO) methodology. An instance of how ESO can provide an optimal design within a prescribed strength limitation is demonstrated by the challenge encountered in aircraft design, specifically the optimization of the topology of a bulkhead utilized in constructing an airframe. Research has demonstrated that topology optimization based on evolutionary structural optimization (ESO) can result in a structure that exhibits stress dispersion that is uniform throughout, thereby resulting in substantial reductions in the overall mass of the structure and enhanced utilization of materials. This study investigates the efficacy of the ESO method in addressing complex topology design problems encountered in manufacturing facilities.

Patil et al., (2013) [9] have presented an analysis of the left lower wishbone limb of an independent suspension system using experimental and finite element methods. The deflection and strain of steel and composite lower wishbone arms exhibit significant fluctuations under static load conditions. The static specifications of carbon fibre suspension control arms are equivalent to those of their steel counterparts. When subjected to identical stress conditions, the composite lower wishbone arm exhibits greater deflection compared to its steel counterpart. On average, the recently introduced suspension arms exhibit a weight reduction of approximately 27% in comparison to their steel counterparts. The composite material lower wishbone arms exhibit a higher natural frequency as compared to their steel counterparts.

Sushil et al (2015) [10] explicate the process of designing, modelling, and analysing a vehicle's front suspension lower arm to evaluate its tension condition and determine the optimal materials. This study primarily focused on the critical locations and strain distributions of the component. This research is centred on the examination of the lower limb of the front suspension through finite element analysis. The study aims to optimise stress loading and analyse deformation.

Aishwarya and Ramanamurthy (2015) [11] The connecting rod serves as the linkage between the piston and the crankshaft. The primary function of this mechanism is to convert the back-and-forth movement of the piston into the circular motion of the crankshaft. This article focuses on the connecting rod of a Hero Honda Splendour Plus motorcycle, a real-world subject that necessitates modelling, analysis, and optimisation. The present scenario suggests that the implementation of structural components is a preferable option over the connecting rod. Moreover, materials such as stainless steel, aluminium, and C70 steel can be considered as viable substitutes. The design of the connecting rod has been executed in CATIA software for both the current solid truss configuration and the enhanced truss design. The implementation of boundary conditions occurs subsequent to the preprocessing of models in ANSYS 14. The ANSYS 14.0 programme was utilised to determine the optimal combination of parameters, including Von Misses stress and strain, deformation, factor of safety, fatigue and life cycle calculation, and biaxiality indicator, for pistons in two-wheeled vehicles.

Zhu et al (2015) [12] the utilisation of stable level set techniques is imperative for the effective management of level set function gradients at the design interface, particularly in the context of structural topology and form optimisation. To guarantee that the level set function satisfies the properties of a signed distance function, it is common practise to employ frequent initialization techniques, commonly referred to as "re-initialization." Nevertheless, the gradual resolution of a growing set of partial differential equations during each iteration results in a deceleration of these re-initialization methods. Re-initialization of any entity carries the potential of introducing unforeseen complications, such as the displacement of the zero-level set. The present study proposes the utilization of the level set method along with a distance-suppression scheme to enhance the optimization of the structure's topology and shape. The preservation of the level set function in close proximity to a signed distance function at the structural boundaries, along with its constancy at points far from the structural limits, is achieved through the incorporation of an energy functional into the level set equation. As a result, the current methodology not only eliminates the requirement for re-initialization but also simplifies the procedure of creating the initial level set function.

Gadade and Todkar (2015) [13] provide a comprehensive description and analysis of the A-type front lower suspension arm utilised in commercial vehicles. The primary impetus for this investigation was to determine the anticipated duration of the constituent when subjected to stationary loads. The A-type lower suspension limb was fabricated utilising computer-aided design (CAD) software. Consistent with the concept, a prototype constructed from AISI 1040 material was fabricated. The present analysis employed a hyper mesh to import the relevant model. The model's weakest aspect was identified through the meshing of the load application to the hub bush. However, it is imperative to conduct physical testing to verify the findings.

Arun et al (2016) [14] conducted a structural integrity analysis of a metal sheet component known as a lower control arm, which is utilized in front suspension systems. The lower control limb facilitates the vertical movement of the wheel. It is customary to affix steel brackets to rubber bushings on the chassis. The present methodology utilized for the regulation of the lower limb of the governing leg.

Liew et al (2017) [15] conducted an investigation into the optimization of an aluminium casting for the lower front control arm. The lower control arm was devised utilizing CATIA. Furthermore, the software package Hyperworks was employed to evaluate the structural soundness and enhance the mass efficiency. The initial steel-fabricated constituent exhibited a weight that was 20% greater than the enhanced iteration. The findings indicate a reduction of 25% in weight and a fatigue life span of approximately 396,000 cycles. The revised design of the lower arm for the front suspension is deemed appropriate for a passenger vehicle in the C-segment and satisfies the criteria for fatigue life cycle.

Khode et al (2017) [16] the suspension system plays a critical role in determining a vehicle's performance, safety, and pollution level, thereby rendering it the most pivotal component of the vehicle. The unsprung mass in a suspension system refers to the mass of those constituents that are not sustained by the suspension, but are rather affixed to the vehicle directly. Excessive unsprung mass exacerbates problems related to handling, comfort, and noise. The components of wheels, bearings, hubs, springs, shocks, and the Lower Control Arm collectively constitute the unsprung weight of a vehicle. The lower control arm, a wishbone-shaped metallic component, is responsible for attaching the wheel to the car's chassis. The automotive sector has extensively employed diverse optimization techniques across various load conditions to minimize weight and enhance performance. The objective of this study was to ascertain the optimal parameters for the lower control arm of a Macpherson suspension system when subjected to static loads. A pre-existing configuration of the lower control arm of a light commercial vehicle has been selected for additional examination.

Marzbanrad and Hoseinpour (2017) [17] conducted a study aimed at reducing the weight of a MacPherson control arm through the optimization of its topology and shape. The contemporary automotive sector is constantly seeking cost-effective and lightweight constituents to meet the increasing need for more fuel-efficient and ecologically sustainable automobiles. Consequently, a reduction in material waste during the production of automobile components may potentially lead to a decrease in the overall weight of the vehicle. In consideration of the dynamic pressures that automotive components are subjected to, the incorporation of fatigue criteria is imperative in the design process to mitigate potential fatigue damage. Initially, a set of uneven roads based on power spectral density is devised to replicate a demanding loading scenario for the control arm. The identification of crucial loading conditions is conducted through a comprehensive multi-body dynamics analysis of the complete vehicle model. The HyperMesh software employs topology optimization techniques to meet the endurance life requirement, resulting in a significant reduction of mass up to 50%. Subsequently, the subsequent phase entails generating a computer-aided design (CAD) model utilizing CATIA software and refining the design iteratively to attain the targeted dimensions and weight.

Kale et al (2018) [18] the control arm of an automobile serves the purpose of connecting the sprung and unsprung masses of the said vehicle. The lower control arm is subject to various forces. If exposed to such force, the control arm could potentially undergo deformation. The lower control arm has experienced a malfunction. One of the primary considerations is the necessity to reduce the weight of automotive components. The objective of this article is to employ finite element analysis for the purpose of investigating and enhancing the lower extremity. The CAD software is utilized to construct the model of the limb responsible for regulating the lower body. The study entails a stationary assessment of the functional limb. The topography of the lower limb has been optimized to minimize its mass.

Hafizi et al (2018) [19] conducted a study on the front lower control arm (FLCA), focusing on the optimization of its design and production technique for the aluminium cast. The updated concept for the front lower control arm has been put into practice in this instance. The lower control extremity utilized in this study was generated using CATIA. Hyperworks is utilized for the purpose of analyzing the structural soundness of the product, while simultaneously minimizing superfluous mass. The proposed strategy entails a reduction of 20% in the aggregate weight of the front lower control arm. The aforementioned limb is composed of a metallic material. The statistics obtained indicate a significant reduction in weight of approximately 25%, accompanied by a fatigue life cycle of 396,000 cycles.

3. CONCLUSION

The studies presented focused on various aspects of structural analysis and optimization of different components used in automotive and aerospace industries. The studies explore techniques such as finite element analysis (FEA), topology optimization, evolutionary structural optimization (ESO), and the use of advanced software tools like ANSYS and CATIA to improve the design, performance, and efficiency of these components. The goal is to achieve lighter weight, cost-effectiveness, enhanced structural integrity, and improved material utilization while ensuring that the components meet the required specifications and performance standards. The findings from these studies contribute to the advancement of engineering practices and offer valuable insights into the design and optimization of critical components in various industries.

REFERENCES

[1] M. P. Bendsoe and O. Sigmund, Topology optimization: theory, methods, and applications, vol. 2nd Editio, no. 724. 2003.

[2] G. Chiandussi, I. Gaviglio, and A. Ibba, “Topology optimisation of an automotive component without final volume constraint specification,” Adv. Eng. Softw., vol. 35, no. 10–11, pp. 609–617, 2004.

[3] M. Sridharan and S. Balamurugan, “Design and Analysis of Lower Control ARM,” Int. J. Innov. Res. Sci. Eng. Technol. (An ISO Certif. Organ., vol. 3297, no. 4, pp. 6510–6528, 2007.

[4] W. Saleem, F. Yuqing, and W. Yunqiao, “Application of topology optimization and manufacturing simulations - A new trend in design of Aircraft components,” Imecs 2008 Int. Multiconference Eng. Comput. Sci. Vols I Ii, vol. II, pp. 1791–1796, 2008.

[5] Nawar A. Al-Asady, "Automobile body reinforcement by finite element optimization", Department of mechanical Engineering. UKM, Malaysia, 2009.

[6] IiM. M. Noor and M. M. Rahman, Fatigue Life Prediction of Lower Suspension Arm Using strain-life approach, Euro Journals Publishing, Inc., 2009.

[7] M.M. Rahman et al., "Linear Static Response of Suspension Arm Based on Artificial Neural Network Technique", Advanced Materials Research, vol. 213, pp. 419-426, 2011.

[8] R. Das and R. Jones, "Topology optimisation of a bulkhead component used in aircrafts using an evolutionary algorithm," Procedia Eng., vol. 10, pp. 2867–2872, 2011.

[9] A. M. Patil1, A.S. Todkar, Prof. R. S. Mithari, "Experimental & Finite Element Analysis of Left Side Lower Wishbone Arm of Independent Suspension System "IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE), vol. 7, Issue 2, 2013.

[10] Sushil kumar P. Taksande, A.V. Vanalkar, "Methodology for Failure Analysis of Car Front Suspension Lower Arm- a Review" International journal for Scientific Research & Development Volume-3 Issue- 5, 2015.

[11] S. Aishwarya and E. V. V. Ramanamurthy, "Design and optimization of connecting rod for 4 - stroke petrol engine by using finite element analysis," ARPN J. Eng. Appl. Sci., vol. 10, no. 11, pp. 4994–4999, 2015

[12] B. Zhu, X. Zhang, and S. Fatikow, "Structural topology and shape optimization using a level set method with distance-suppression scheme," Comput. Methods Appl. Mech. Eng., vol. 283, pp. 1214–1239, 2015.

[13] 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.

[14] 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.

[15] 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.

[16] S. S. Khode, P. A. N. Patil, and P. A. B. Gaikwad, "Design Optimization of a Lower Control Arm of Suspension System in an LCV by using," Int. J. Innov. Res. Sci. Eng. Technol., vol. 6, no. 6, pp. 11657– 11665, 2017.

[17] J. Marzbanrad and A. Hoseinpour, "Structural optimization of MacPherson control arm under fatigue loading," Teh. Vjesn. - Tech. Gaz., vol. 24, no. 3, pp. 917–924, 2017.

[18] A. R. Kale, P. A. Tadamale, and N. D. Patil, "Analysis and Optimization of Upper Control Arm of," Evol. Mech. Eng., vol. 1, no. 1, pp. 1–6, 2018.

[19] M. Hafizi, A. Rahman, M. S. Salleh, M. Suffian, A. Razak, and M. R. Mohamad, "Design and Optimization of Front Lower Control Arm (FLCA) for C - Segment Passenger Car," Int. J. Eng. Technol., vol. 7, no. 3, pp. 71–75, 2018.

[20] Prashant Gunjan , Amit Sarda, "Design and Analysis of Front Lower Control Arm by Using Topology Optimization" IJARIIE-ISSN(O)-2395-4396

Corresponding Author

Lokesh Yadav*

Research Scholar, IITM Group of Institutions, Murthal (Haryana)