A Review of Analysis and Optimization of FLCA

Aman K Patel[1], Pankaj Sidar[2] and S. C. Gajbhiye[3]

[1]*M. Tech. Scholar,* [23]*Assistant Professor*

[123]*Department of Mechanical Engineering, Vishwavidyalaya Engineering College, Lakhanpur (C. G.), INDIA*

ABSTRACT

Chassis parts are a critical part of a vehicle, leaving no room for error in the design and quality the present process relates to a computer-aided structure analysis and design graphic performance tool and method, and more particularly, to a computer-aided structure analysis of lower control arm (LCA) and which is analyzed and designed, thereby meet the customer requirements of LCA. Lower control is a kind of independent suspension used in vehicles. During actual working conditions, the maximum load is transferred from the upper arm to the lower arm, which creates the possibility of failure in the hand. Therefore, it is important to focus on stress control of lower control arm to improve and change existing design. This present study is to optimize the lower control arm by topological optimization with varying different material, and reducing the von-mises stress, deformation, shear stress as well as mass to reduce the batch production cost and to increase the strength of LCA.

Keywords: *FLCA, Finite Element Analysis, Optimization.*

1. INTRODUCTION

Suspension arm is one of the principle segments in the suspension frameworks. It can be seen in different sorts of the suspensions like wishbone or twofold wishbone suspensions. The vast majority of the circumstances it is called as A-type control arm. It joins the wheel center to the vehicle outline taking into account a full scope of movement while keeping up appropriate suspension arrangement. Uneven tire wear, suspension clamour or misalignment, guiding wheel shimmy or vibrations are the primary driver of the disappointment of the lower suspension arm. The majority of the cases the disappointments are cataclysmic in nature. In this way, the auxiliary uprightness of the suspension arm is urgent from configuration perspective both in static and dynamic conditions. As the Finite Element Method (FEM) gives better representation of this sort of the disappointments so FEM examination of the pressure disseminations around regular disappointment starts locales is fundamental. Thus, this work it is proposed to do the auxiliary investigation of lower suspension arm of light commercial vehicle utilizing FEM. The suspension arm gets more consideration by numerous looks into like investigation dynamic examinations of the engine vehicle suspension framework utilizing the point-joint organizer's plan. The mechanical framework is supplanted by a proportionate compelled arrangement of particles and after that the laws of molecule progression are utilized to determine the conditions of movement. Displaying and reproduction are vital when managing complex designing frameworks. The best method to enhance item quality and unwavering quality is to incorporate them in the plan and assembling process.

Fig 1. Front Lower Control Arm

2. LITERATURE REVIEW

In this Chapter focuses on the literatures on lower control arm and related used in suspension system of vehicles.

Bendsoe and Kikuchi (1988) optimal shape design of structural elements based on boundary variations results in final designs that are topologically equivalent to the initial choice of design, and general, stable computational schemes for this approach often require some kind of remeshing of the finite element approximation of the analysis problem. This paper presents a methodology for optimal shape design where both these drawbacks can be avoided. The method is related to modern production techniques and consists of computing the optimal distribution in space of an anisotropic material that is constructed by introducing an infimum of periodically distributed small holes in a given homogeneous, isotropic material, with the requirement that the resulting structure can carry the given loads as well as satisfy other design requirements. The computation of effective material properties for the anisotropic material is carried out using the method of homogenization. Computational results are presented and compared with results obtained by boundary variations.

Kothawale and Kharde (2003) This paper depicts examination of more level suspension arm utilizing f. E. An. Approach. This paper might have been get ready lowlife model utilizing PRO-E product & limited component Investigation utilizing Ansys product. The paramount hugeness of examination will be with check those structural quality from claiming LCA utilizing element drives. The point about this paper dissection may be with show the how limited component dissection is making a difference on complete item improvement cycle. A result it setting off with recoveries great deal of cost, as each vehicle Hosting for the most part 3-4 phases done finish item improvement cycle, phases need aid Proto-I, Alpha-II, Gamma-III & Beta-IV. This paper might have been hint at those acceptance for limited component examination outcomes with real physical example testing. In this examine they closed that a what amount of those outcomes Toward utilizing Investigation product and also physical trying of model would comparable or not.

Chiiandussi et al (2004) explored the results obtained by using a topology optimisation code to solve a three-dimensional problem concerning a real automotive component. The implemented optimisation method is based on the maximization of the total potential energy with a volume constraint by optimality criteria. The volume of the optimal solution depends on the imposed static (displacement, stress, stiffness) and dynamic (natural frequency) constraints and has not to be specified a priori. The optimisation process converges toward a quite well-defined structure made of the base material with a very little percentage of elements characterized by intermediate material properties.

Sridharan and Balamurugan (2007) studied of model and to perform structural analysis of a LOWER CONTROL ARM (LCA) used in the front suspension system, which is a sheet metal component. LCA is modeled in Pro-E software for the given specification. To analyze the LCA, CAE software is used. The load acting on the control arm are dynamic in nature, buckling load analysis is essential. First finite element analysis is performed to calculate the buckling strength, of a control arm. The FEA is carried out using Solid works stimulation package. The design modification has been done and FEA results are compared. The influencing parameters which are affecting the response are identified. After getting the final result of finite element analysis optimization has been done using design of experiment method. Taguchi's design of experiments has been used to optimize the number of experiments. By reducing thickness of the sheet metal and by suggesting the suitable material the production cost of lower control arm is reduced. This leads to cost saving and improved material quality of the product.

Saleem et al (2008) Structural optimization tools and computer simulations have gained the paramount importance in industrial applications as a result of innovative designs, reduced weight and cost-effective products. Especially, in aircraft and automobile industries, topology optimization has become an integral part of the product design process. In this paper nonparametric topology optimization has been applied on a commercial aircraft vertical stabilizer component using ANSYS software. Suitable loads and constraints are applied on the initial design space of the component to accommodate for fin gust, rudder deflection, lateral gust, and other loads experienced by an aircraft during actual flight maneuvering. An integrated approach has also been developed to verify the structural performance and to overcome the problem of no manufacturable topology optimization results. Post machining distortions are also simulated by using element deactivation technique first by developing an initial residual stress field through Sequential Coupled Field analysis. CATIA is used to convert the optimized FE model into geometry-based CAD model and then virtual machining is done. At the end topology assisted design model is compared with the actual part that is being manufactured for the aircraft. It is inferred that topology optimization results in a better and innovative product design with enhanced structural performance and stability.

Nawar and Asady (2009) Finite Elements Analysis (FEA) is the mathematical representation of complex engineering problems to obtain a unique solution for each segment of the system; it can be used to obtain structural, heat transfer, static and various other engineering systems solutions. FEA is based on approximating the mechanical and physical behaviour of each tiny segment of the system by a mathematical representation. This paper dealt about simple FEA for a truss (rod).

Noor and Rahman (2009) FEA analysis must be compared with experimental data to validate it to ensure its accuracy in order to maximize their advantages in the future, Strain analysis data were reasonably correct by the FE model and helped researchers to identify the exact position where the strain gauges must be fixed, Component (part) data should be collected experimentally, even though fatigue damage occurred at low cycles range, it was justified by the great number of the cycles.

Rahman et al (2009) worked the fatigue life behaviour of lower suspension arm using strain life approach. The main objectives of this study are to predict the fatigue life and identify the critical location and to select the suitable materials for the suspension arm. Aluminum alloys are selected as some suspension arm materials. The fatigue life predicted utilizing the finite element-based fatigue analysis code. The structural model of the suspension arm was utilizing the Solid works. The finite element model and analysis were performed utilizing the finite element analysis code. In addition, the fatigue life was predicted using the strain life approach subjected to variable amplitude loading. The three types of variable amplitude are considered in this study.

Das and Jones (2011) demonstrates the application of a modified Evolutionary Structural Optimisation (ESO) algorithm for optimal design of topology for an aerospace component. The capabilities of ESO for producing an optimal design against a specified strength constraint are illustrated using an aerospace design problem of optimisation of the topology of a bulkhead used in an aircraft structure. It has been shown that topology optimisation using ESO can result in considerable reduction in the weight of a structure and an optimum material utilization by generating a uniformly stressed structure. The paper evaluates and establishes the ESO method as a practical tool for optimum topology design problems for complex industrial structures.

Patil et al., (2013), this paper describes Experimental & Finite Element Analysis of Left Side Lower Wishbone Arm of Independent Suspension System. Under the static load conditions deflection and stresses of steel lower wishbone arm and composite lower wishbone arm are found with the great difference. Carbon fibre suspension control arms that meet the same static requirements of the steel ones they replace. Deflection of Composite lower wishbone arm is high as compared to steel lower wishbone arm with the same loading condition. The redesigned suspension arms achieve an average weight saving of 27% with respect to the baseline steel arms. The natural frequency of composite material lower wishbone arm is higher than steel wishbone arm.

Sunil et al (2015) in his paper presents design, modeling and analysis of car front suspension lower arm to study the stress condition and to select the suitable materials for the front suspension lower arm. The main objectives of this study to determine critical locations and strain distributions of the component. The paper aims to complete Finite Element Analysis of the front suspension lower arm which consist the stress optimization loadings and analysis for deformation.

Aishwarya and Ramanamurthy (2015) The connecting rod is the mediating member between the piston and the Crankshaft. Its main function is to convert the reciprocating motion of the piston into rotary motion of the crank. This paper describes about a real time problem of using Cast Iron connecting rod in Hero Honda Splendor + motorbike it's modelling and analysis and optimization of connecting rod. Here, the connecting rod is replaced by various materials like stainless steel, aluminium, C70 steel and also a design change by inducing truss member is suggested. The connecting rod is modelled using CATIA software for both existing solid and modified truss designs. Boundary conditions are applied to the models after finishing the pre - processing work in ANSYS 14.0 software. The best combination of parameters like Von misses stress and strain, Deformation, Factor of safety, fatigue and life cycle calculation, bi-axiality indication for two wheeler piston were done in ANSYS 14.0 software. This project also tends to optimize the design by calculating weight and stiffness for various materialistic designs by using the output values of mass and volume of the connecting rod which will also be obtained from the software. This paper will conclude whether the modified design is safe along above selected materials. And will be presenting the best design for future reference.

Zhu et al (2015) in level set methods for structural topology and shape optimization, the level set function gradients at the design interface need to be controlled in order to ensure stability of the optimization process. One popular way to do this is to enforce the level set function to be a signed distance function by periodically using initialization schemes, which is commonly known as re-initialization. However, such re-initialization schemes are time-consuming, as additional partial differential equations need to be solved in every iteration step. Furthermore, the use of re-initialization brings some undesirable problems; for example, it may move the zero-level set away from the expected position. This paper presents a level set method with distance-suppression scheme for structural topology and shape optimization. An energy functional is introduced into the level set equation to maintain the level set function to close to a signed distance function near the structural boundaries, meanwhile forcing the level set function to be a constant at locations far away from the structural boundaries. As a result, the present method not only can avoid the need for re-initialization but also can simplify the setting of the initial level set function. The validity of the proposed method is tested on the mean compliance minimization problem and the compliant mechanisms synthesis problem. Different aspects of the proposed method are demonstrated on a number of benchmarks from the literature of structural optimization.

Gadade and Todkar (2015) this paper describes Design, analysis of A-type front lower suspension arm in commercial vehicle. The main objective of this study was to calculate working life of the component under static loading. The A-type lower suspension arm was developed by using CAD software. Actual model was manufacture as per design by using AISI 1040 material. This paper result was this model imported in the hyper mesh. After meshing apply load on hub bush, they found the weaker section in the model but it is required validating the FEA results with actual experimental test.

Arun et al (2016) The main objective work was to model and to perform structural analysis of a lower control arm used in the front suspension system, which is a sheet metal component. Lower control arm allows the up and down motion of the wheel. It is usually a steel bracket that pivots on rubber bushings mounted to the chassis. The existing method in lower control arm.

Liew et al (2017) design optimization of aluminium cast for the front lower control arm was investigated. CATIA software was utilized to design the lower control arm. Hyperworks software was also used to analyse the structural strength and optimize the parts weight. The target of the new design was a 20% weight reduction from the existing part fabricated using steel material. The results showed a significant reduction of the overall weight as high as 25% with a fatigue life cycle approximately 396,000 cycles. Hence, the new design of front lower arm has fulfilled the criteria of fatigue life cycle and is suitable to be used in a C-segment passenger car.

Khode et al (2017) the most important component in vehicle is a suspension system, which directly affects the safety, performance and noise level. The unsprung mass is the mass of the suspension components which is directly connected to them, rather than supported by the suspension. High unsprung weight exacerbates issues like wheel control, ride quality and noise. Unsprung weight includes the mass of components such as the wheel axles, wheel bearings, wheel hubs, springs, shock absorbers, and Lower Control Arm. The lower control arm is a wishbone-shaped metal strut that attaches the wheel to the vehicle's frame. Different optimization techniques under various load conditions have been widely used in automobile sector for lightweight and functioning enhancement. This study deals with Finite Element Analysis of the Lower control arm of Macpherson suspension system and its optimization under static loading condition. The existing design of lower control arm from one of the light commercial vehicles is selected for the study.

Marzbanrad and Hoseinpour (2017) studied the topology and shape optimization of a MacPherson control arm has been accomplished to achieve lighter weight. Present automotive market demands low cost and light weight component to meet the need of fuel efficient and cost-effective vehicle. This in turn gives the rise to more effective use of materials for automotive parts which can reduce the mass of vehicle. Since automotive components are under dynamic loads which cause fatigue damage, considering fatigue criteria seems to be essential in designing automotive components. At first, in order to create severe loading condition for control arm some rough roads are generated through power spectral density. Then, the most critical loading conditions are obtained through multi body dynamics analysis of a full vehicle model. Then, the topology optimization is performed based on fatigue life criterion using HyperMesh software, which resulted in 50 % mass reduction. In the next step a CAD model is created using CATIA software and shape optimization is performed to achieve accurate dimensions with less mass.

Kale et al (2018) work in control arm in automotive vehicle acts as a linkage between sprung and unsprung mass of vehicle. Lower control arm is subjected to various loads. Due to this type of loading, there are chances of bending of control arm. Hence failure of lower control arm occurs. Weight reduction of vehicle component is one of the main concerns. The aim of the paper is to analyze and optimize the lower arm using finite element analysis. The model of lower control arm is done in CAD software. Static analysis of lower control arm is carried out in analysis software. Topology optimization of lower arm is performed for weight reduction. After topology optimization static analysis of optimized lower control arm is carried out. Results obtained from analysis where studied to check whether the design is within the yield strength. Cost analysis of existing and optimized lower control arm has been carried out to know the cost effectiveness. The purpose of automobile suspension system is passenger comfort and vehicle control. The lower control is one of the important parts of suspension system.

Hafizi et al (2018) employed, the design optimization and fabrication process of aluminium cast for front lower control arm (FLCA) were investi- gated. In this work, the new design concept of front lower control arm is employed. CATIA software was employed in this work to design the concept of the lower control arm. After that, Hyperworks software is used to analyze the structural strength and opti- mized the weight of the part. The target of the new design is 20% reduction of the overall weight of the front lower control arm which fabricated using steel material. The obtained results show a significant reduction of the overall weight as high as 25% with a fatigue life cycle 396,000 cycles. This finding proved that the new design of front lower arm has fulfilled the criteria of fatigue life cycle and suitable to be used in a C-segment passenger car.

3. CONCLUSION

This present study is to optimize the lower control arm by topological optimization with varying different material, and reducing the von-mises stress, deformation, shear stress as well as mass to reduce the batch production cost and to increase the strength of LCA. All parts manufactured by rare parts will meet or exceed original equipment manufacturers specifications. Some of the most critical factors in LCA is to choose right materials, and ensure greater strength at lower cost of production. In this investigation is to model and to perform the structural analysis of LCA of automotive front Suspension System by the process of CAE to determine the von-mises stress, deformation and shear stress as well as natural frequency.

REFERENCES

[1] Martin Philip Bendsoe and Noboru Kikuchi, “Generating optimal topologies in structural design using a homogenization method,” *Comput. Methods Appl. Mech. Eng.*, vol. 71, pp. 197–224, 1988.

[2] Dattatray Kothawale, Dr. Y.R. Kharde “Analysis of Lower Control Arm in Front Suspension System Using F.E.A. Approach” International Journal of Engineering Research and Development Volume 5 Issue, 2003.

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

[4] 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.

[5] 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.

[6] 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.

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

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

[9] 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.

[10] 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.

[11] 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.

[12] 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.

[13] 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.

[14] 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.

[15] 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.

[16] 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.

[17] 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.

[18] 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.

[19] 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.

[20] 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.

[21] 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.