A Project Stage-II Report on

Weight Reduction by Static Structural Analysis of Automotive Lower Control Arm

By

Mr. Aniruddha Jadhav Exam. Seat No – B190360969

Mr. Nilay Pawale Exam. Seat No – B190361088

Mr. Atharva Kalokhe Exam. Seat No – B190360992

Mr. Aditya Bangale Exam. Seat No – B190360828

Guide

Prof. R.N. Khachane

तमसो मा ज्योतिर्गमय

Sinhgad Institutes

Department of Mechanical Engineering

Sinhgad Technical Education Society's

Smt. Kashibai Navale College of Engineering

[2022-23]

Sinhgad Technical Education Society's

Smt. Kashibai Navale College of Engineering

SINHGAD TECHNICAL EDUCATION SOCIETY-PUNE

तमसो मा ज्योतिर्गमय

Sinhgad Institutes

CERTIFICATE

This is to certify that *Mr*. Aniruddha Jadhav Exam. Seat No – B190360969

Mr. Nilay Pawale Exam. Seat No – B190361088

Mr. Atharva Kalokhe Exam. Seat No – B190360992

Mr. Aditya Bangale Exam. Seat No – B190360828

have successfully completed the Project Stage – II entitled "*Weight Reduction by Static Structural Analysis of Automotive Lower Control Arm*" under my supervision, in the partial fulfillment of *Bachelor of Engineering - Mechanical Engineering* of Savitribai Phule Pune *University.*

Date: -

Place: - Pune

Seal

Prof. R.N. Khachane
Guide

Prof…………….
External Examiner

Prof. T. S. Sargar
Head of Department

Dr. A.V. Deshpande
Principal

ACKNOWLEDGEMENT

I take this opportunity to thank all those who have contributed in successful completion of this Project Stage -II work. I would like to express my sincere thanks to my guide **Prof. R.N. Khachane** who have encouraged me to work on this topic and provided valuable guidance wherever required. I also extend my gratitude to **Prof. T. S. Sargar (H.O.D Mechanical Department)** who has provided facilities to explore the subject with more enthusiasm.

I express my immense pleasure and thankfulness to all the teachers and staff of the **Department of Mechanical Engineering of Smt. Kashibai Navale College of Engineering** for their co-operation and support.

Mr. Aniruddha Jadhav Sign……………….

Mr. Nilay Pawale Sign……………….

Mr. Atharva Kalokhe Sign……………….

Mr. Aditya Bangale Sign……………….

Abstract

The lower control arm is a unique type of independent suspension used in automobile vehicles. During the actual working condition, the maximum load is transferred from upper arm to the A arm which creates possibility of failure in the arm. Hence it is essential to focus on the stress analysis of A suspension arm to improve and modify the existing design. A suspension arm is an important part used in a suspension system of a car. suspension arm performance a major role in managing the motion of the wheel during bump, turning, and breaking. This research proposed the design optimization and static analysis of steel cast for the front A suspension arm was investigated. CATIA software was utilized to design the A suspension arm. ANSYS 19 software was also used for analyze 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. Testing and validation of new design using strain gauge analysis.

TABLE OF CONTENTS

LIST OF TABLES AND FIGURES

List of Tables

List of Figures

CHAPTER 1: INTRODUCTION

1.1 - Introduction

The control arm is a unique type of independent suspension used in automobile vehicles. During the actual working condition, the maximum load is transferred from upper arm to the A arm which creates possibility of failure in the arm. Hence it is essential to focus on the stress analysis of A control arm to improve and modify the existing design. A control arm is an important part used in a suspension system of a car. control arm performance a major role in managing the motion of the wheel during bump, turning, and breaking. This paper proposed the design optimization and model analysis of steel cast for the front A control arm was investigated. CATIA software was utilized to design the A control arm. ANSYS 19 software was also used to analyze 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. Testing and validation of new design using FFT analysis. Model analysis carried out to find out natural frequencies of A control arm and validation are done with the help of FFT analyzer and Impact Hammer Test.

Figure 1.1 – Automobile Lower Control Arm

1.2 - Background of the Project

The lower control arm gets more attention by many researches like study dynamic analyses of the motor vehicle suspension system using the point-joint coordinate's formulation. The mechanical system is replaced by an equivalent constrained system of particles and then the laws of particle dynamics are used to derive the equations of motion. Modelling and simulation are indispensable when dealing with complex engineering systems. It makes it possible to do an essential assessment before systems are developed. It can provide support in all stages of a project from conceptual design, through commissioning and operation. The most effective was to improve product quality and reliability is to integrate them in the design and manufacturing process.

The challenge for automotive manufacturer to produce the lightweight components but at the same time need to maintain the performance of the components. To produce the lightweight components, the main criteria to overlook are the advance material and the manufacturing technology.

Weight reduction enables the manufacturer to develop the same vehicle. If all the components in vehicle can contributed for about 10% to 20% of weight reduction, it is estimated that 20% of vehicle weight reduction results in 8–10% of fuel economy improvement. There several ways to achieve the lightweight components in vehicle. Changing a new lightweight material such as composites and aluminum alloy can give a good weight reduction to vehicle and improve the fuel economy [1].

Nowadays, several automotive manufacturers already change from steel alloy and cast iron to alternative lightweight material such as composite and aluminum alloy. The market pattern for material shows gradual decrease for steel and cast-iron usage in automotive industries. [1].

Ford as one of the automotive manufacturers widely used the aluminum alloy in body in white structure and closure such as panel door outer, roof, panel hood outer, trunk lid, tailgate, etc. For examples, Ford F150 model 2015 mostly used Aluminum alloy for the body in white structures and at the closure panels. Meanwhile, the Cadillac ATS and CT6 used mixing material such as using aluminum casting, high strength steels and sheet metals on their BIW and closures panels [1].

1.3 - Lower Control Arm

In automotive suspension, a control arm, also known as an A-arm, is a hinged suspension link between the chassis and the suspension upright or hub that carries the wheel. In simple terms, it governs a wheel's vertical travel, allowing it to move up or down when driving over bumps, into potholes, or otherwise reacting to the irregularities of a road surface. Most control arms form the lower link of a suspension. Control arms play a crucial role in the suspension system of a vehicle. They help to keep the wheels aligned and maintain proper tire contact with the road, which is essential for safety and stability.

Dependent suspension system:

A dependent suspension normally has a beam (a simple 'cart' axle) or (driven) live axle that holds wheels parallel to each other and perpendicular to the axle. When the camber of one-wheel changes, the camber of the opposite wheel changes in the same way (by convention on one side this is a positive change in camber and on the other side this a negative change). De Dion suspensions are also in this category as they rigidly connect the wheels together. Example: Hotchkiss suspension and trailing arm suspension comes under this Category.

Independent suspension system:

An independent suspension allows wheels to rise and fall on their own without affecting the opposite wheel. In this case, the wheels are connected through universal joints with a swing axle. Suspensions with other devices, such as sway bars that link the wheels in some ways are still classed as independent. Example: The two important types of independent systems are Macpherson strut and Double wishbone system.

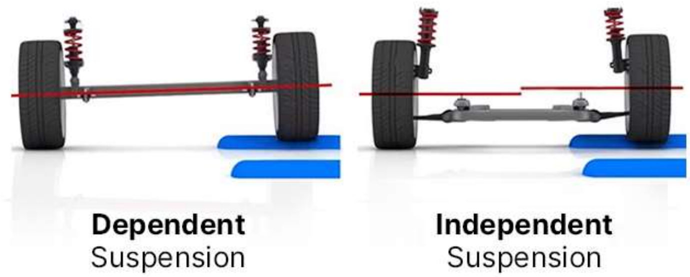


Figure 1.2 – Dependent and Independent Suspension

Lower Control Arm:

The lower control arm is the most vital component in a suspension system. There are two control arms, lower control arm and upper control arm. 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 other end supports the lower ball joint. Significant number of loads are transmitted through the control arm while it serves to maintain the contact between the wheel and the road and thus providing the precise control of the vehicle. There are many types of control arms are available. The selection of the arm is mainly based on the type of suspension system.

In automotive suspension, a control arm, also known as an A-arm, is a hinged suspension link between the chassis and the suspension upright or hub that carries the wheel. In simple terms, it governs a wheel's vertical travel, allowing it to move up or down when driving over bumps, into potholes, or otherwise reacting to the irregularities of a road surface. Most control arms form the lower link of a suspension. [1].

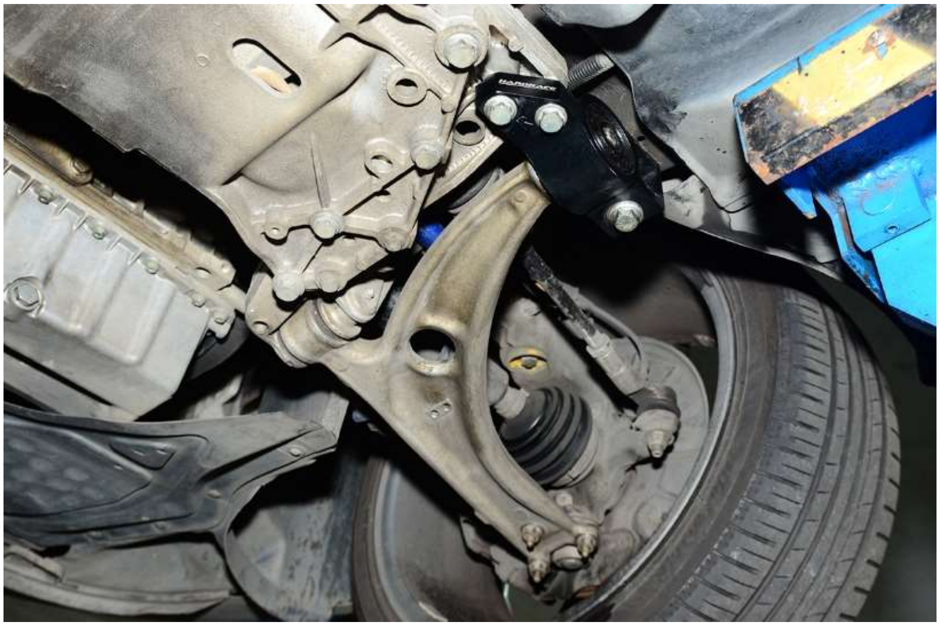


Figure 1.3 – Automobile Lower Control Arm Location

The control arm made with hardened steel and compound with superior quality bushing. A good control arm can regulate the movement of wheels quickly each time you turn or corner.

1.4 – Significance of Project

The suspension system caries the vehicle body and transmit all forces between the body and the road without transmitting to the driver and passengers. The suspension system of a car is used to support its weight during varying road conditions. The suspension system is made of several parts and components. These include the front and rear. The suspension arm gets more attention by many researches like study dynamic analyses of the motor vehicle suspension system using the point-joint coordinate's formulation. Modeling and simulation are indispensable when dealing with complex engineering systems. It can provide support in all stages of a project from conceptual design, through commissioning and operation. The most effective way to improve product quality and reliability is to integrate them in the design and manufacturing process.

CHAPTER 2: LITERATURE SURVEY

1) **M. Sadiq A. Pachapuri, Ravi G. Lingannavar, Nagaraj K. Kelageri, Kritesh K. Phadate, "Design and analysis of lower control arm of suspension system" Materials Today: Proceedings,47(2), May 2021 -**

Vehicles ranging from bullock carts to modern vehicles use suspension system such as, leaf springs, pneumatic and hydraulic systems. The main aim of using a suspension system is to prevent the shocks reaching the body of the vehicle from the rough roads and ensure that the vehicle's wheels remain in contact with the road. There are two types of forces acting on the suspension system of a vehicle, the first is the force from the road disturbance, and the second is the load disturbance. Road unevenness could be of high magnitude and low frequency (such as mountains) and the small magnitude and large frequency (because of rough roads). Load disturbance includes the forces induced by the changing acceleration, braking and cornering. A suspension system should respond very smoothly against road disturbances and should be robust against load disturbances.

In automotive suspension, a control arm is a hinged suspension link between the chassis and the suspension or hub that carries the wheel. The chassis end of a control arm usually rubber busing, is attached by a single pivot. It controls the position of the tire end in only a single degree of freedom, maintaining the radial distance from the chassis mount. The single bushing does not control the arm from moving back and forth; this motion is constrained by a separate link or radius rod. Wishbones are triangular and have two widely spaced chassis bearings, which constrain the tire end of the wishbone from moving back and forth, controlling two degrees of freedom, and without requiring additional links. Most control arms form the lower link of a suspension with few designs using them as the upper link, usually with a lower wishbone. The lower arm should be sturdy, in many cases reported by the users of a particular vehicle, fluttering noise heard over the humps. This paper calculates the forces acting on the lower arm of a four-wheeler with critical loading conditions as the Finite Element Analysis carried for the McPherson type suspension system.

The static analysis performed to determine the location of maximum deflection and stress distribution while the vehicle is stationary and moving over the hump at different speeds. The free-free and constrained modal analysis performed on the suspension arm to find the natural frequency. The fatigue analysis performed to find the life of the arm. Further, by performing topological optimization leads to reduction in the weight.
Vehicles ranging from bullock carts to modern vehicles use sus-pension system such as, leaf springs, pneumatic and hydraulic systems. The main aim of using a suspension system is to prevent the shocks reaching the body of the vehicle from the rough roads and ensure that the vehicle's wheels remain in contact with the road. There are two types of forces acting on the suspension system of a vehicle, the first is the force from the road disturbance, and the second is the load disturbance.

Road unevenness could be of high magnitude and low frequency (such as mountains) and the small magnitude and large frequency (because of rough roads). Load disturbance includes the forces induced by the changing acceleration, braking and cornering. A suspension system should respond very smoothly against road disturbances and should be robust against load disturbances.The suspension system consists of spring, damper and strut. The function of spring is to absorb the shocks and store the energy, but damper (shock absorber) is required to dissipate it. Many types of springs are used in the automobile suspension system like leaf spring, helical coil spring, torsion spring, air spring, etc. Mostly hydraulic or pneumatic dampers are used in the car suspension having a piston and a rod with fluid inside it, which slows down the spring motion. [2]

2) **Gururaj Dhanu and Prof. R.S.Kattimani, “Comparison Study of Lower Control Arm with Different Materials” International Research Journal of Engineering and Technology, Volume: 03 Issue: 10, Oct 2016 -**

In suspension system, the control arm is very important component. The double wishbone suspension system contains lower control arm and upper control arm. The McPherson structural suspension system contains only lower control arm. Between wheel assembly and vehicle chassis the control arms are rigidly placed.
The control arm is connected to the chassis with the help of bush which is placed in pivot joints. Rear and front bushes are used in control arms. it has U-shaped configuration. At the apex of the control arm a ball joint receptacle is formed.

The ball joint receptacle is adapted to cooperate with a ball joint assembly and may include a ball joint housing integrally formed with the control arm. Typical modern control arm incorporates a separate ball joint housing which is inserted in to the apex of control arm. The bushing apertures are designed to retain pipe housings for mating engagement with a pivot bar assembly forming a portion of the vehicle suspension system.

The pivot bar typically extends through both bushing apertures allowing the control arm to pivot about the assembly in response to road conditions affecting the vehicle suspension system. For the construction of control arm high strength alloy steel is used. Control arm consist of modulus section which is between the apex and pivot points. Due to the various functions of control arm, it is one of the important components in suspension system. When vehicle takes turn various types of forces are acting on the wheels which are transmitted to the control arm through attachments such as ball joint assembly and so on, to the wheel. Here in this analysis the main concern is to find out the maximum stress region and stress values in control arm and compare these values with yield strength of materials and the result of experimental testing of control arm at the same load are mentioned in the load case of the control arm.

The lower control arm is an important part of front suspension system. It is used to control wheel trace and transmits the load exerting on the wheel by the ground to the various parts of the car. When the car was in motion, the lower control arm is subjected to complex loads alternating with time. There may be different shapes of lower control arms are available.

The software provides modules to carry out various types of analysis, such as structural analysis, thermal analysis, fluid analysis and computational fluid analysis. The current study involves study of mechanical properties of the material. There are basically two types of analysis one is static analysis and second one is dynamic analysis. Here in this project implicit static analysis is carried out to obtain results. Quadratic tetrahedron is also called ten node tetrahedron. It is second complete polynomial adherent of the isoparametric tetrahedron family. In this element the stress calculation is considerably better than the four nodes. This element proceeds the compensations of the existence of fully automatic tetrahedral mashers.

This is a cad model which can be analyzed to determine structural the stability during turning, to study the behavior of material under different material properties and to compare weight to strength ratio under different material conditions. To study the structural stability of lower control arm under turning conditions. Structural strength of lower control arm is good and safe to manufacture with either steel or aluminum. [3]

3) **Mohd Viqaruddina, D.Ramana Reddy, "Structural optimization of control arm for weight reduction and improved performance", Materials Today: Proceedings, 4(8),January 2017 -**

Typical in high performance automotive and aerospace applications is the demand for reduced vehicle mass while maintaining adequate performance and safety. Formula SAE competition is no exception. The nature of the autocross style course favors vehicles with good acceleration capabilities, and both the fuel economy and acceleration events add to the desire for a lightweight vehicle design.

In this paper, discussed the static analysis and torsion analysis of the Control arm by using Radios software and improve the stiffness of the Control Arm and weight reduction of component by changing the geometrical dimension and structural properties. This design is given by topology optimization for compare the base run analysis and optimized analysis. Meshing is carried out by using 10 nodes tetrahedral element in Hyper Mesh & topology optimization is carried out for the given design space.

The topology optimization given the idea of optimum material layout based on load & boundary conditions. Using optimum material layout, the component geometry is finalized by keeping the strength of component constant & 30% reduction in weight.
The main objective of this paper is to 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.
Suspension system is an integral part of automobile. It is used to prevent the road shocks being transmitted to vehicle frame and other components of vehicle. Its main function is to provide stability, safety and comfort. It maintains traction between the tyre and road surface. Lower control arm is an important component in the suspension system. It is present in both Mac- Pherson suspension and Double wishbone suspension system. Control arm connect the cars suspension to the actual vehicle frame.

One end of lower control arm is connected to knuckle through a ball joint and other end is attached to vehicle subframe. Lower control arm forms the unsprung weight of vehicle. High unsprung weight complicates wheel control issues under hard acceleration or braking. It may cause severe wheel hop, compromising traction and steering control. Unsprung weight increases the overall weight of suspension system and finally of vehicle.

The main objective of paper is to reduce the unsprung weight of vehicle; by reducing weight of lower control arm using topology optimization. The cost of manufacturing is also reduced. Topology optimization include removing unnecessary material from the base model to reduce weight using given sets of objectives and constraints.

Excess material is removed from the low stressed region of lower control arm. The existing design of left lower control arm from one of the light motor vehicles having Mac-Pherson suspension system is selected for the study (Maruti Suzuki Swift Dzire). It is essential to focus on the stress and deformation study to develop the changes in existing design. FEA approach is used for static structural analysis and topology optimization.

In order to carry out analysis, left lower control arm from one of the LCV is selected having Mac- Pherson suspension system. Reverse engineering of existing lower control arm is done using Coordinate Measuring Machine (CMM). With the help of the coordinates obtained CAD model is generated in CATIA software. In Ansys static structural analysis of control arm is carried out by applying required loading and boundary conditions. Excess material is removed from low stressed region by Topology optimization and the model redesigned and optimized. Static structural analysis of optimized CAD model is done to evaluate stresses and total deformation. [4]

Stability, road handling and comfort of vehicle depend on optimum design of suspension system. Mostly all passenger cars and light trucks use independent suspension system because of inherent advantages over rigid suspension systems. Suspension system is the term given to the system of springs, shock absorbers and linkages that connect a vehicle to its wheels. When a tire hits an obstruction, there is a reaction force and the suspension system tries to reduce this force. The size of this reaction force depends on the unsprung mass at each wheel assembly. In general, the larger the ratio of sprung weight to unsprung weight, the less the body and vehicle occupants are affected by bumps, dips, and other surface imperfections such as small bridges. A large sprung weight to unsprung weight ratio can also impact vehicle control. There are three different types of suspensions namely: Dependent (Rigid Axle), independent and semi -independent suspensions. In the independent suspension system, there are no linkages between two hubs of same axle and it allows each wheel to move vertically without affecting the opposite wheel.

4) **Zolkarnain Marjom, Mohd Hafizi Abdul Rahman, Mohd Shukor Salleh, Mohd Suffian Razak, Mohamad Ridzuan Mohamad Kamal, Liza Anuar and Nur Adzly Mohamad Saad, "Introduction of Design and Optimization of Front Lower Control Arm (FLCA) for C-Segment Passenger Car", International Journal of Engineering & Technology, 71,2018 -**

In this paper, the design optimization and fabrication process of aluminium cast for front lower control arm (FLCA) were investigated. 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 optimized 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.

Stability, road handling and comfort of vehicle depend on optimum design of suspension system. Mostly all passenger cars and light trucks use independent suspension system because of inherent advantages over rigid suspension systems. Suspension system is the term given to the system of springs, shock absorbers and linkages that connect a vehicle to its wheels.

When a tire hits an obstruction, there is a reaction force and the suspension system tries to reduce this force. The size of this reaction force depends on the unsprung mass at each wheel assembly. In general, the larger the ratio of sprung weight to unsprung weight, the less the body and vehicle occupants are affected by bumps, dips, and other surface imperfections such as small bridges. A large sprung weight to unsprung weight ratio can also impact vehicle control. [5]

5) **M.H.A. Rahman, M.S. Salleh, A. Abdullah, S.H. Yahaya, M.S.A. Razak, M.R.M. Kamal, Z.Marjom, L.Anuar and N.A.M. Saad, “A new design optimization of light weight front lower control arm” , Journal of Advanced Manufacturing Technology, Volume. 12 No. 1, January - June 2018 -**

This paper proposed the design optimization of aluminum 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 analyze 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.

In this literature it presents the design optimization of aluminum cast for the front lower control arm. CATIA software was utilized to design the lower control arm. Hyper works software was also used to analyze 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.

Aluminum alloy is widely used in automotive industry in order to give lightweight vehicle and improve energy efficiency. Nowadays aluminum alloy is widely used in body structure and closure such as door, hood, trunk lid and others. For example, Ford F150, 2015 model mostly used Aluminum sheets for the body structures closure panels used “mixed materials” on body structure using aluminum casting, high strength steels and sheet metals

There are many components involved in the front suspension assembly such as knuckle, lower control arm, spring, etc. This study focused on front lower control arm (FLCA) as a target light weight component. FLCA is a connecting linkage between knuckles to sub frame underneath vehicle. In general, lower control arm design needs to be robust to enable multi-loading and lighter weight as it serves as the main hardpoint alignment. As a part of chassis structure components, it plays a major role in anchoring suspension of hard points for lateral and longitudinal loading. In platform development of a vehicle, depending on vehicle crash strategies, lower control arm as well is to absorb or delay crash energy, particularly in crucial crash case such as 45° offset frontal impact.

The commercial part of front lower control arm is fabricated from metal stamping process. The main objective of this research is to design a new lightweight of front lower control arm for the C-Segment vehicle using topology optimization process. After several iterations of design and optimization process of aluminum cast FLCA, the weight reduction of aluminum cast FLCA achieved the target of 20%.

Based on the final design concept of FLCA, the total weight is 2.55 kg, which is about 25% of weight reduction compared with the current metal stamping FLCA weight 3.40 kg and still maintains the structural strength performance and fatigue durability performance. The new design of this aluminum cast lower control arm has unique design compared with the current commercial design.
It has shown some novelty in term of the design shape of the body part with a combination of the aluminum cast for body parts and also sleeve metal stamping for at rear lower control arm bush hardpoint. The I-beam cross section provides higher stiffness and moment for the parts to sustain the higher bending moment. [6]

6) **Kale AR, Tadamalle AP and Patil ND, "Analysis and Optimization of Lower Control Arm", Crimnson publishers, Evolution in mechanical engineering, ISSN: 2640-9690, August 07, 2018 -**

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.

In this research it presents to analyze and optimize the lower arm using finite element analysis. Control arm in automotive vehicle acts as a linkage between sprung and unsprung mass of vehicle.Weight reduction of vehicle component is one of the main concerns. The model of lower control arm is done in CAD 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 were 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 result of stress generated by static analysis of existing lower control arm is 223.62MPa. After optimization the result obtained for stress analysis of optimized lower control arm is 240.59MPa. Thus, the design is safe according to yield strength criteria. The weight of existing lower control arm is 2.6042kg and that of optimized part is 2.203kg. Thus, the weight reduction is 15.39% by using optimized control arm. The cost saving by using optimized lower control arm over existing control arm is 8.8%. [7]

7) **Swapnil S. Khode, Prof. Amol N. Patil, Prof. Amol B. Gaikwad, "Design Optimization of a Lower Control Arm of Suspension System in an LCV by using Topological Approach", International Journal of Innovative Research in Science, Engineering and Technology Volume 6, Issue 6, June 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 Mac-pherson suspension system and its optimization under static loading condition. This journal deals with Finite Element Analysis of the Lower control arm of Mac-pherson suspension system and its optimization under static loading condition. The existing design of lower control arm from one of the light commercial vehicles. In order to determine the deformation and stress distribution in the current design, the finite element analysis is carried out.

In this research it presents to optimize the lower control arm of Mac-pherson suspension system under the current boundary conditions for weight reduction. The baseline model of the lower control arm is created by using solid modelling software viz. CATIA. ANSYS Workbench is used for Finite Element Analysis and OPTISTRUCT solver module is used to generate the optimized model. The weight reduction in one lower control arm is observed to be 17.5%. As deflection and stress of modified LCA is within the range. In topology optimization process using Hyperworks Optistruct software, the base concept design was optimized based on standard suspension abusive load cases loading at FLCA hardpoint as shown in Table 1. The topology optimization is a method to optimize the design in design space with the constraint of loads and some boundary conditions. It is a stress-based optimization through load path on the geometry.
It gives the best selection of design based on load path on material to reduce the weight of material. In terms of FLCA design, the non-load path areas based on suspension abusive loading were eliminated to reduce the material. This process was done in several iterations to get the optimum minimum of 20% weight reduction as shown in Figure.

The lower control arm is the most vital component in a suspension system. There are two control arms, lower control arm and upper control arm. 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 other end supports the lower ball joint. Significant number of loads are transmitted through the control arm while it serves to maintain the contact between the wheel and the road and thus providing the precise control of the vehicle. There are many types of control arms are available. The selection of the arm is mainly based on the type of suspension system. Thus, the modified design is safe. Weight of the final optimized model is 0.99 kg. The total reduction in mass is observed 17.5% by keeping Factor of safety for optimized design within permissible limits. Thus, the objective of weight reduction of unsprung mass and cost reduction has been achieved. [8]

8) **Yong-Dae Kim, Jae-Eun Jeong, Jin-Su Park, In-Hyung Yang, Tae-Sang Park, Pauziah Binti Muhamad, Dong-Hoon Choi, Jae-Eung Oh, "Optimization of the lower arm of a vehicle suspension system for road noise reduction by sensitivity analysis", Mechanism and Machine Theory, 69, 278–302, 2013 -**

As advances in automotive design and manufacturing technology have emerged, the demand for noise reduction and ride quality improvement has increased. In particular, under normal driving conditions, drivers are most exposed to road noise, which can be unpleasant. Furthermore, it can reduce ride quality and brand awareness. Noises within the passenger compartment are categorized as structural and air-borne noises, depending on the transfer path characteristics.

This journal introduces an analyzed characteristics of road noise using vehicle tests and identified the 200– 230 Hz range as the most important frequency for road noise reduction. Moreover, vibration sources in the vehicle suspension system were identified through transfer path analysis and coherence analysis. In order to achieve noise reduction using sensitivity analysis, the lower arm of the vehicle suspension system was found to be the most important design variable.

For design optimization, we employed a robust and efficient sequential approximate optimization method, named PQRSM (Progressive Quadratic Response Surface Method) suitable for solving practical design optimization problems. The estimates based on a model proposed from optimization were in accord with the results of the experiment and road noise reduction was achieved by applying the optimally designed lower arm of the vehicle suspension system to a real vehicle. This study identified the vibration sources of the vehicle suspension system with superior coherence and proposed an optimal design to reduce road noise that is structure-borne noise and obtained to the following conclusions. [9]

9) **Liang Tang, JieWu, Jinhao Liu, Cuicui Jiang, and Wen-Bin Shangguan, "Topology Optimization and Performance Calculation for Control Arms of a Suspension" by Hindawi Publishing Corporation Advances in Mechanical Engineering, Article ID 734568, Volume 2014 -**

Lower control arm is an important part in the suspension system. It is connected between subframe and knuckle. It holds the vehicle wheels in alignment. High unsprung weight complicates steering control and traction control issues. This paper deals with the reduction of unsprung mass of vehicle by topology optimization of lower control arm. For analysis existing lower control arm of Mac- Pherson suspension system is selected. In this journal it presents the topology optimization model including ball joints and bushing for topology optimization of an aluminum CA is established, where a ball joint is simplified as rigid elements and the elastic properties of a rubber bushing are estimated using Mooney-Rivlin constitutive law.

A method for treating with multiple loads in topology optimization of CA is presented. Inertia relief theory is employed in the FEA model of the CA in order to simulate the large displacement motion characteristics of the CA. ACA is designed based on the topology optimization results, and the strength, natural frequency, and rigidity of the optimized CA are calculated.

The main contribution of this paper is the establishment of the topology optimization model for the CA with considerations of the elastics of ball joints and rubber bushings, and the method for treating with multiple loads in topology optimization of CA. The topology optimization results are compared with and without the modeling the stiffness of ball joints and rubber bushings. It is concluded that the elastics of ball joints and bushings should be modeled in the topology optimization of a Control Arm in order to meet the requirements of stress, stiffness, and first order natural frequency for the Control Arm.

The Topology optimization gives the optimum material layout according to the design space and loading condition using given sets of objectives and constraints. The main aim of topology optimization is to reduce weight by removing excess material from low stressed region. To solve any topology optimization problem, three parameters must be specified, namely design variables (material density), design objective (weight reduction) and design constraints (volume) Topology optimization is used to generate optimal shape of mechanical structure. Deformation in optimized model is found to be 8.84 mm. The maximum deformation of optimized design and existing design are not varying by considerable amount. The actual lower control arm is designed by analyzing CAD model. Pockets are made on the surface of control arm according to the CAD model. Structural analysis is done to determine the stress distribution and deformation by applying mechanical load. Critical areas of location predicted where the stress distribution is high. The yield strength of material is 415 MPa and generated von-Mises stress in lower control arm should be less than yield strength of material. [10]

CHAPTER 3: PROBLEM DEFINITION AND OBJECTIVES

3.1 Problem Definition:

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 display device and method, and more particularly, to a computer-aided structure analysis of A control arm and which is analyzed and designed, thereby meet the customer requirements of LCA.

This project is to optimize the A control arm by doe FFT analyzer and Impact Hammer Test by suggesting suitable material, and reducing sheet metal thickness, to reduce the batch production cost and to increase the strength of LCA.

3.2 Objectives:

1. To prepare CAD design using CATIA V5.
2. Static structural Analysis of Automotive A type control arm
3. Optimization of Automotive A type control arm.
4. To perform static structural analysis of the suspension arm using ANSYS Software.
5. To optimize the lower control arm for weight reduction" (unsprung weight) up to 5-20% and suggest alternate design.
6. Testing and validation of Automotive A type control arm with the help of Universal Testing Machine.
7. To compare the factor of safety for optimized and baseline design of lower control arm by keeping factor of safety for optimized design within permissible limits.
8. To perform experimentation on physical model of Lower Control Arm.

CHAPTER 4: METHEDOLOGY

The methodology of our project would be starting with the topic selection, then working on the topic by researching and creating a detailed plan for execution of the project. Then executing the project by manufacturing and then testing the model of the project. Then at last finishing the miscellaneous work and concluding the Project. The following is the stepwise proposed technical methodology:

- CAD Modeling of Existing Lower Control Arm –
  To prepare CAD design using CATIA V5 and Solidworks. The CAD model is prepared to apply all the dimensions and the features of the component in the analysis of the component.

- Analysis of Design in FEA of existing lower control arm
  Static structural Analysis of Automotive A type control arm. To find out the stiffness, buckling strength of the existing model.

- Apply Topology Optimization on the design and analyze
  Topology optimization (TO) is a shape optimization method that uses algorithmic models to optimize material layout within a user-defined space for a given set of loads, conditions, and constraints. Optimization of Automotive A type control arm.

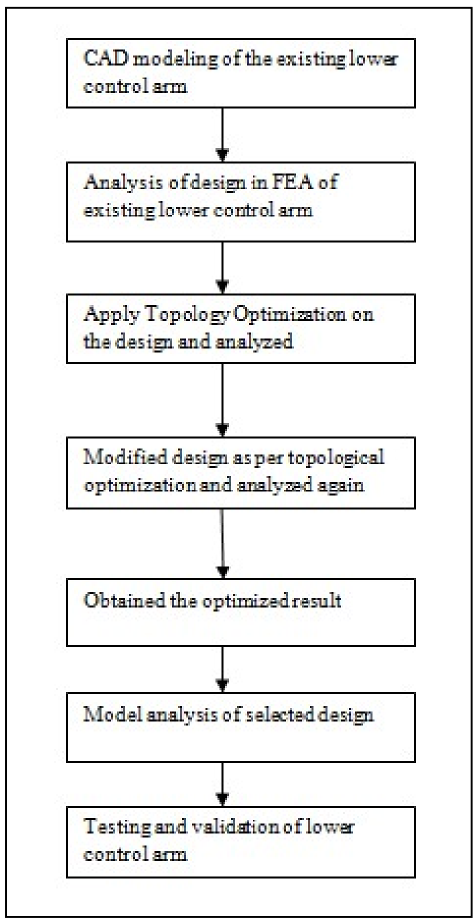
- Modified Design as per topological optimization and analyze again
  To find out the stiffness, buckling strength of the new model. To optimize the lower control arm for weight reduction (unsprung weight) up to 5-20% and suggest alternate design.

- Obtain the optimized results
  The results are obtained according to the applied parameters to the model in the software.

- Model analysis of selected design
  Finite element values of the existing design and new designs are compared and better design is identified. Influencing parameters which are affecting the response are identified. The results obtained are compared with the real time so that the results match. Finally, the analysis becomes successful.

- Testing and validation of lower control arm.
  Testing and validation of Automotive A type control arm with the help of Universal Testing Machine.

Table 1 - Flowchart for the Methodology



CHAPTER 5: DESIGN, ANALYSIS AND DESIGN OPTIMIZATION

5.1 FEA (FINITE ELEMENT ANALYSIS)

The finite element method (FEM), is a numerical method for solving problems of engineering and mathematical physics. Typical problem areas of interest include structural analysis, heat transfer, fluid flow, mass transport, and electromagnetic potential. The analytical solution of these problems generally require the solution to boundary value problems for partial differential equations. The finite element method formulation of the problem results in a system of algebraic equations. The method yields approximate values of the unknowns at discrete number of points over the domain. To solve the problem, it subdivides a large problem into smaller, simpler parts that are called finite elements.

Finite element analysis is based on principles that include boundary conditions, such as forces and pressures, as well as three governing equations:

- Equilibrium equations, which find when the opposing forces or influences are balanced.
- Strain-displacement relations, which measure the deformation that the design experiences under any given external impacts.
- Constitutive equations, which are relations between two physical quantities, specific to the given metal or substance, which predict the material's response to external stimuli.

For finite element analysis to perform its necessary simulations, a mesh -- containing millions of small elements that together form the shape of a structure -- must be created. Calculations must be performed on every single element; the combination of each of these individual answers provides the final result for the full structure.

CAD: -

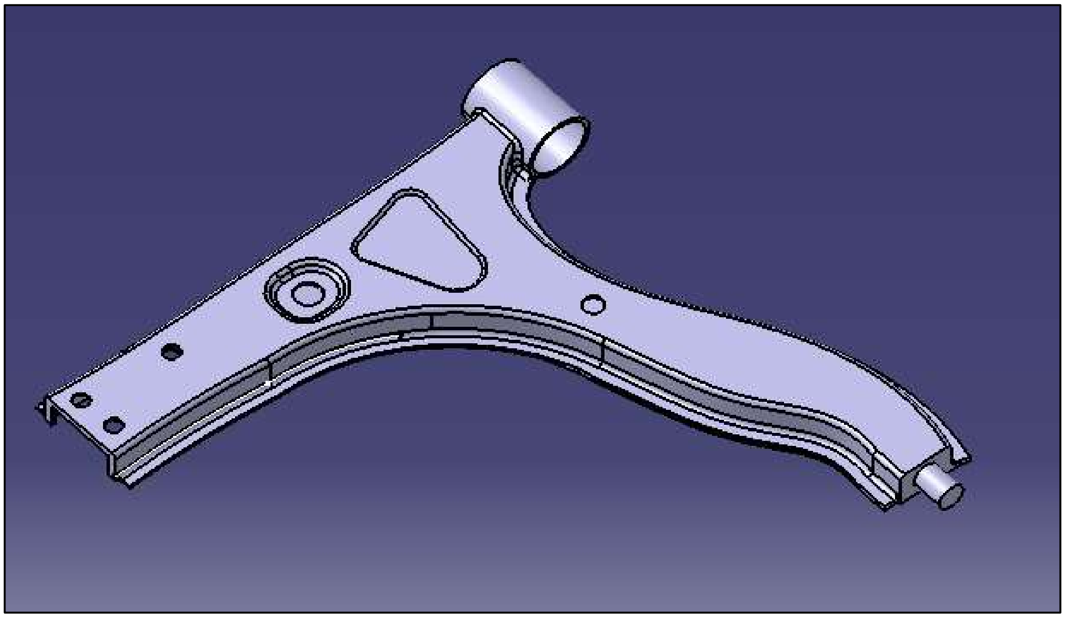


Fig. 5.1 - CATIA model

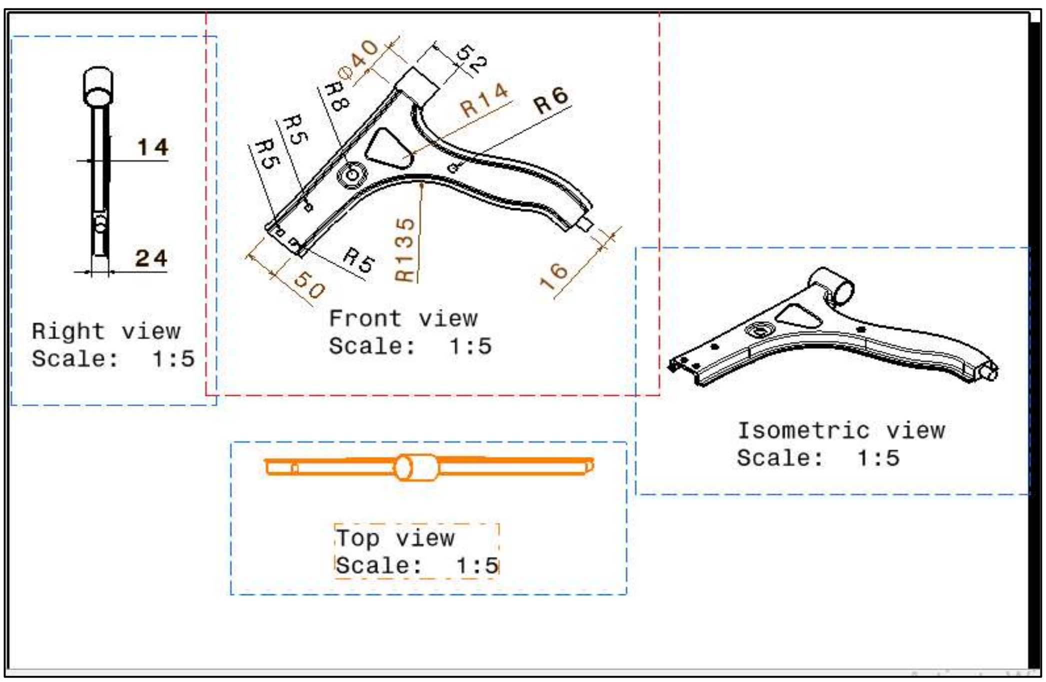


Fig.5.2 - Drafting of existing model

Fig.5.3 – Complete Model Assembly

Fig.5.4 – Ball Joint CAD

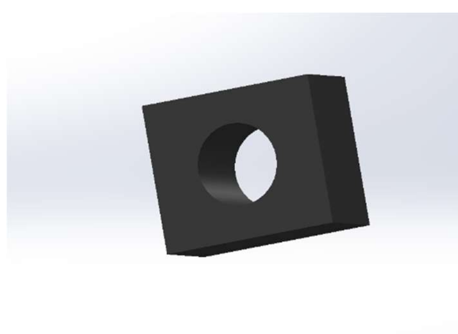


Fig.5.5 – Ball Joint CAD

In present research for analysis ANSYS (Analysis System) software is used. Basically, its present FEM method to solve any problem. Following are steps in detail

1. Geometry
2. Discretization (Meshing)
3. Boundary condition
4. Solve (Solution)
5. Interpretation of results

Step 1: Details of material namely copper, steel, grey cast iron, composite material, fluid domain material is defined in engineering data. i.e., ANSYS default material is structural steel.

Step 2: Import of geometry created in any CAD software namely CATIA, PRO E, SOLIDWORK, INVENTOR etc. in geometry section. If any correction is to be made it can be created in geometry section in Design modeller or space claim.

Step 3: In model section after import of component

- Material is assigned to component as per existing material
- Connection is checked in contact region i.e., bonded, frictionless, frictional, no separation etc. for multi body components.
- Meshing or discretization is performed i.e., to break components in small pieces (elements) as per size i.e., preferably tetra mesh and hexahedral mesh for 3D geometry and for 2 D quad or trial are generally preferred.

Step 4: Boundary condition are applied as per analysis namely in fixed support, pressure, force, displacement, velocity as per condition.

Step 5: Now problem is well defined and solve option is selected to obtain the solution in the form of equivalent stress, strain, energy, reaction force etc.

Table 2 - Table of Material properties

Material	Structural Carbon Steel – ASTM A36
Tensile Ultimate Strength	460 MPA
Tensile Yield Strength	250 MPA
Young's Modulus	2x10^11 PA
Poission's Ratio	0.3
Bulk Modulus	1.6667x10^11 PA
Shear Modulus	7.6923x10^11 PA

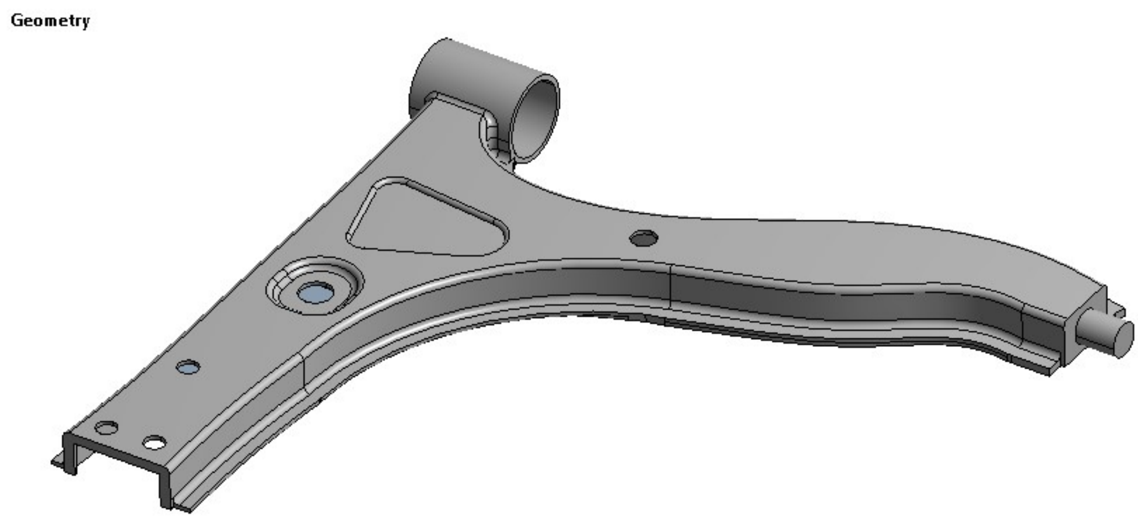


Fig.5.6 - Existing design of lower control arm

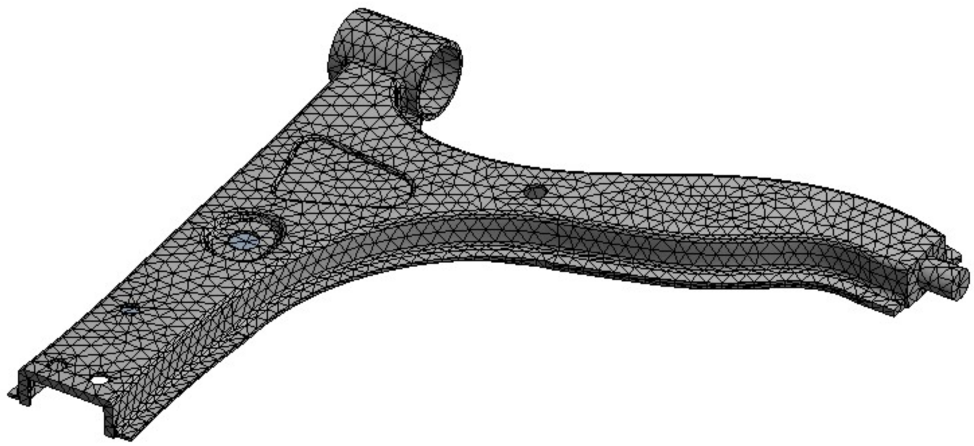


Fig.5.7 - Meshing of model

Table 3 - Meshing of model

Details of "Body Sizing" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Type	Element Size
Element Size	4.0 mm

Elements Details:

Table 4 - Elements Details

Statistics	
Nodes	28547
Elements	14197

5.2 Boundary Condition

A boundary condition for the model is the setting of a known value for a displacement or an associated load. For a particular node you can set either the load or the displacement but not both.

The main types of loading available in FEA include force, pressure and temperature. These can be applied to points, surfaces, edges, nodes and elements or remotely offset from a feature. The way that the model is constrained can significantly affect the results and requires special consideration. Over or under constrained models can give stress that is so inaccurate that it is worthless to the engineer. In an ideal world we could have massive assemblies of components all connected to each other with contact elements but this is beyond the budget and resource of most people. We can however, use the computing hardware we have available to its full potential and this means understanding how to apply realistic boundary conditions.

Details of Model:

Vehicle Name - Maruti Suzuki Baleno

Weight – 865 KG

So 865 x 9.81 = 8485 N

Weight on Each Wheels Considered as – 8485/4 = 2121.4 N

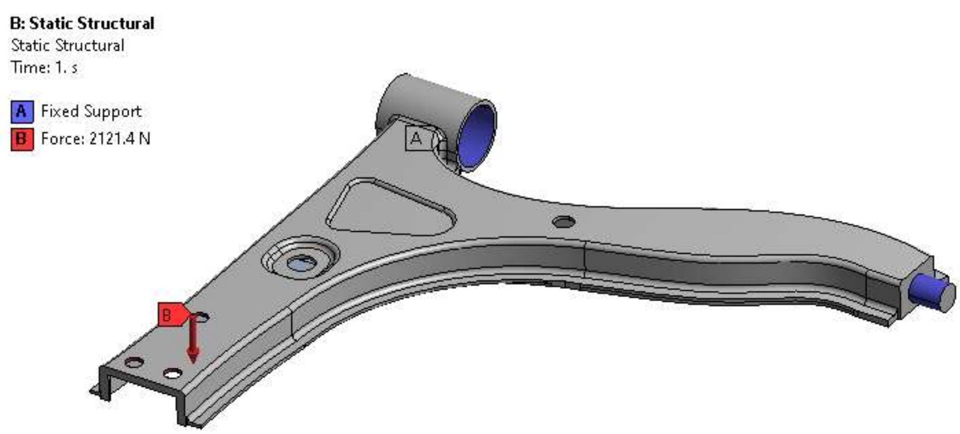


Fig.5.8 - Boundary condition of model

Total Deformation

The total deformation & directional deformation are general terms in finite element methods irrespective of software being used. Directional deformation can be put as the displacement of the system in a particular axis or user defined direction. Total deformation is the vectors sum all directional displacements of the systems.

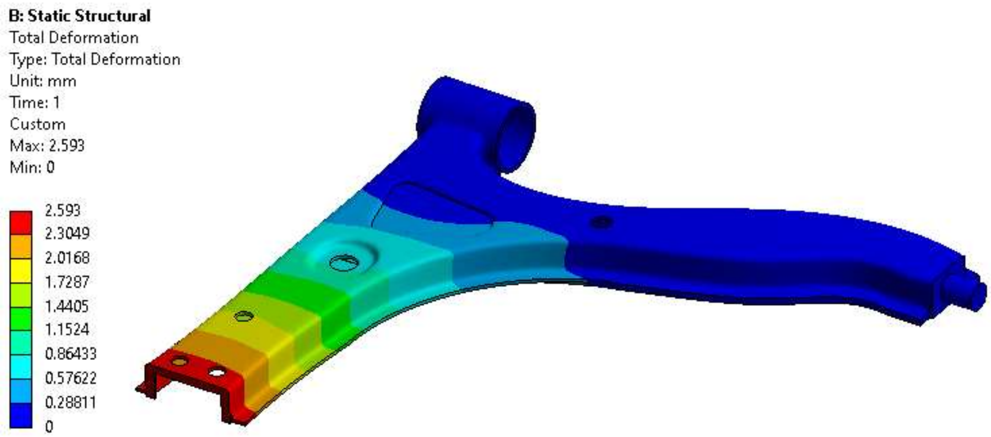


Fig.5.9 - Total Deformation of model

Steps in Topology Optimization

The topology optimization consists of the following sequence of steps.

• Define the design space

• Define optimization parameters

• Material removal process and detail design

Step 1: Details of material namely copper, steel, grey cast iron, composite material, fluid domain material is defined in engineering data. i.e., ANSYS default material is structural steel.

Step 2: Import of geometry created in any CAD software namely CATIA, PRO E, SOLIDWORK, INVENTOR etc. in geometry section. If any correction is to be made it can be created in geometry section in Design modeller or space claim.

Step 3: In model section after import of component

- Material is assigned to component as per existing material
- Connection is checked in contact region i.e., bonded, frictionless, frictional, no separation etc. for multi body components.

- Meshing or discretization is performed i.e., to break components in small pieces (elements) as per size i.e., preferably tetra mesh and hexahedral mesh for 3D geometry and for 2 D quad or trial are generally preferred.

Step 4: Boundary condition are applied as per analysis namely in fixed support, pressure, force, displacement, velocity as per condition.

Step 5: Now problem is well defined and solve option is selected to obtain the solution in the form of equivalent stress, strain, energy, reaction force etc.

Step 6: Topology optimization tool is selected from ANSYS list view and drag and dropped in solution section of static structural so that it takes its all-boundary condition, geometry, all details and perform topology optimization on selected component.

Step 7: Define the topology density and element density section as output. After performing topology algorithm on component, selected parts contain red, brown and grey colour on it. So, red region colour indicates material removal area along with marginal (brown colour) and grey colour to keep material.

Step 8: After selection of specific shape material is removed from component and reanalysis of component is performed to observe the sustainability of existing optimized component under same boundary condition.

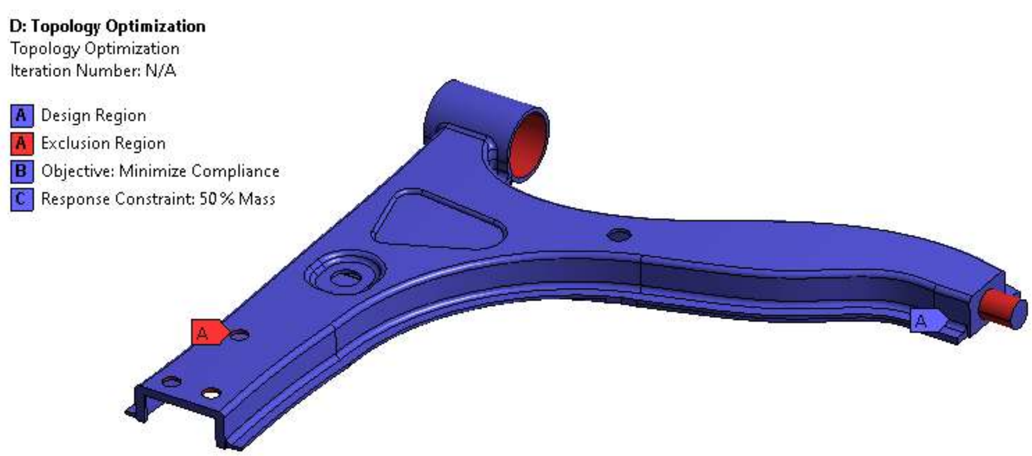


Fig.5.10 - Boundary condition for topology optimization region

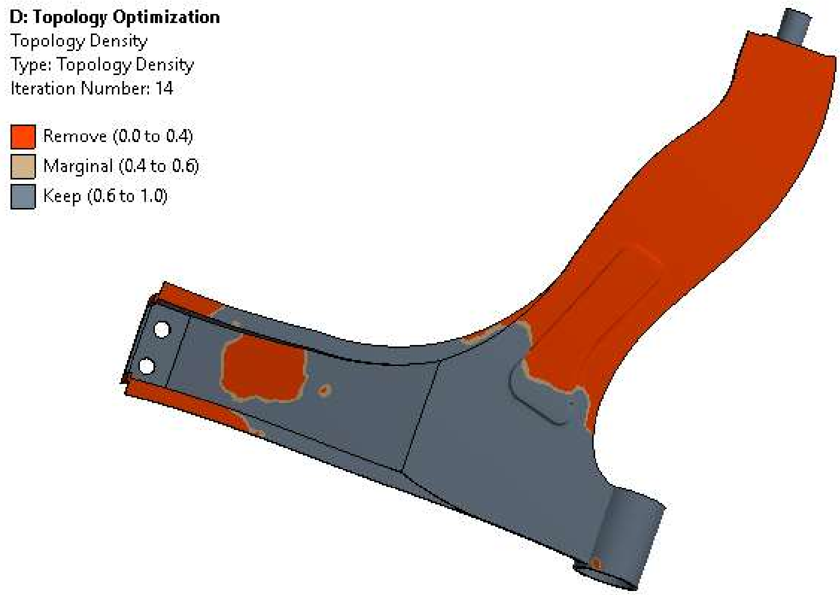


Fig.5.11 - Topology optimized results

Fig.5.12 - Topology optimized results

- Red region indicates material removal area region.

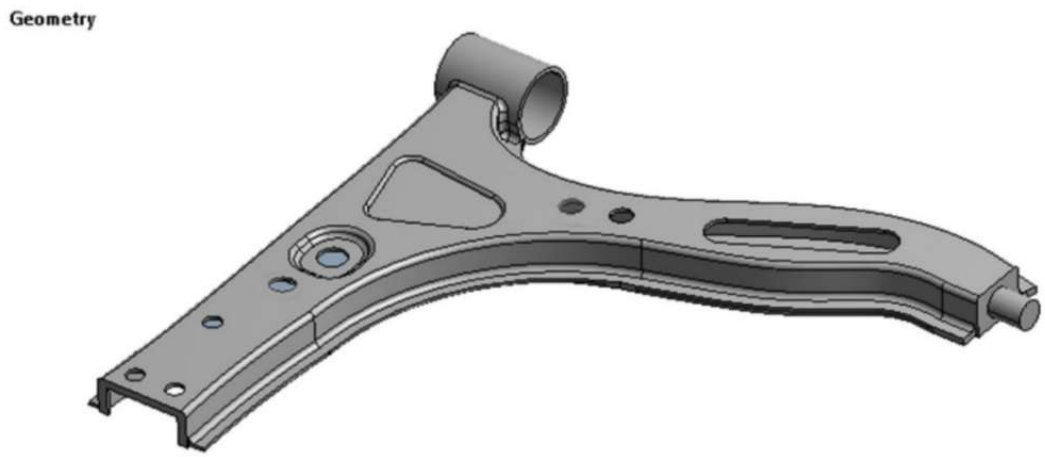


Fig.5.13 - Optimized lower control arm

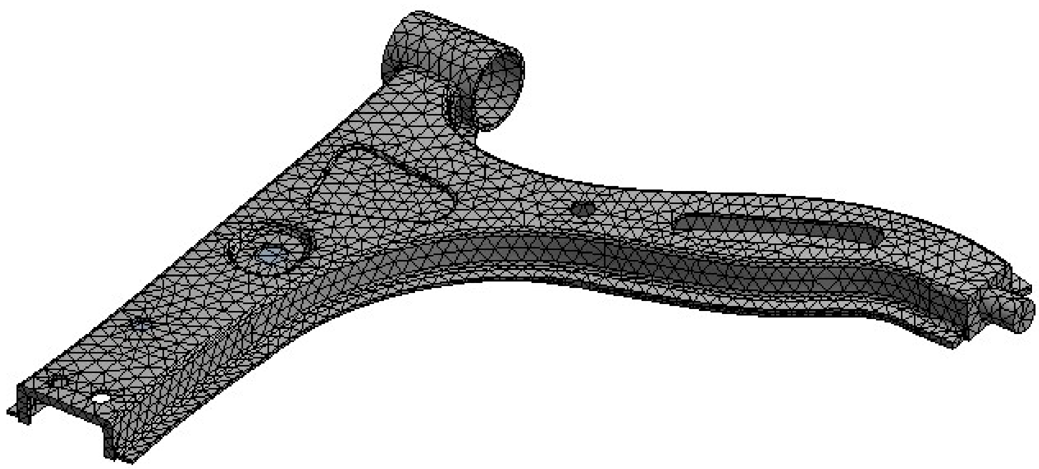


Fig.5.14 - Details of optimized lower control arm

Table 5 - Details of optimized lower control arm

Statistics	
Nodes	27382
Elements	13451

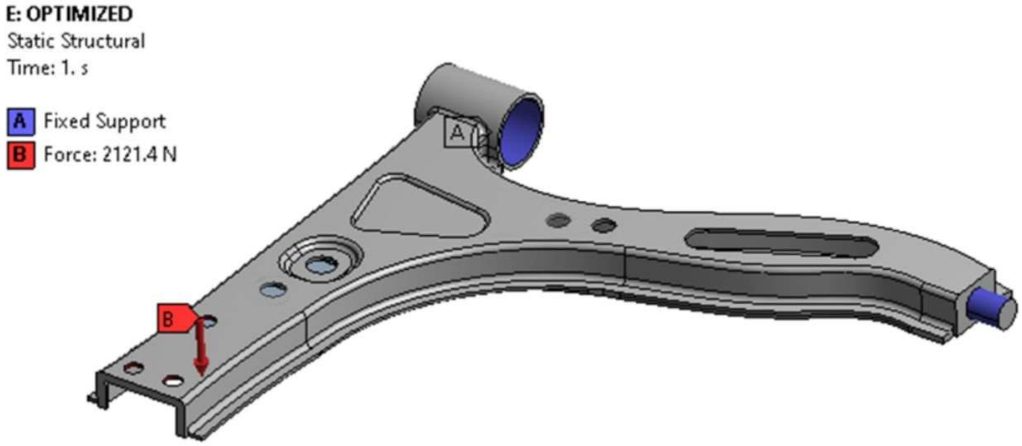


Fig.5.15 - Boundary condition for optimized lower control arm

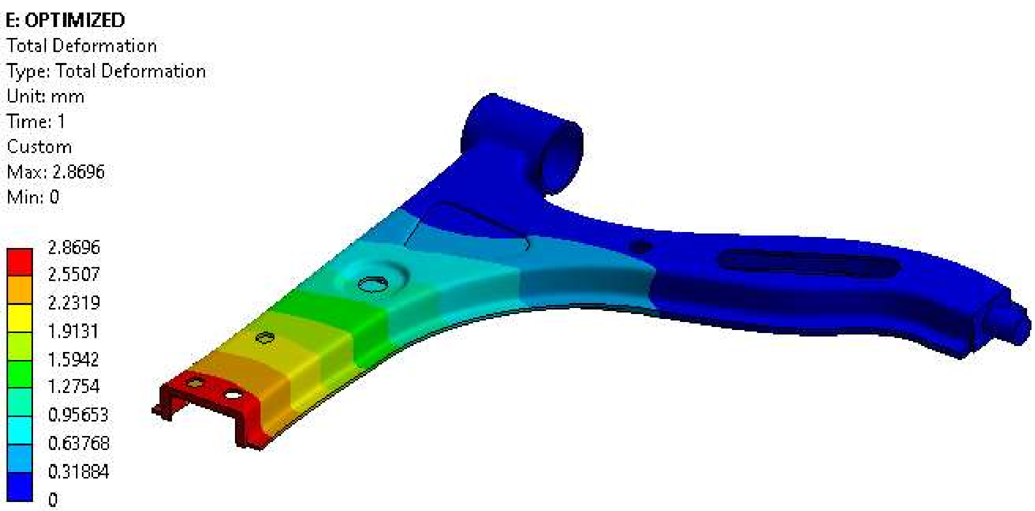


Fig.5.16 - Optimized lower control arm deformation result

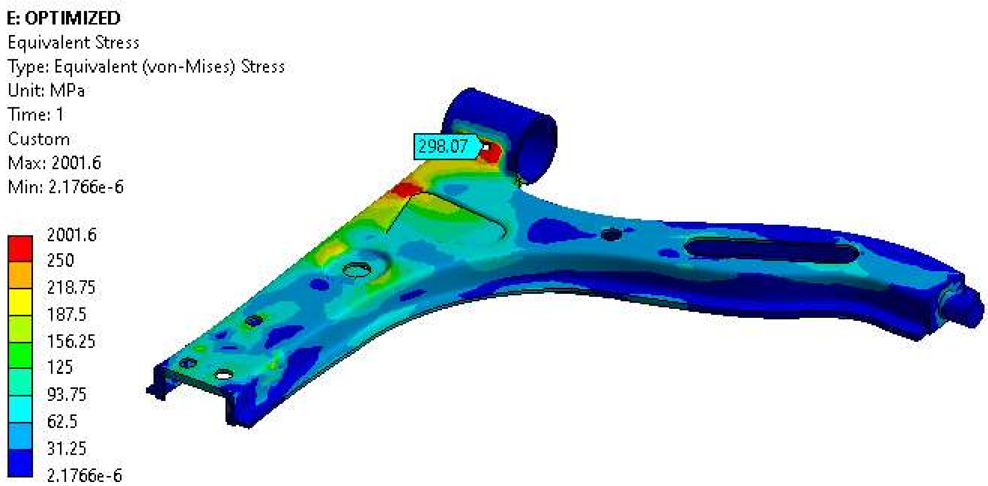


Fig.5.17 - Optimized lower control arm equivalent stress result

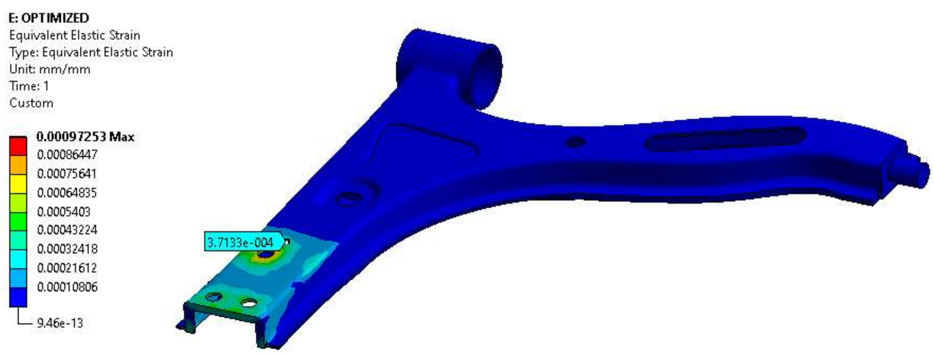


Fig.5.18 - Optimized lower control arm equivalent strain result

- Strain is observed around 371 microns.

5.3 Experimental Test

A Universal Testing Machine (UTM) is used to test both the tensile and compressive strength of materials. Universal Testing Machines are named as such because they can perform many different varieties of tests on an equally diverse range of materials, components, and structures.

Universal Testing Machines can accommodate many kinds of materials, ranging from hard samples, such as metals and concrete, to flexible samples, such as rubber and textiles. This diversity makes the Universal Testing Machine equally applicable to virtually any manufacturing industry.

Table 6 – Specification of Universal Testing Machine

1	Max Capacity	400KN
2	Measuring range	0-400KN
3	Least Count	0.04KN
4	Clearance for Tensile Test	50-700 mm
5	Clearance for Compression Test	0- 700 mm
6	Clearance Between column	500 mm
7	Ram stroke	200 mm

8	Power supply	3 Phase, 440Volts, 50 cycle. A.C
9	Overall dimension of machine (L*W*H)	2100*800*2060
10	Weight	2300Kg

Fig.5.16 - Testing in Universal Testing Machine

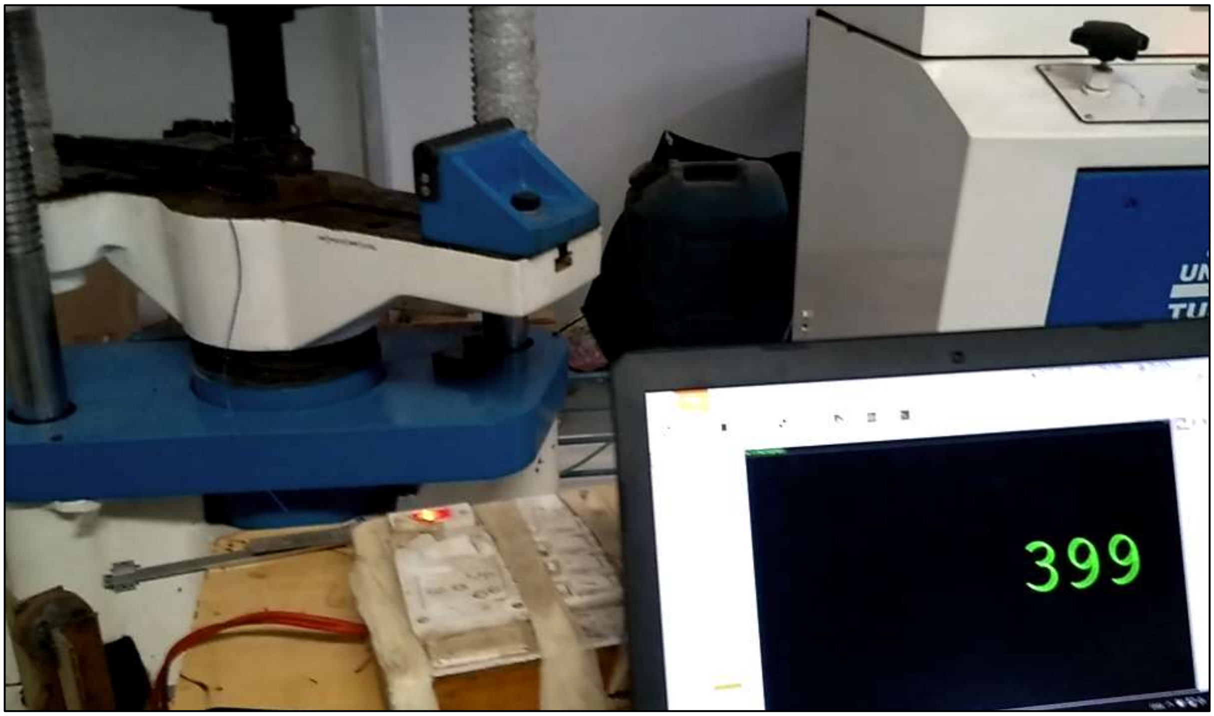


Fig.5.17 - Experimental testing results

Table 7 -Evaluation of Results

S. No	Parameters	Existing component		Optimized component	
d1	Deformation (mm)	Max	2.593mm	2.869 mm	
		Min	0 mm	0 mm	
2	Von-mises stress (Mpa)	Max	350.22 Mpa	399 Mpa	
		Min	0.0012 MPa	0.0014 MPa	
3	Mass reduction		2.55	2.39	6.07% weight reduction
4	Factor of Safety	1.31		1.15	

CHAPTER 6: RESULTS AND DISCUSSION

6.1 Results obtained from the project

Elements Details of Ansys -

- Nodes – 28547 for Existing component and 27382 for Optimized component.
- Elements - 14197 for Existing component and 13451 for Optimized component.

Universal Testing Machine Results -

- The Deformation Maximum is 2.593mm for Existing component and 2.869 mm for Optimized component.
- The Deformation Minimum is 0mm for Existing component and 0mm for Optimized component.
- The Von-mises stress Maximum are 350.22 MPA for Existing component and 399 MPA for Optimized component.
- The Von-mises stress Minimum are 0.0012 MPa for Existing component and 0.0014 MPA for Optimized component.
- Mass of the Existing component is 2.55 KG for Existing component and 2.39 KG (6.07% weight reduction) for Optimized component.
- Factor of Safety of the Existing component is 1.31 for existing component and 1.15 for Optimized component.

The existing design has been modified, by reducing the thickness of the existing profile. The optimization of lower control arm is done by applying the Design of Experiments method. The parameters are identified. The FEA is done on LCA and the buckling load has been compared with the existing component. The sheet metal thickness of the new design has been reduced from 3.6 mm to 3mm. For the mass production the cost has been reduced to Rs.34000 for 2000 products in a batch as assumed. For the modified design the structural rigidity has been increased for the LCA, when compared to the existing design. Finally, mass of the control arm has been reduced up to 6.07 % when compared with existing model.

CHAPTER 7: CONCLUSION

This project can be extended to satisfy the following requirements as the key objectives:

- Implementing new design changes in the existing model can be done
- Identifying more materials for the proposed design and new design
- Increasing the number of levels of the response
- Based on observations design improvements will be made in terms of shape, size and material based on design modification objectives.
- The study will focus on existing design performance, advantage and limitations.

As deflection and stress of modified Lower Control Arm is within the range. Thus, the modified design is safe. Weight of the final optimized model is 2.39 kg. The total reduction in mass is observed 6.07% by keeping Factor of safety for optimized design within permissible limits.

Thus, the objective of weight reduction of unsprung mass and cost reduction has been achieved.

REFERENCES AND BIBLIOGRAPHY

[1] Mohd Hafizi Abdul Rahman, Mohd Shukor Salleh, Mohd Suffian Razak, Mohamad Ridzuan Mohamad Kamal, Zolkarnain Marjom, Liza Anuar and Nur Adzly Mohamad Saad, "Introduction of Design and Optimization of Front Lower Control Arm (FLCA) for C-Segment Passenger Car", International Journal of Engineering & Technology, 71,2018.

[2] M. Sadiq A. Pachapuri, Ravi G. Lingannavar, Nagaraj K. Kelageri, Kritesh K. Phadate, "Design and analysis of lower control arm of suspension system" Materials Today: Proceedings,47(2), May 2021.

[3] Gururaj Dhanu and Prof. R.S.Kattimani, "Comparison Study of Lower Control Arm with Different Materials" International Research Journal of Engineering and Technology, Volume: 03 Issue: 10, Oct 2016.

[4] Mohd Viqaruddina, D.Ramana Reddy, "Structural optimization of control arm for weight reduction and improved performance", Materials Today: Proceedings, 4(8),January 2017

[5] Zolkarnain Marjom, Mohd Hafizi Abdul Rahman, Mohd Shukor Salleh, Mohd Suffian Razak, Mohamad Ridzuan Mohamad Kamal, Liza Anuar and Nur Adzly Mohamad Saad, "Introduction of Design and Optimization of Front Lower Control Arm (FLCA) for C-Segment Passenger Car", International Journal of Engineering & Technology, 71,2018.

[6] M.H.A. Rahman, M.S. Salleh, A. Abdullah, S.H. Yahaya, M.S.A. Razak, M.R.M. Kamal, Z.Marjom, L.Anuar and N.A.M. Saad, "A new design optimization of light weight front lower control arm" , Journal of Advanced Manufacturing Technology, Volume. 12 No. 1, January - June 2018.

[7] Kale AR, Tadamalle AP and Patil ND, "Analysis and Optimization of Lower Control Arm", Crimnson publishers, Evolution in mechanical engineering, ISSN: 2640-9690, August 07, 2018.

[8] Swapnil S. Khode, Prof. Amol N. Patil, Prof. Amol B. Gaikwad, “Design Optimization of a Lower Control Arm of Suspension System in an LCV by using Topological Approach”, International Journal of Innovative Research in Science, Engineering and Technology Volume 6, Issue 6, June 2017.

[9] Yong-Dae Kim, Jae-Eun Jeong, Jin-Su Park, In-Hyung Yang, Tae-Sang Park, Pauziah Binti Muhamad, Dong-Hoon Choi, Jae-Eung Oh, “Optimization of the lower arm of a vehicle suspension system for road noise reduction by sensitivity analysis”, Mechanism and Machine Theory, 69, 278–302, 2013.

[10] Liang Tang, JieWu, Jinhao Liu, Cuicui Jiang, and Wen-Bin Shangguan, “Topology Optimization and Performance Calculation for Control Arms of a Suspension” by Hindawi Publishing Corporation Advances in Mechanical Engineering, Article ID 734568, Volume 2014.

[11] Hazem ali attia, “Dynamic modeling of the double wishbone motor-vehicle suspension system”, European journal of mechanics A/solids 21, 167-174, 2002.

[12] J.C. fauroux, B.C. bouzgarrou, “Dynamic obstaclecrossing of a wheeled rover with double wishbone suspension”. French institute for advanced mechanics (IFMA), EA3867, FR TIMS / CNRS 2856.

[13] Vdovin D & Chichekin I (2016), Loads and Stress Analysis Cycle Automation in the Automotive Suspension Development Process. Procedia Engineering 150, 1276–1279.

[14] Nikoo MF, Azizi H, Parvin N & Naghibi HY (2016), The influence of heat treatment on microstructure and wear properties of friction stirs welded AA6061-T6 / Al 2 O 3 nanocomposite joint at four different traveling speed. Journal of Manufacturing Processes 22, 90–98.

[15] Design optimization of a carbon fiber reinforced composite automotive lower arm, Do-Hyoung Kim a, Dong-Hoon Choi a, HakSung Kim, Composites: Part B 58 (2014) 400–407.

[16] Kim Y, Jeong J, Park J, Yang I, Park T, Muhamad PB, Choi D & Oh J (2013), Optimization of the lower arm of a vehicle suspension system for road noise

reduction by sensitivity analysis. DOI: 10.1016/j.mechmachtheory.2013.06.010.

[17] Chen JK, Hung HY, Wang CF & Tang NK (2017), Effects of casting and heat treatment processes on the thermal conductivity of an Al-Si-Cu-Fe-Zn alloy. International Journal of Heat and Mass Transfer 105, 189–195.

[18] Dae Sung Joo, Nizar AL-Holou, Jonathan M. Weaver, Tarek Lahdhirt, and Faysal Al-Abbas, "Nonlinear modeling of vehicle suspension system", American control conference, vol. 1, 2000, pp 115-119.

[19] Dong-Chan Lee and Jeong-Ick Lee, "Structural optimization concept for the design of an Aluminium control arm", Proceedings of The Institution of Mechanical Engineers Part D-journal of Automobile Engineering, vol. 217, 2003, pp 647-656.

[20] Hiten kumar Patel, "optimization of an aerospace component die design using metal forming simulation capability of hyper-form" HTC (hyper- works technology conference) 2008, pp 6-7.

PHOTOGRAPH OF LOWER CONTROL ARM MODEL

WORK FLOW

Work	Start Date	Finishing Date
General Planning	15/08/2022	29/08/2022
Approval	03/09/2022	27/09/2022
Purchasing	28/09/2022	02/10/2022
Design	03/11/2022	17/11/2022
Analysis of Exiting Component	18/11/2022	02/12/2022
Testing of Existing Component	03/12/2022	13/12/2022
Analysis for optimization	15/12/2022	21/01/2023
Design of Optimized Component	22/01/2022	01/02/2023
Fabrication of Optimized Component	02/02/2023	16/02/2023
Testing of Optimized Component	17/02/2023	20/02/2023
Project Finishing	21/02/2023	26/02/2023