Weight Reduction by Static Structural Analysis of Automotive Lower Control Arm

Aniruddha Jadhav#1, Nilay Pawale *2, Atharva Kalokhe #3, Aditya Bangale #4

#*Department of Mechanical Engineering,*

Smt Kashibai Navale College of Engineering, Savitribai Phule Pune University, Pune

1Aniruddhaj7727@gmail.com

2nilaypawale.skncoe.mech@gmail.com

3atharvkalokhe05@gmail.com

4bangale.adi@gmail.com

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.

***Keywords*— Lower Control Arm, Suspension System, Weight Reduction, Ansys, Catia**

I. 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 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.

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.

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 aluminium alloy can give a good weight reduction to vehicle and improve the fuel economy [1].

II. LITERATURE REVIEW

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.[2]
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.[3]

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.[3]

III. PROBLEM DEFINITION AND OBJECTIVES

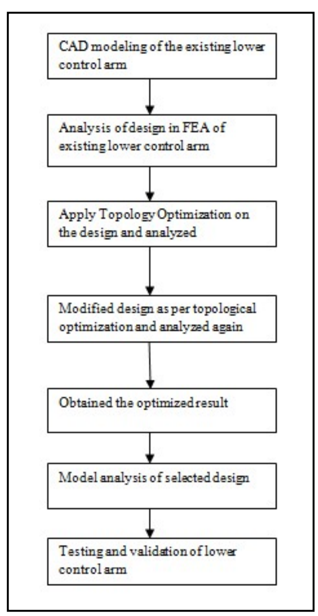
Chassis parts are a critical part of a vehicle, leaving no room for error in the design and quality the present process relates to a computer-aided structure analysis and design graphic display device and method, and more particularly, to a computer-aided structure analysis of A control arm and which is analyzed and designed, thereby meet the customer requirements of LCA.

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. 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.
7. To perform experimentation on physical model of Lower Control Arm.

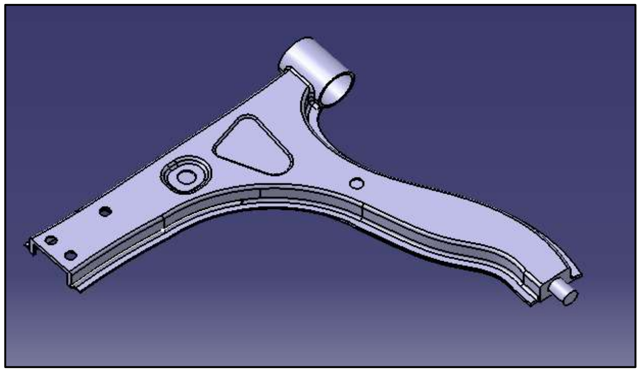
IV. METHODOLOGY

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.



V. DESIGN OF LOWER CONTROL ARM

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 requires 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.



Lower Control Arm 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.

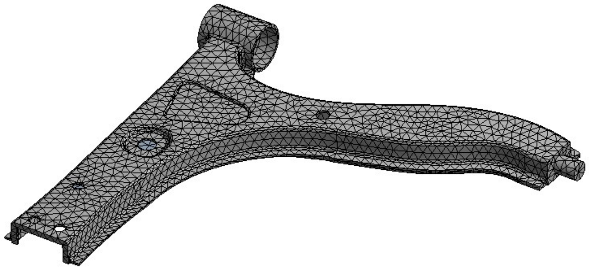
Step 4: Boundary condition are applied as per analysis namely in fixed support, pressure, force, displacement, velocity as per condition.

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 1. Material properties

Properties of Outline Row 3: Structural Steel

	A	B	C
1	Property	Value	Unit
2	Material Field Variables	Table	
3	Density	7.85E-09	tonne mm^-3
4	Isotropic Secant Coefficient of Thermal Expansion		
5	Coefficient of Thermal Expansion	1.2E-05	C^-1
6	Isotropic Elasticity		
7	Derive from	Young's Modulus and Poiss...	
8	Young's Modulus	2E+05	MPa
9	Poisson's Ratio	0.3	
10	Bulk Modulus	1.6667E+05	MPa
11	Shear Modulus	76923	MPa



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

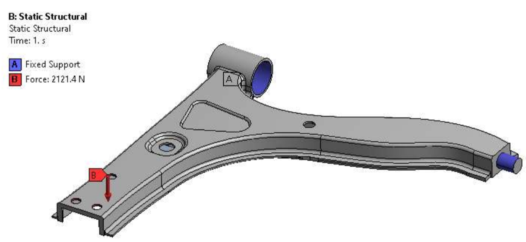
Fig 1. Meshing of model

Statistics	
Nodes	28547
Elements	14197

Fig 2. Elements Details

A. 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.



VEHICLE NAME- MARUTI SUZUKI BALENO
WEIGHT – 865 KG
SO 865 X 9.81 – 8485 N
WEIGHT ON EACH WHEEL IS CONSIDERED AS 8485/4 – 2121.4 N

Fig 3. Boundary condition of model

B. 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.

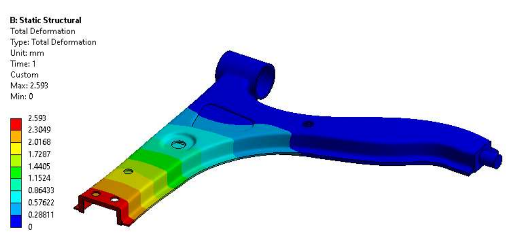


Fig 4. Total Deformation of model

C. 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.

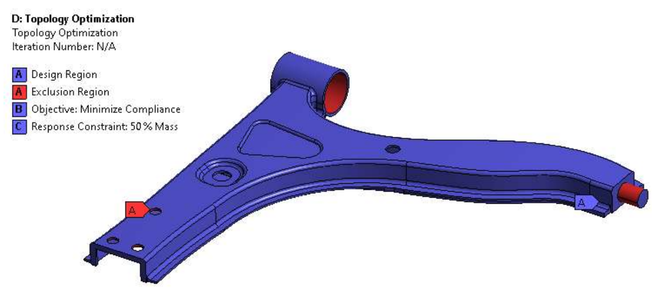


Fig 5. Boundary condition for topology optimization region

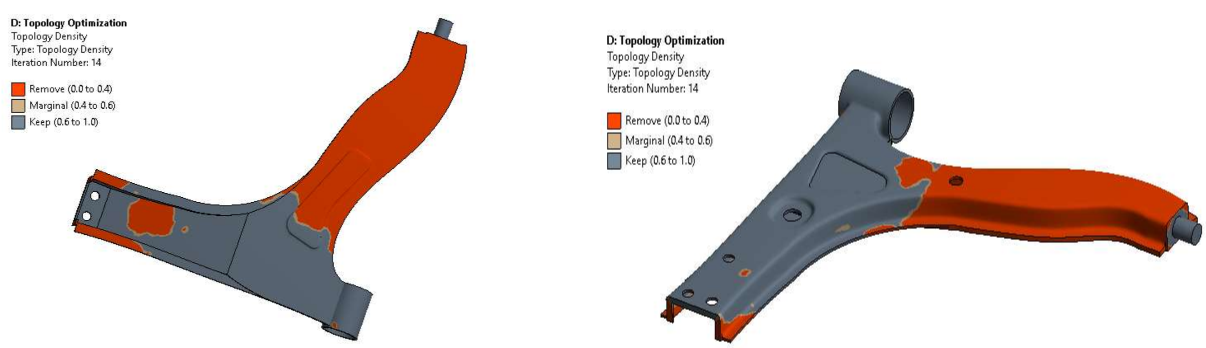


Fig 6. Topology optimized results

- Red region indicates material removal area region.

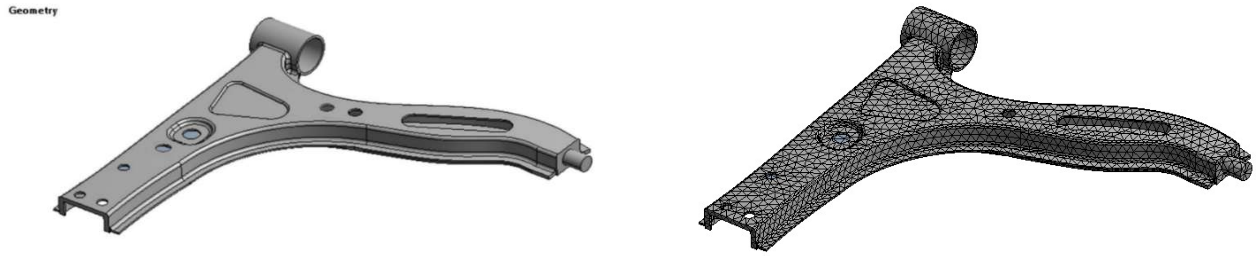


Fig 7. Optimized lower control arm

Statistics	
Nodes	27382
Elements	13451

Fig 8. Details of optimized lower control arm

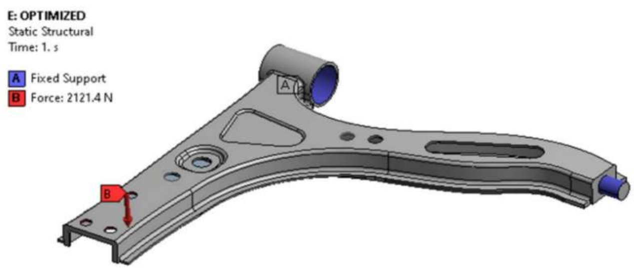


Fig 9. Boundary condition for optimized lower control arm

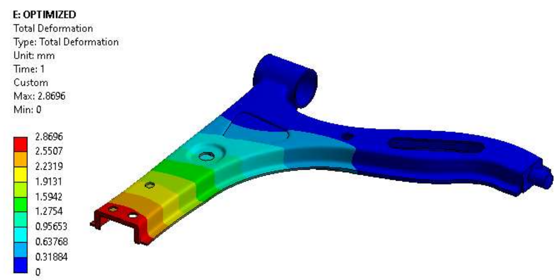


Fig 10. Optimized lower control arm deformation result

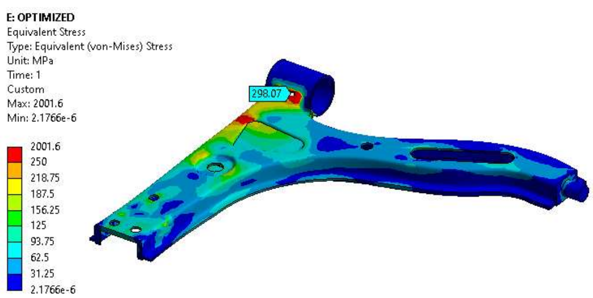


Fig 11. Optimized lower control arm equivalent stress result

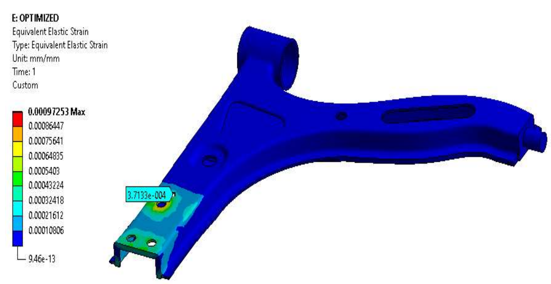


Fig 12. Optimized lower control arm equivalent strain result

- Strain is observed around 371 microns.

VI. 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 2 -SPECIFICATION OF UTM

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 13. Testing in Universal Testing Machine

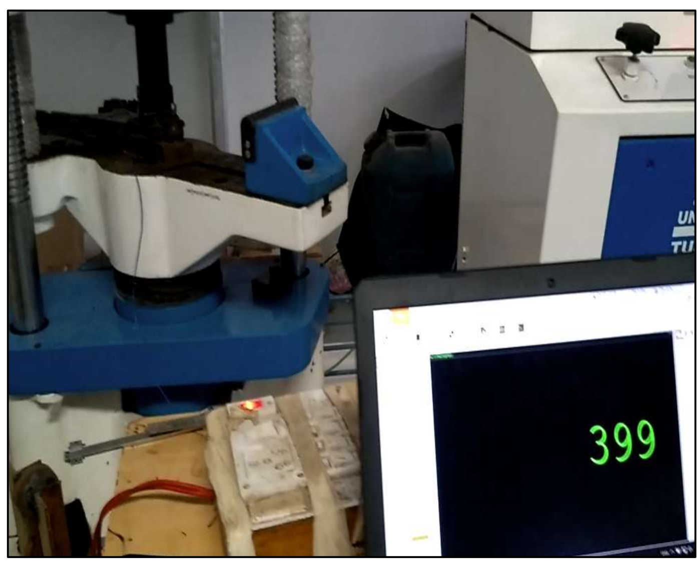


Fig 14. Experimental testing results

TABLE 3 - 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	

VII. SCOPE OF FUTURE WORK

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.

VIII. CONCLUSION

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.

ACKNOWLEDGEMENT

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

We express our immense pleasure and thankfulness to all the teachers and staff of the Department of Mechanical Engineering of Shrimati Kashibai Navale College of Engineering for their co-operation and support.

REFERENCES

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

[2] Design and analysis of lower control arm of suspension system, M. Sadiq A. Pachapuri, Ravi G. Lingannavar, Nagaraj K. Kelageri, Kritesh K. Phadate Department of Mechanical Engineering, KLE Dr. M. S. Sheshgiri College of Engineering and Technology, Belagavi 590008, India

[3] COMPARISION STUDY OF LOWER CONTROL ARM WITH DIFFERENT MATERIAL, by Gururaj Dhanu and Prof. R.S.Kattimani M.Tech (Machine Design), B.L.D.E. A's V. P. Dr.P.G.H College of Engineering and Technology, Vijayapur, Karnataka, India, Professor, Dept. of Mechanical Engineering, B.L.D.E. A's V. P. Dr.P.G.H College of Engineering and Tehnology, Vijayapur, Karnataka, India

[4] Structural optimization of control arm for weight reduction and improved performance by Mohd Viqaruddina, D.Ramana Reddy

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

[6] A new design optimization of light weight front lower control arm, 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, Journal of Advanced Manufacturing Technology (JAMT), ISSN: 1985-3157 Vol. 12 No. 1 January - June 2018.

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

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

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

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