International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 05 Issue: 09 | Sep 2018 www.irjet.net p-ISSN: 2395-0072

Durability Analysis and optimization of an Automobile Lower Suspension Arm Using FEA and Experiment Technique

Abhishek Mukund Puranik[1], Prof. V.M. Bansode[2], Prof. Sainand N. Jadhav[3] , Asst. Prof. Y.C. Jadhav[4]

[1]*PG Student, Department of Mechanical Engineering, NBN Sinhgad School of Engineering, Ambegaon (Bk), Pune-41, India*

[2]*Professor, Department of Mechanical Engineering, NBN Sinhgad School of Engineering, Ambegaon (Bk), Pune-41, India*

[3]*Professor, Department of Mechanical Engineering, NBN Sinhgad School of Engineering, Ambegaon (Bk), Pune-41, India*

[4]*Professor, Department of Mechanical Engineering, NBN Sinhgad School of Engineering, Ambegaon (Bk), Pune-41, India*

---***---

Abstract - *This paper deals with finite element analysis for optimization of lower suspension arm of 4W suspension system. The main function of the lower suspension arm is to manage the motion of the wheels & keep it relative to the body of the vehicle. The control arms hold the wheels to go up and down when hitting bumps. This paper describes the analysis of lower suspension arm to determine its stress behavior during its operation and scope for permissible topology optimization. The main objective is to have robust optimized less stressed lower suspension arm system. The optimization of lower suspension arm system is directly related to its strength or stiffness to with stand imposed force on the vehicle created by the road. Hence, Durability analyses and optimization is done in iterations, through Finite element method and computer aided engineering software's it is possible to come out with new final optimized design of lower suspension arm. The new design again is tested for initial loading conditions and responses of suspension system are confirmed for safe and fatigue life. Thereafter, best fitting optimized model is fabricated according to the new design of suspension system. Fabricated model is tested considering initial loading conditions using suitable experimental setup like universal testing machine, Fatigue testing machine and validated with FEA results.*

Key Words: **Lower Suspension Arm, Optimization Technique, Durability Analysis**

1.INTRODUCTION

Introduction about lower suspension arm

The performance of a vehicle is judged by driving comfort and safety, and both are provided by the suspension system. The suspension system is designed to carry the weight of vehicle and transmits al the forces between vehicle body and road. From a design point of view, there are two main types of disturbances on a vehicle namely the road and load disturbances. Therefore, the suspension system design is method of establishing compromise between these two disturbances. The Wishbone lower arm is a type of independent suspension used in motor vehicles. The general function of control arms is to keep the wheels of a motor vehicle on ground when the road conditions are not smooth. The control arm suspension consists of two arms, upper arm and lower arm. Based on the model and purpose of the vehicle, the upper and lower control arms have different structures. If we compare both the suspension arm, lower suspension arm is better shock absorber and upper suspension arm because of its position and load bearing capacities in the automotive industry. Independent suspension system gives the best rides, which permit the wheels to move independently of each other.

Figure 1: Automobile Lower Suspension Arm

There are various types of the suspensions like wishbone or double wishbone suspensions and Suspension arm is one of the main components in these suspension systems. Suspension arm joins the wheel hub to the vehicle frame allowing for a full range of motion while maintaining proper suspension alignment. The main causes of the failure of the lower suspension arm are uneven tire wear, suspension noise or misalignment, steering wheel shimmy or vibrations. Most of the cases the failures are catastrophic in nature. Therefore, the structural integrity of the suspension arm is crucial from design point of view both in static and dynamic conditions. Finite element method gives the exact visualisation for these kinds of failures Therefore, FEM analysis of the stress distributions around typical failure initiations sites are necessary. Therefore, in this study, static analysis and fatigue analysis of lower suspension arm needs to be carry out for light commercial vehicle using FEM.

Literature Survey

Miss. P. B. Patil and Prof. M. V. Kharade[1] in their research "Finite Element Analysis and Experimental Validation of Lower Control Arm" have analyze the lower suspension arm by FEA and experimental technique. The paper is referred for loading conditions & boundary conditions for the FEA of lower suspension arm

Dattatray Kothawale and Dr. Y. R. Kharde[2] in their work "Analysis of Lower Control Arm in Front Suspension System Using F.E.A. Approach" have used FEA for analysis of lower suspension arm considering al, the dynamic forces such as road bump, cornering, braking, accelerating, etc. The paper is referred to decide the dynamic forces acting on lower suspension arm the

Bhushan S. Chakor and Y.B.Choudhary[3] in their paper "Analysis and optimization of upper control arm of suspension system" analyze and optimize upper suspension arm using FEA and experimental technique. This paper is mainly referred for the methodology followed to carry out the study

Thomas D. Gillespie [4] in his book "Fundamentals of Vehicle Dynamics" have explained the study of vehicle dynamics. The book is referred to calculate the dynamic forces which are considered in FEA technique

Pratik S. Awati and Prof. L.M.Judulkar[5] in their research "Modal and Stress Analysis of Lower Wishbone Arm Along With Topology" aims to complete FEM analysis of a suspension link for bending vibrations, pitching, bouncing and combined mode dynamic analysis for deformation and stresses
Y. Nadota and V. Denierb[6] in their paper "Fatigue failure of suspension arm: experimental analysis and multiaxial criterion" have developed an experimental device to study fatigue phenomena for nodular cast iron automotive suspension arms. On the base of a detailed fracture analysis, it is shown that the major parameter influencing fatigue failure of casting components are casting defects.

M. M. Rahman, M. M. Noor, K. Kadirgama1, Rosli A. Bakar and M.R.M. Rejab[7] in their work "finite element modeling, analysis and fatigue life prediction of lower suspension arm" explores the finite element modeling, analysis and fatigue life prediction of lower suspension arm using the strain-life approach. Aluminum alloys are selected as a suspension arm material. The structural model of the suspension arm was utilizing the Solid works
N.A. Kadhim1, S. Abdullah, A.K. Ariffin and S.M. Beden[8] in their resear "Fatigue Failure Behaviour Study of Automotive Lower Suspension Arm" have studied fatigue life of automotive lower suspension arm under variable amplitude loadings. In simulation, the geometry of a sedan car lower suspension arm has been used
D. Taylor, P. Bologna and K. BelKnani[9] in their work "Prediction of fatigue failure location on a component using a critical distance method" have done the prediction of fatigue failure from notches and other stress concentrators is complicated by factors relating to the local notch geometry and stress field
Ashish Powar, Hrishikesh Joshi, Sanket Khuley and D.P. Yesane[10] in their paper, "Analysis and Topological Optimization of Motorcycle Swing-Arm" have stated that life cycles above 1e5 can be considered as infinite life cycles.

Problem Identification

During the car crash accidents, when the car is damaged from front side, it was seen that the lower suspension arm is the strongest and is just slightly damaged. Therefore, it can be said that the lower suspension arm carries over strength. From this observation the material optimization team from Tata Motors found the scope to optimize lower suspension arm. This optimization aims to reduce the unnecessary over strength of the arm. This work intended to design, modeling and analysis of car lower suspension arm to study and analysis of lower suspension arm to determine its stress behavior during its operation and scope for permissible topology optimization.

Objectives

The main objectives of this study to determine critical locations of stress and strain distributions of the lower suspension arm and to remove excess material from stress free region. The paper aims to complete Finite Element Analysis of the lower suspension arm which consist the analysis for stress, deformation and optimization.

Therefore, the existing model is to be analyzed by using FEA for fatigue life cycle. Also, the optimized final iteration design is to be checked for the fatigue life cycles. The values of life cycles of both the designed are to be compared. Finally, the final iteration of optimized design is to be fabricated. The fabricated model is to be checked for fatigue life cycles to validate the FEA results experimentally.

2.METHODOLOGY.

Since the study contains optimization of lower suspension arm, the methodology includes FEA as well as experimental analysis of the optimized design of lower suspension arm. This study has followed the below methodology to achieve required results.

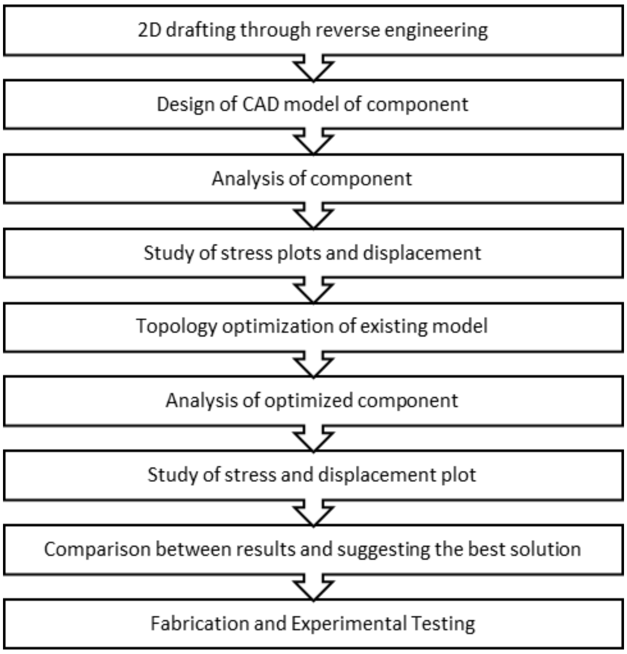


Figure 2 Methodology for implementation

3.FINITE ELEMENT ANALYSIS OF EXISTING LOWER SUSPENSION ARM

Reverse Engineering of existing lower Suspension Arm

Existing lower suspension arm was reverse engineered to prepare the CAD mode. Blue light scanning method is used for reverse engineering.

Figure 3 Lower suspension arm during blue light scanning

Points generated from blue light scanning is transferred to CatiaV5. Model is created using surfaces. The model looks as below in CatiaV5

Figure 4: Suspension arm in CATIA V5

Analytical force calculations

If we consider a moving vehicle, there are two major forces acting on the suspension arm. One is load due to road bump and other is braking load. These are the two cases which are majorly affects to the design of a lower suspension arm. Therefore, for the finite element analysis, we consider the load by these two cases.

Assumptions made during calculations:

1. 50:50 weight ratio is considered as standard assumption for ideal case
2. Average speed of car and bump height is considered
3. Suspensions is considered as a rigid component, that's why excluding damping effect.

Calculation of load

Loads on Transverse link:

- Road bump case
- Braking case

Car wheel designation: Indica

Kerb vehicle weight (GVW) = 995 kg

Therefore, weight on front side = 497 kg

(Assumption at ideal 50:50 weight ratio)

Weight on one side of wheel = 497/2 = 248.5 kg

Road bump case:

Let, Speed of vehicle = 14km/hr (3.8m/s)

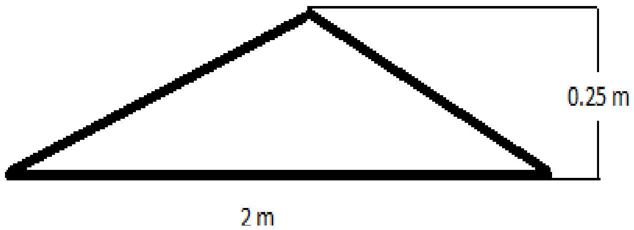


Figure 5: Road Bump

$$U = \frac{x}{t}$$

$$t = \frac{1}{3.8} = 0.25 \text{ m/sec}$$

$$U \text{ vertical} = \frac{x\text{vertical}}{\text{time}} = \frac{0.25}{0.25} = 1 \text{m/s}$$

$$A \text{ vertical} = \frac{U\text{vertical}}{\text{time}} = 4 \text{ m/sec2}$$

Wheel acceleration force (inertia force) = mass X acceleration

= 248.4 X 4

≈1000 N

Braking case:

Vehicle de accelerates (i.e braking) at a constant 0.5 G

Braking force = mass X acceleration X0.5 G (5)

= 248.4 X 9.81X 0.5

≈1250 N

Finite element analysis

Meshed Model: Following figure shows the meshed model of lower suspension arm. The meshing is done using Hypermesh 14. Number of nodes and elements formed after meshing are 9217 and 35848 respectively.

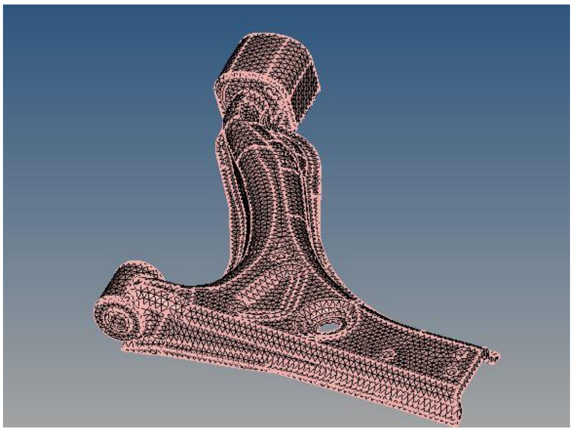


Figure 6: Meshed model of a lower suspension arm

Loading and Boundary Condition: Rigids are being formulated for the ease of application of boundary conditions. The rigids are concentrated on an independent node on which the forces are to be applied. The boundary conditions include braking and bump loads. The constraints are put on the mounting areas arresting all degrees of freedom as shown. In the below figure, two pivots shown by red colour are considered as mounting area. The direction and values of forces are shown in below figure.

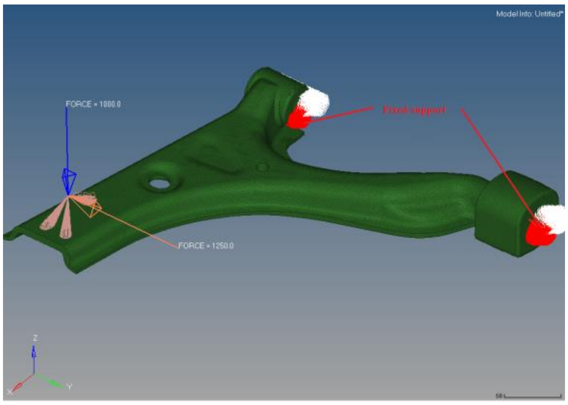


Figure 7: Constraint at the mounting location & Force applied

Table 1: Material properties of lower suspension arm

Property	Value
Young's Modulus, E	210 GPa
Poisson's Ratio ,ν	0.29
Density, ρ	7850 kg/m^3

Static Analysis Results

The reversed CAD model is analysed for static stresses and deformations using ANSYS 15.0. Results of the analysis are shown in the below figures.

a. Deformation for existing design:

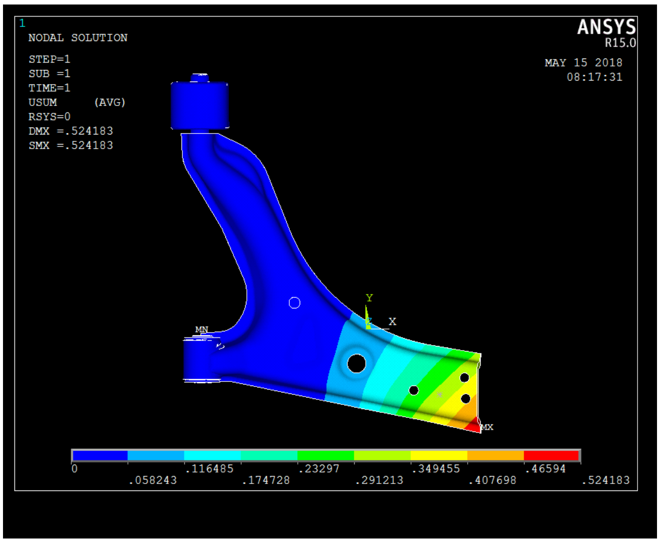


Figure 8: Displacement result for Existing Design

From above plot the maximum displacement value for existing design of lower suspension arm is 0.524 mm

b. Von Mises stress for existing design:

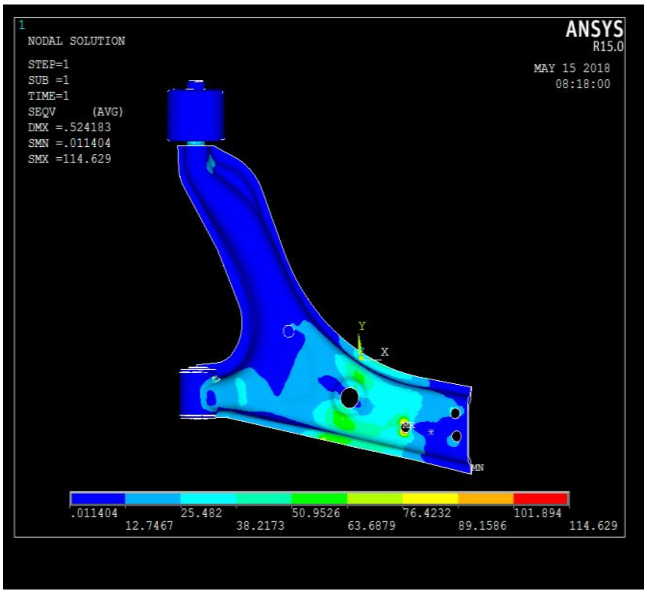


Figure 9: Von Mises stress for existing design

From above plot the maximum stress value for existing design lower control arm is 114.629 MPa which is less than yield strength; hence the design for lower suspension arm is safe.

Fatigue Life analysis for existing design

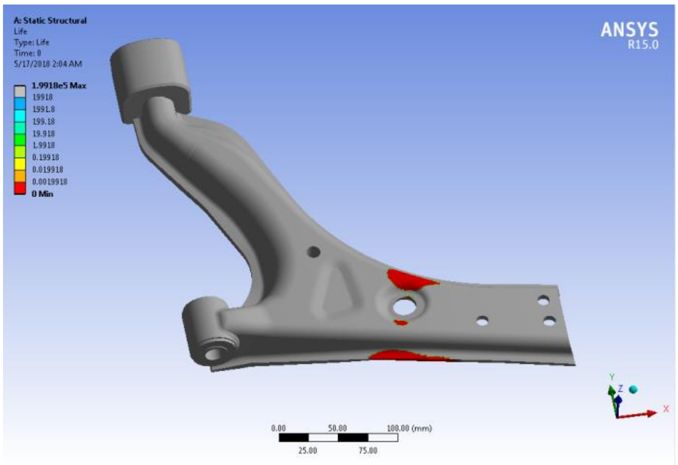


Figure 10: Fatigue life of Existing Design

From finite element analysis for fatigue, the no. of life cycles for existing design is 1,99,180 cycles. Therefore, the optimized design should have the no. of life cycles nearer to 199180 cycles.

Life of Existing Design: 1,99,180 cycles

4.OPTIMIZATION OF EXISTING MODEL

Topology optimization: Topology optimization is an optimization process in which it gives the optimum material layout according to the design space and loading case.

The weight reduction is done using Topology optimization by meeting the strength, safety factor targets. And the corresponding weight reduction is analysed.

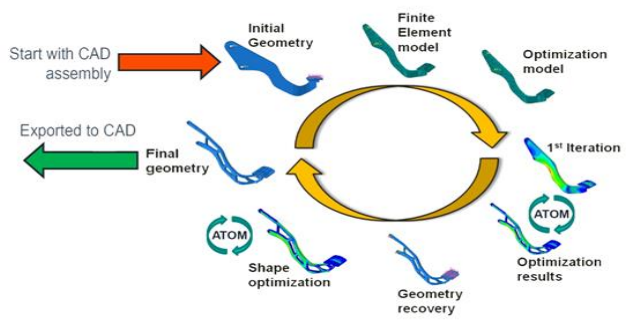


Figure 11: Loop of topology optimization

Topology optimization of existing Lower Suspension Arm:

After observing FEA results of existing Lower Suspension Arm and above discussed optimization techniques we can go for topology optimization to reduce weight, material and cost.

Iteration 1

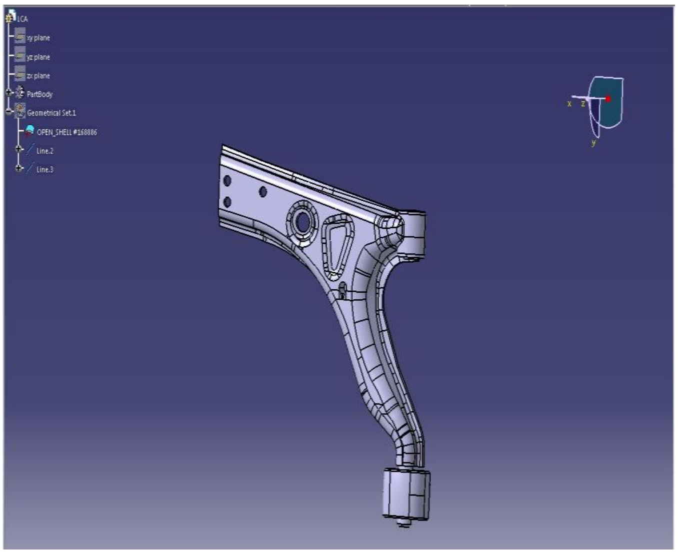


Figure 12: Optimized CAD model - Iteration-1

Fig: Optimized CAD model Iteration-1

From FEA results, the deformation is 0.549 mm, which is very low. Also, the stress induced is 119.56 MPa, which is well below the critical limit; hence the Lower Suspension Arm is safe in optimization Iteration-1. Therefore, for more weight reduction, the design is further optimized in Iteration 2.

Iteration 2

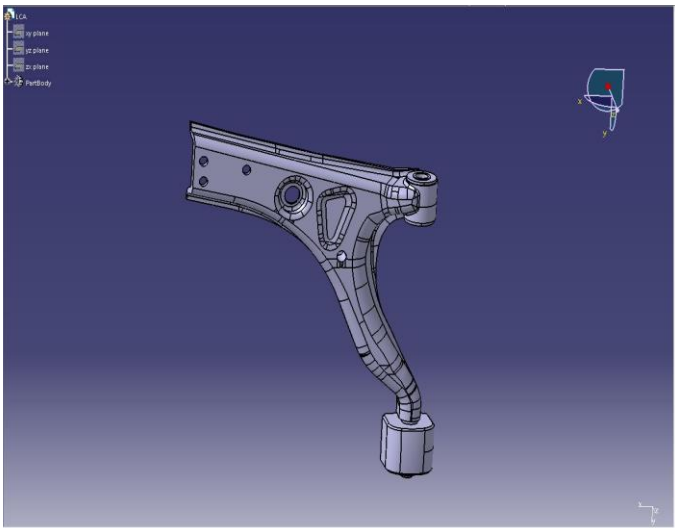


Figure 13: Optimized CAD model - Iteration-2

From FEA results, the deformation is 0.574 mm, which is very low. Also, the stress induced is 124.50 MPa, which is well below the critical limit; hence the Lower Suspension Arm is safe in optimization Iteration-2. Therefore, for more weight reduction, the design is further optimized in Iteration 3.

Iteration 3

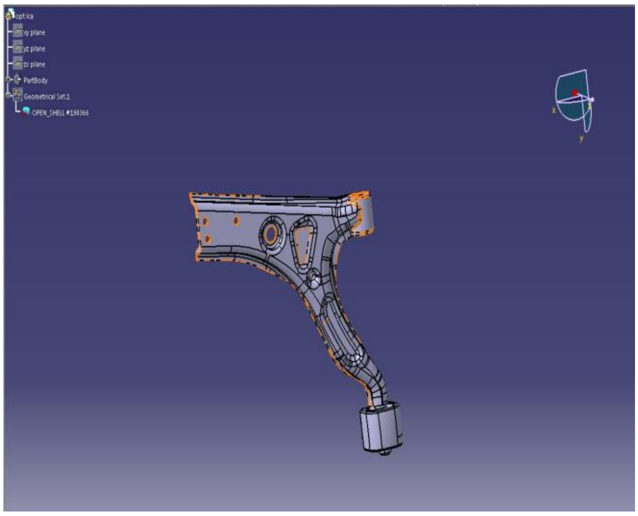


Figure 14: Optimized CAD model - Iteration-3

From FEA results, the deformation is 0. 543 mm, which is very low. Also, the stress induced is 136.035 MPa. Since the factor of safety considering this stress is about 1.5, we consider this as the final iteration of optimization. Hence, further optimization is not possible since, it may result in very low factor of safety.

5.FINITE ELEMENT ANALYSIS OF OPTIMIZED MODEL

As mentioned above, the Iteration 3 gives the optimum stress value and deformation, Iteration 3 of optimized design is selected.

Results for Stress and Deflection of Lower Suspension Arm Iteration-3:

Static Analysis Results for Optimized design:

Result for Deflection for optimized design:

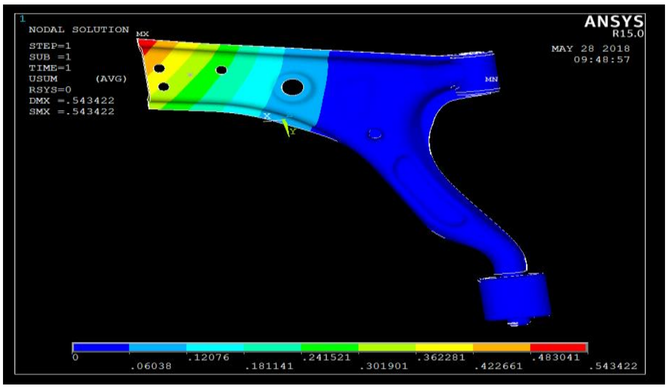


Figure 15: Displacement result for Iteration 3

From fig., the deformation of final iteration of optimized model is 0.543 mm the maximum deformation of optimized design and existing design are not varying by considerable amount.

a. Result of Stress for optimized design:

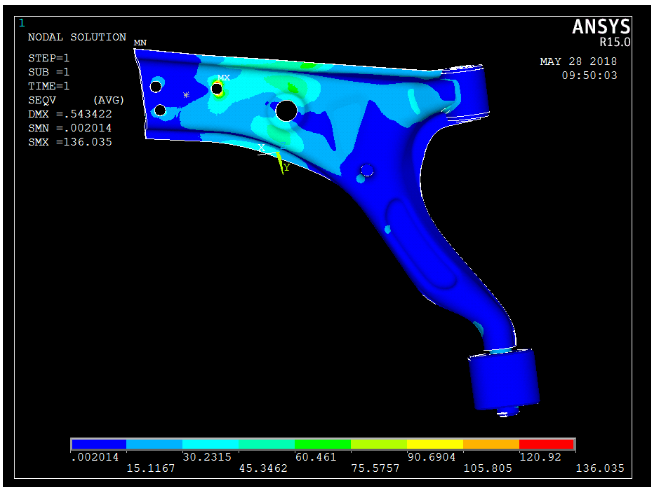


Figure 16: Von-mises stress for Iteration 3

As the stress induced is 136.035 MPa, which is well below the critical limit but amount of deformation is increasing gradually. Hence, keeping factor of safety in view Iteration-3 Lower Suspension Arm is considered final and safe optimization model

Results and discussions

Compression of structural analysis results

Table 2: Comparison of structural analysis

Material Steel	Existing	Optimized Iteration-1	Optimized Iteration-2	Optimized Iteration-3
Deformation (mm)	0.524 mm	0.549 mm	0.574 mm	0.543
Stress (Mpa)	114.62Mpa	119.56 Mpa	124.50 Mpa	136.03 Mpa
weight	6.97 Kg	6.75Kg (3% weight reduction)	6.66 Kg (4.4% weight reduction)	6.31 Kg (9.4% weight reduction)

Fatigue Life analysis for optimized design

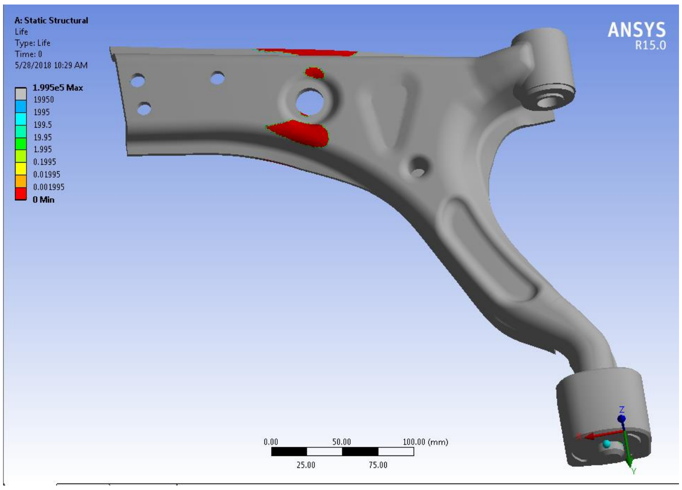


Figure 17: Fatigue life of Iteration 3

From finite element analysis for fatigue, the no. of life cycles for iteration 3 are 199090 cycles. The no. of cycles of optimized specimen and existing design are almost same. Therefore, it can be said that the optimized design is satisfying FEA results.

Life of optimized design: 1,99,090cycles

Comparison of Fatigue life:

Table 3: Comparison of Fatigue life

Fatigue Life	Maximum (cycles)
Existing steel Model	199180
Optimized 6 mm slot model (Iteration 3)	199090

6.EXPERIMENTAL ANALYSIS OF OPTIMIZED LOWER SUSPENSION ARM

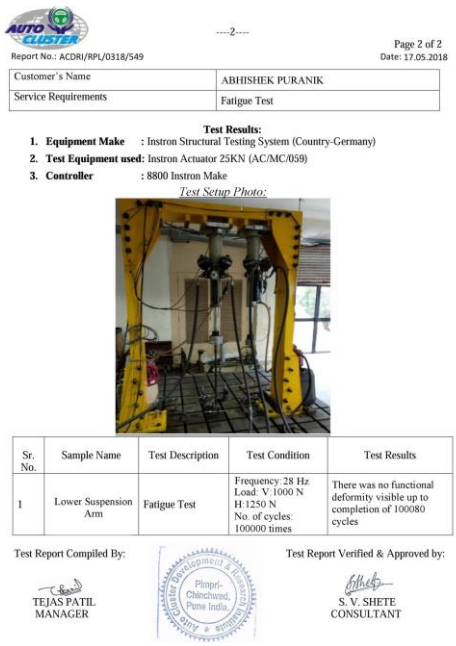
Manufacturing of optimized design of lower suspension arm

In fabrication existing lower suspension arm is considered and machined according to final topology optimization suggested through finite element analysis. Machining process involves markings in existing lower suspension arm according to new design drawings followed by cutting, grinding, welding and finishing processes. Finally machined optimized model is considered for further testing process.

Figure 18: Final machined optimized model

Experimental Validation of optimized

For the experimental validation of optimized lower suspension arm, we have used Instron Actuator. Road bump load of 1000N is applied vertically on the arm and Braking load of 1250N is applied horizontally on arm. Both the loads are applied on wheel hub side connection of lower suspension arm. The certificate and the result of the experimental validation is provided in below image.



----2----

Report No.: ACDRI/RPL/0318/549

Page 2 of 2
Date: 17.05.2018

Customer's Name	ABHISHEK PURANIK
Service Requirements	Fatigue Test

Test Results:

1. **Equipment Make** : Instron Structural Testing System (Country-Germany)
2. **Test Equipment used:** Instron Actuator 25KN (AC/MC/059)
3. **Controller** : 8800 Instron Make

Test Setup Photo:

Sr. No.	Sample Name	Test Description	Test Condition	Test Results
1	Lower Suspension Arm	Fatigue Test	Frequency: 28 Hz Load: V:1000 N H:1250 N No. of cycles: 100000 times	There was no functional deformity visible up to completion of 100080 cycles

Test Report Compiled By:
TEJAS PATIL
MANAGER

Auto Cluster Development & Research Institute
Pimpri-Chinchwad, Pune India.

Test Report Verified & Approved by:
S. V. SHETE
CONSULTANT

Figure 19: Certificate of fatigue testing results of optimized model

In fatigue testing optimized Lower Suspension Arm is subjected to repeated loading case and ran for stipulated cycles and with respective frequency. In fatigue analyses if a component successfully completes one lakh cycles the component is considered as having infinite life. From the above testing the component is ran for one lakh cycles in one hour with frequency of 28 Hz and component was stable and safe even after one lakh cycles and passed fatigue life

7.RESULTS AND DISCUSSION

When the above loads are applied on the specimen, there was no crack initiation till cyacle100080. And we always consider that above 1 lakh cycle the component under loading condition is considered as infinite number of cycle.

The difference between weight of existing and optimize lower control arm along with the fatigue life cycle of existing and optimized designed is as shown below table. Below table shows the comparison and results from FEA and Experimental techniques.

Table 4: Results and discussions

Component	Weight	Analysis method	Fatigue Life
Existing Component	6.97 Kg	FEA Results	199180
Optimized Designed Component	6.31 Kg (9.4% weight reduction)	FEA Results	199090
		Experimental Testing Results	100080 (Infinite)

8.Conclusions

- The reversed CAD modeling and analysis for static and fatigue life cycle of existing lower suspension arm showed possibility of optimization as the stresses and displacement induced were well within the prescribed limits.
- Therefore, topology optimization is carried out of the existing lower suspension arm at less stress concentrated area. As per the FEA analysis the lower suspension arm is being optimized by iterative method and thus optimizing the weight of the component.
- The final iteration of topology optimization shows 0.543mm of maximum displacement and 136.03MPa of maximum stress which is having 1.54 FOS.
- The total weight reduction in final iteration of optimization is 9.4% of the total lower suspension arm weight.
- The optimized model is then fabricated and validated for fatigue life testing by experimental method

REFERENCES

1. Miss. P. B. Patil and Prof. M. V. Kharade "Finite Element Analysis and Experimental Validation of Lower Control Arm" 2016 IJEDR | Volume 4, Issue 2 | ISSN: 2321-9939
2. Dattatray Kothawale and 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 e-ISSN: 2278-067X, p-ISSN: 2278-800X, Volume 5, Issue 12 (February 2013), PP. 18-23
3. Bhushan S. Chakor and Y.B.Choudhary "Analysis and optimization of upper control arm of suspension system" International Journal For Technological Research In Engineering Volume 5, Issue 1, September-2017 ISSN (Online): 2347 - 4718
4. Thomas D. Gillespie "Fundamentals of Vehicle Dynamics" Published by Society of Automotive Engineers, Inc
5. Pratik S. Awati, Prof. L.M.Judulkar, "Modal and Stress Analysis of Lower Wishbone Arm Along With Topology", International Journal of Application or Innovation in Engineering & Management (IJAIEM) Volume 3, Issue 5, May 2014 ISSN 2319 - 4847
6. Y. Nadota, V. Denierb, "Fatigue failure of suspension arm: experimental analysis and multiaxial criterion", Elsevier, Engineering Failure Analysis 11 (2004) 485–499
7. M. M. Rahman1, M. M. Noor1, K. Kadirgama1, Rosli A. Bakar1, M.R.M. Rejab, "finite element modeling, analysis and fatigue life prediction of lower suspension arm, Malaysian Technical Universities Conference on Engineering and Technology June 20-22, 2009
8. N.A. Kadhim1, S. Abdullah2, A.K. Ariffin and S.M. Beden, "Fatigue Failure Behaviour Study of Automotive Lower Suspension Arm", Key Engineering Materials Vols. 462-463 (2011) pp 796-800
9. D. Taylor a, P. Bologna b, K. BelKnani b, "Prediction of fatigue failure location on a component using a critical distance method", Elsevier, International Journal of Fatigue 22 (2000) 735–742
10. Ashish Powar, Hrishikesh Joshi, Sanket Khuley and D.P. Yesane "Analysis and Topological Optimization of Motorcycle Swing-Arm" International Journal of Current Engineering and Technology E-ISSN 2277 – 4106, P-ISSN 2347 – 5161