Experimental and FEA of Optimized Existing Lower Control Arm

Sudarshan Bhalshankar and Amol Ubale

March 9, 2021

Experimental and FEA of optimized existing lower control arm

Sudarshan Dilip Bhalshankar.
Mechanical Engineering, CAD/CAM.
Zeal College of Engineering & Research.
Pune, Maharashtra, India.
Sudarshanbhalshankar111@gmail.com

Dr. Amol Ubale
Associate professor and Dean Students, Mechanical Engineering
Zeal College of Engineering & Research.
Pune, Maharashtra, India.
amol.ubale@zealeducation.com

Abstract : The lower control arm is an interesting kind of autonomous suspension utilized in four wheel vehicles. During the genuine working condition, the most extreme load is moved from upper arm to the lower control arm which makes plausibility of failure in the arm. Henceforth it is fundamental to concentrate on the stress investigation of lower control arm to improve and alter the current design. A lower control arm is a significant part utilized in a suspension arrangement of a vehicle. So, this arm execution a significant job in dealing with the movement of the wheel during knock, turning, and breaking. In present research design of lower control arm is done in CATIA software. ANSYS19 software was also used for analyze the structural strength and optimize the parts weight along with modal analysis to determine natural frequency with mode shape and validate the results with FFT technique. The target of the new design was 8% weight reduction from the existing part fabricated using steel material. Testing and validation of new design using FFT analyzer is done and it is found that average percentage error in experimental and theoretical analysis of natural frequency is around 0.92%.

Keywords—LCA, FEA, Modal Analysis, Impact Hammer Test

I. INTRODUCTION

Electric-versatility, CO_2 emanation limits, fuel, an Earth-wide temperature boost and vitality costs are a portion of the variables driving lightweight car structure. Lightweight structure requires appropriate, financial assembling advances notwithstanding the utilization of lightweight materials. Thus, it is a test to car producer to create the lightweight vehicle without trading off their presentation. Weight decrease empowers the producer to build up a similar vehicle execution with a littler motor, and such a littler motor empowers the utilization of a littler transmission and fuel tank. With these expanding influences, it is assessed that 10% of vehicle weight decrease brings about 8–10% of mileage improvement. The suspension framework caries the vehicle body and transmit all powers between the body and the street without transmitting to the driver and travellers. The suspension arrangement of a vehicle is utilized to help its weight during fluctuating street conditions. The suspension framework is made of a few sections and parts. These incorporate the front and back. In the car business, dealing with characteristics of vehicle is a significant issue. These characteristics are extraordinarily influenced by the suspension framework. The suspended segment of the vehicle is connected to the wheels. To pad the effect of street inconsistencies suspension arm is associated. Suspension arm is the principle part in car suspension framework. It conveys all the various burdens made because of sporadic streets. There are different kinds of suspensions like wishbone or twofold wishbone suspensions. There are loads of research works which comprise of suspension framework, various kinds of suspension framework, upper and lower control arm. The lower control arm is exposed to numerous heaps because of variety in net weight and effects because of vacillation of street surface and extra powers. The un-sprung mass is the mass of the suspension segments which is legitimately associated with them, instead of upheld by the suspension. High un-sprung weight intensifies issues like wheel control, ride quality and commotion. Un-sprung weight incorporates the mass of parts, for example, the wheel axles, wheel course, wheel centre points, springs, safeguards, and Lower Control Arm. The lower control arm gets more consideration by numerous explores like examination dynamic investigations of the engine vehicle suspension framework utilizing the point-joint facilitates detailing.

Song and et al.(2010) suggested surrogate models which were Simulation-based used for a different applications of automotive industry. They have done the FEM analysis; both of the Kriging model and response surface model (RSM) were used for optimization of upper control arm. The weight of the upper control arm was considered as the design objective, with the allowable maximum von Mises stress as the constraint objective. The optimization results were obtained by using RSM and KRG were confirmed by FEM analysis. The authors also carried out fatigue analysis for verification of the final design durability. Whereas Dattatray Kothawale and et al.(2013) have done finite element analysis for MacPherson type suspension system A Lower control arm (LCA) of 4Wheel vehicle. The main function of the A control arm was to manage the motion of the wheels & keep it relative to the body of the vehicle. The CAD Model was prepared by using PRO-E Software & finite element analysis by using Ansys software. Various dynamic loads like road bump, kerb strike, braking, cornering & acceleration load case were studied in dtail. By applying all this forces in X, Y and Z directions non-linear static analysis was performed using Ansys software [4]. Balasaheb Gadade and et al.(2015)carried out the work which was mainly focused on the finite element based stress analysis of A – Type A suspension arm. The main objective of this study was to calculate working life of the component under static loading. Actual model was manufacture as per Design by using AISI 1040 material. The finite element modeling and analysis were performed by using HYPERMESH software. A simple design approach was used to calculate effect of stresses on A – Type A suspension arm element under static loading condition. The experimental work included validation of the FEA results with actual testing of the model under stress [6]. Further Sridharan and et al.(20016) worked on modeling and performing structural analysis of a Lower Control Arm (LCA) used in the front suspension system, made up of sheet metal. LCA was modeled in Pro-E software for the given specification. For the analysis of the LCA, Computer Aided Engineering (CAE) software was used. Dynamic load acting on the lower was considered for study purpose; also buckling load analysis was done. First finite element analysis was performed to calculate the buckling strength, of a control arm. After getting the final result of finite element analysis optimization has been done by using design of experiment method (DOE). Taguchi's DOE was used to optimize the number of experiments. By reducing thickness of the sheet metal and by suggesting the suitable material the production cost could be reduced. Hence it resulted in cost saving and improved material quality of the product.[3] Viqaruddina and et al.(2017) have designed the system by topology optimization for compare the base run analysis and optimized analysis. Meshing was carried out by using 10 nodes tetrahedral element in Hyper Mesh & topology optimization was 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 [2]. Also Seifried and et al.(2018) have aimed to reduce the mass of flexible members without deteriorating the accuracy of the system. The structural optimization based on topology optimization of members of flexible multi-body system was introduced and the effects of uncertainty in the optimization process were investigated. Two sources of uncertainty, namely the model uncertainty and the un-certainty in usage were addressed. As an application example, a two-arm manipulator was used to examine and illustrate the effects of uncertainties such as different objective functions, choices of model reduction method as well as changes in the trajectory and payload of the manipulator [1]. Whereas Gunjan and et al.(2018) have quoted that the structural integrity of the suspension arm was crucial from design point of view both in dynamic as well as static loading conditions. Hence the authors presented modeling and analysis of car front suspension lower arm for studying the stress condition and to select the suitable materials for the front suspension lower arm. The main objectives of their study were to determine critical locations and strain distributions of the component. The main aim was to complete FEA of the front suspension lower arm which consist the stress optimization loadings and analysis for deformation.[5]

II. PROBLEM STATEMENT

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. The design graphic display device and method, and more particularly, to a computer-aided structure analysis of A control arm is required to be done in detail. Further analysis and designing is required. It is observed that lot of research is done on for design of the A control arm of suspension system but still scope is available for the optimization to meet the customer requirements of Lower Control Arm (LCA).

This work consists of dynamic analysis of an arm using harmonic excitation for investigation dynamic behaviors. The designing is done with computer aided design software and theoretical analysis and then actual experimentation is carried out to validate the theoretical analysis.

III. OBJECTIVES

1. To determine the problem associated due to vibration on lower control arm and preparation of cad model of existing lower control arm
2. Modeling and analysis of lower control arm for static and modal analysis using ANSYS software.
3. To perform topology optimization for weight reduction using optimization module in ANSYS to obtain optimized design.
4. To perform harmonic analysis to determine frequency response for existing and optimized design.
5. Comparison of experimental and FEA results.

V. METHODOLOGY

Step 1:- Exhaustive literature survey to study existing work and to find research gap for project work with necessary parameters which are studied in detail.
Step 2:- Finding the gap in the previous research to define the problem statement to carry out research work.
Step 3: - After deciding the components, the 3 D Model and drafting is done with the help of CAD software.
Step 4: - According to the theoretical analysis actual component is manufactured and then assembled together.
Step 5: -The experimental testing is carried out on prototype with the help of FFT analyzer and results are compared.

CATIA MODEL OF EXITING LOWER CONTROL ARM

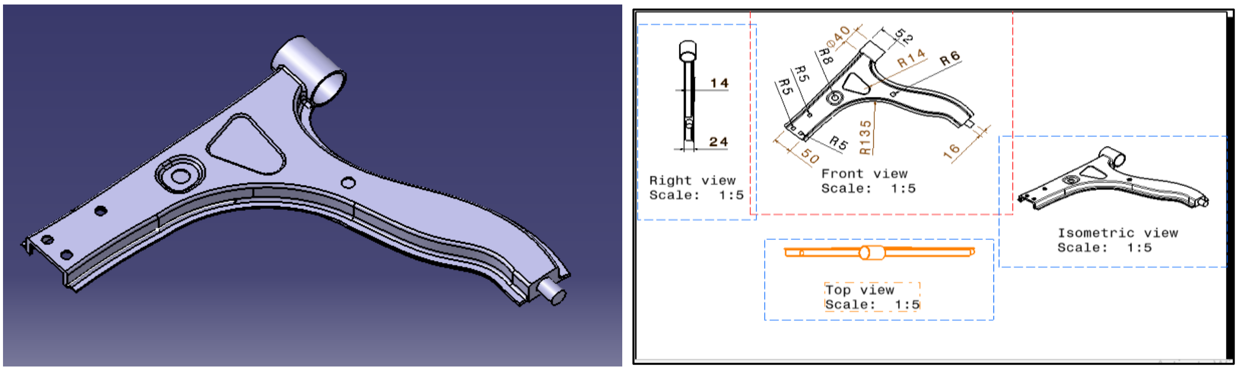
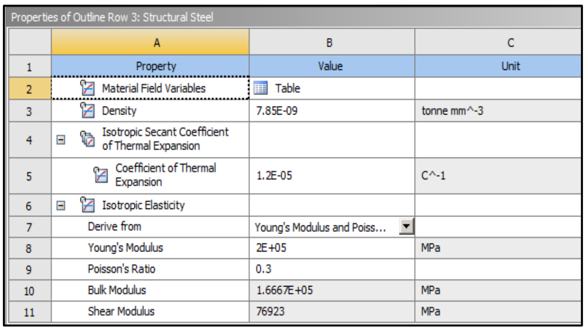


Fig. 1CATIA and drafting of lower control arm

Table1. 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

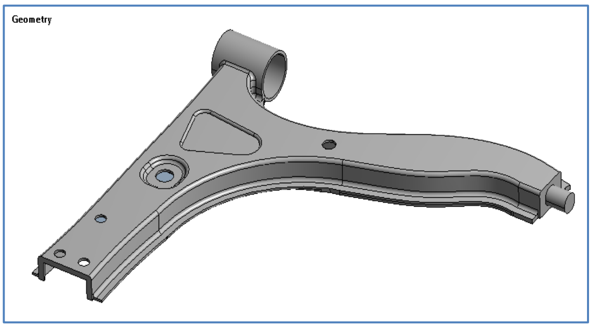
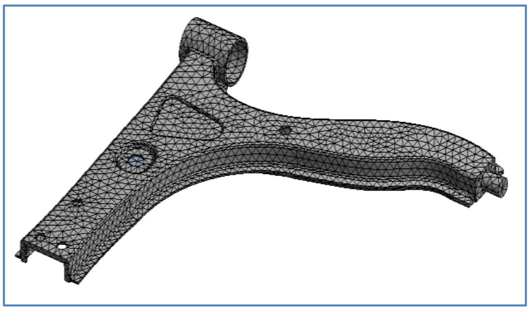
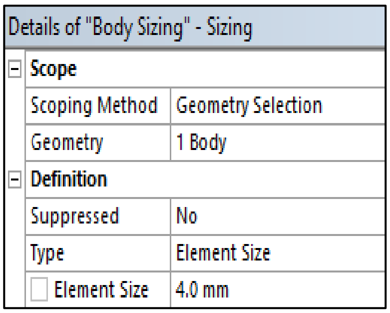


Fig.2 CATIA model imported in ANSYS

Meshing the cad model

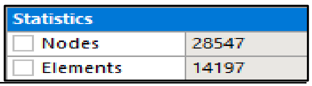
In ANSYS meshing is performed as similar to dicretization process in FEA procedure, in which it breaks whole components in small elements and nodes. So, in analysis boundary condition equation are solved at this elements and nodes. ANSYS Meshing is a general-purpose, intelligent, automated high-performance product. It produces the most appropriate mesh for accurate, efficient Multi-physics solutions. A mesh well suited for a specific analysis can be generated with a single mouse click for all parts in a model. Full controls over the options used to generate the mesh are available for the expert user who wants to fine-tune it.





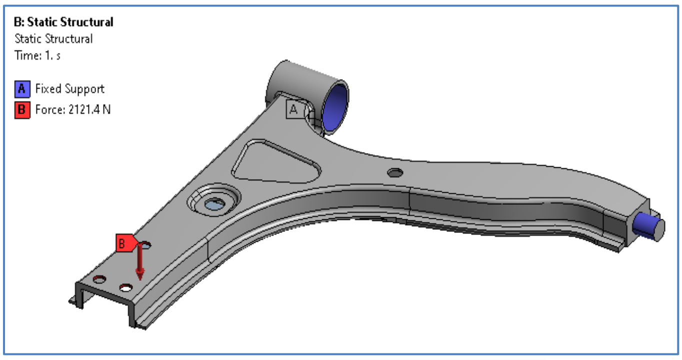
Details of "Body Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Type	Element Size
☐ Element Size	4.0 mm



Statistics	
☐ Nodes	28547
☐ Elements	14197

Fig.3 Details of meshing of lower control arm

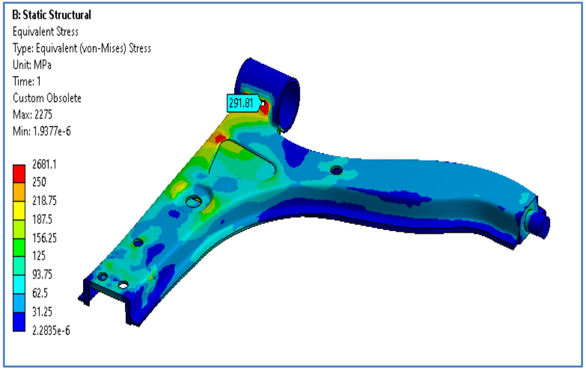


B: Static Structural
Static Structural
Time: 1. s

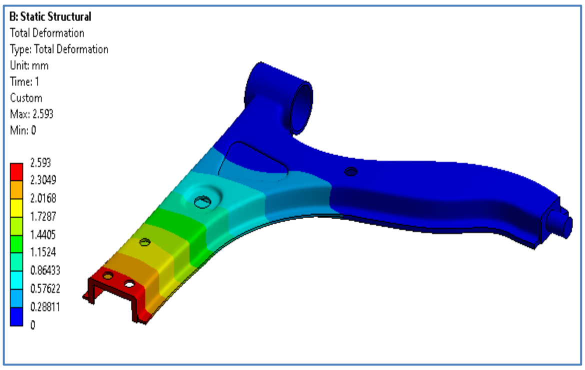
VEHICLE NAME- MARUTI SUZUKI BALENO
WEIGHT – 865 KG
SO 865 X 9.81 – 8485 N
WEIGHT ON EACH WHEEL IS CONSIDERED AS 8485/4 – 2121.4 N

Fig.4 Applying boundary condition

In present FEA analysis fixed support is applied as per existing condition along with force as per vehicle specified.



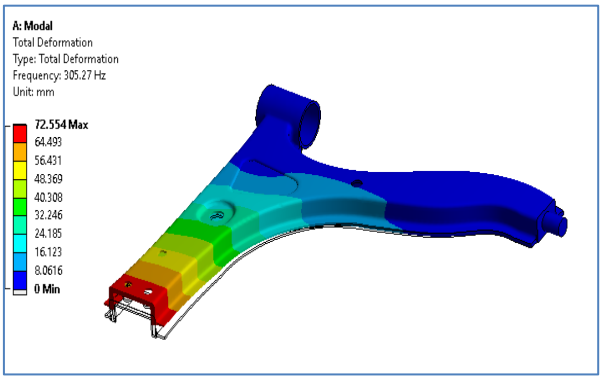
B: Static Structural
Equivalent Stress
Type: Equivalent (von-Mises) Stress
Unit: MPa
Time: 1
Custom Obsolete
Max: 2275
Min: 1.9377e-6



B: Static Structural
Total Deformation
Type: Total Deformation
Unit: mm
Time: 1
Custom
Max: 2.593
Min: 0

Fig.5 Equivalent stress results **Fig.6** Total deformation results

MODAL ANALYSIS OF EXISTING SUBFRAME



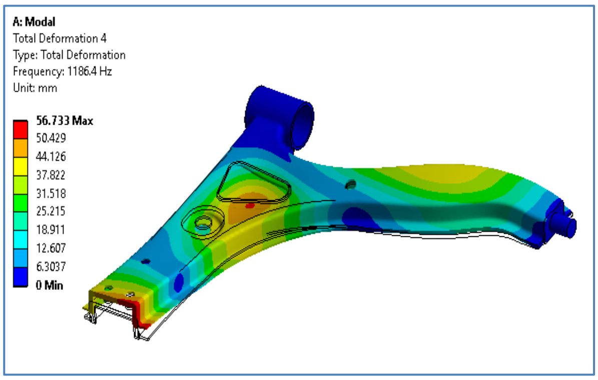
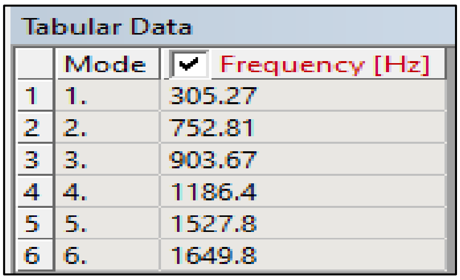


Fig.7 Mode shape 1

Fig.8 Mode shape 4

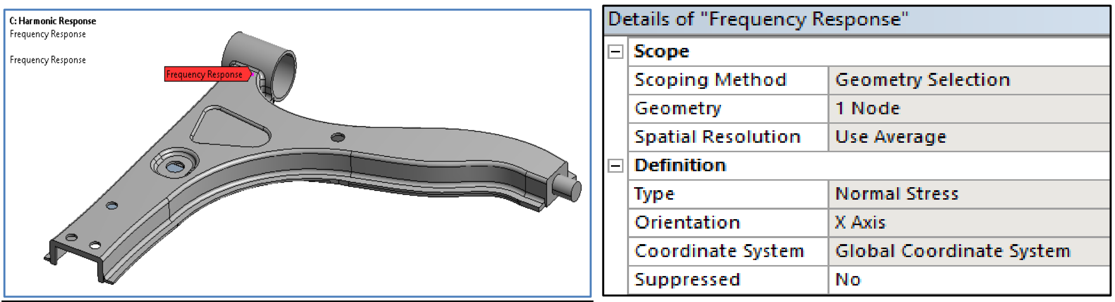
Table2. Tabular data of mode shape frequency



Tabular Data

	Mode	☑ Frequency [Hz]
1	1.	305.27
2	2.	752.81
3	3.	903.67
4	4.	1186.4
5	5.	1527.8
6	6.	1649.8

Harmonic analysis of existing lower control arm



Details of "Frequency Response"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Node
Spatial Resolution	Use Average
Definition	
Type	Normal Stress
Orientation	X Axis
Coordinate System	Global Coordinate System
Suppressed	No

Fig.9 Boundary condition for harmonic frequency response

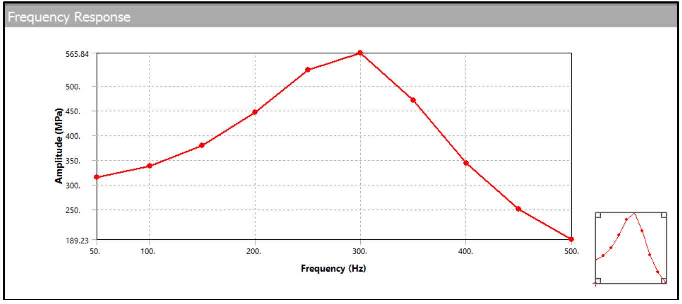
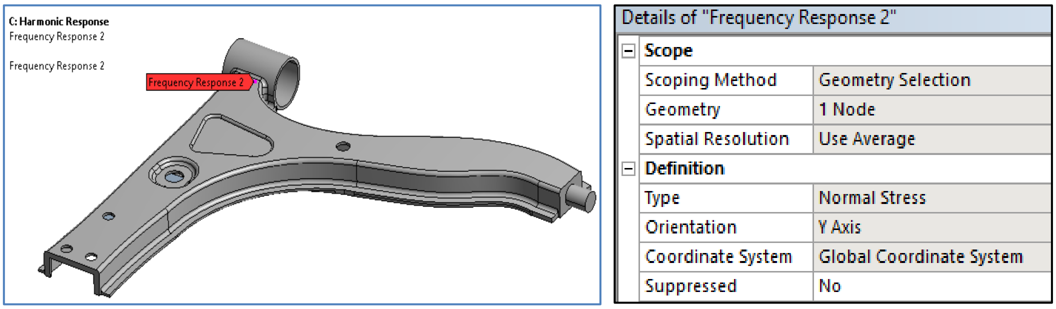


Fig.10 frequency response result along x axis



Details of "Frequency Response 2"

⊟ **Scope**	
Scoping Method	Geometry Selection
Geometry	1 Node
Spatial Resolution	Use Average
⊟ **Definition**	
Type	Normal Stress
Orientation	Y Axis
Coordinate System	Global Coordinate System
Suppressed	No

Fig.11 Boundary condition for harmonic frequency response

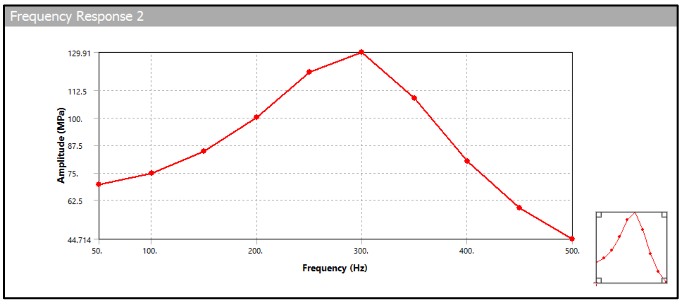


Fig.12 frequency response result along y axis

TOPOLOGY OPTIMIZATION

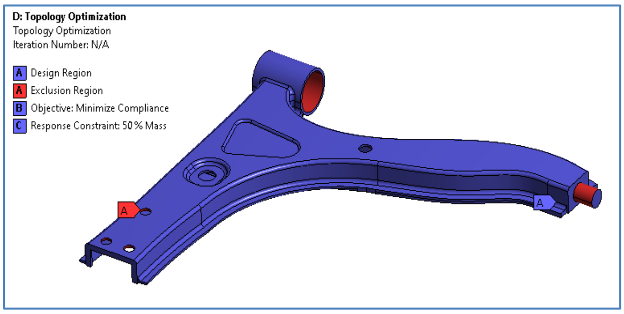


Fig.13 Boundary condition for topology optimization region

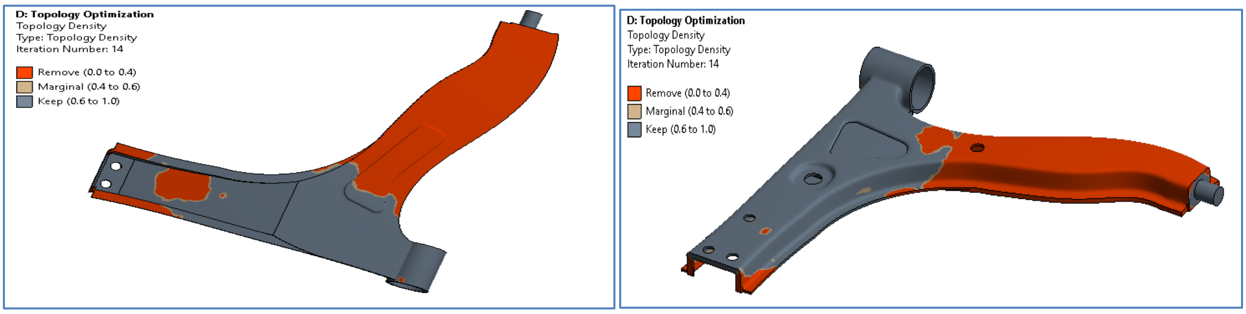


Fig.14 Topology optimized results

- Red region indicates material removal area region.

MODIFIED LOWER CONTROL ARM

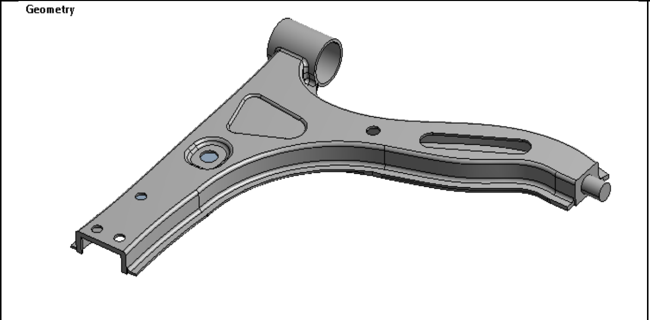


Fig.15 CATIA of modified LCA

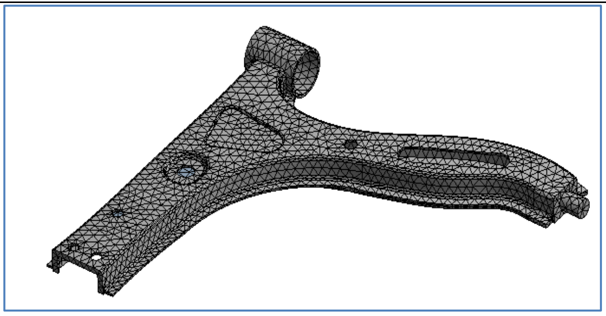


Fig.16 Details of meshing

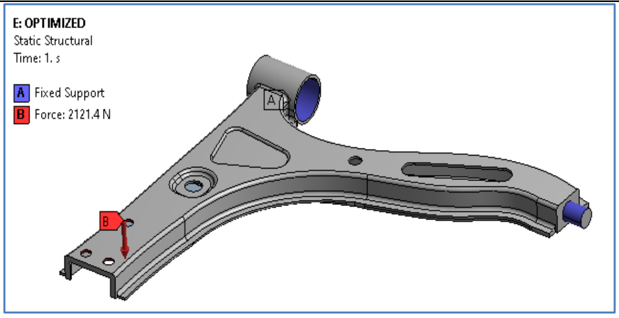


Fig.17 Boundary condition for modified LCA

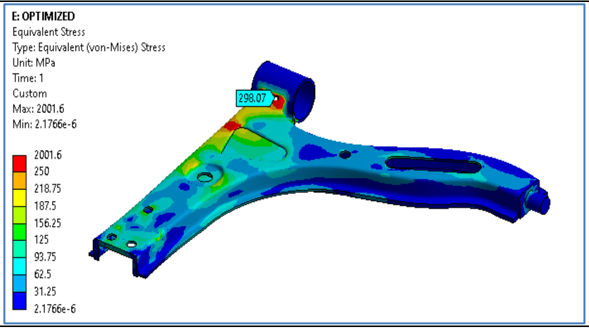


Fig.18 Equivalent stress results

With the modified design of the lower arm again analysis is carried out by applying the similar loading boundary conditions as applied for the existing system and results are checked.

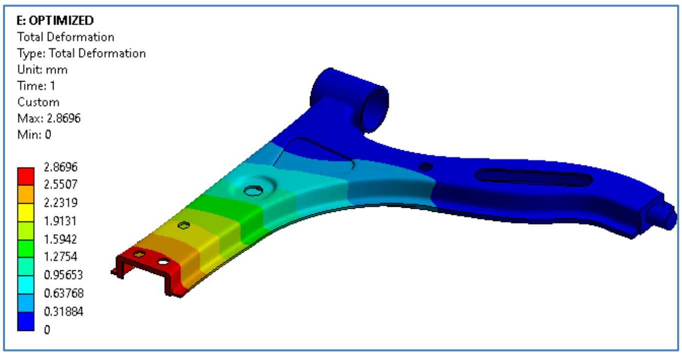
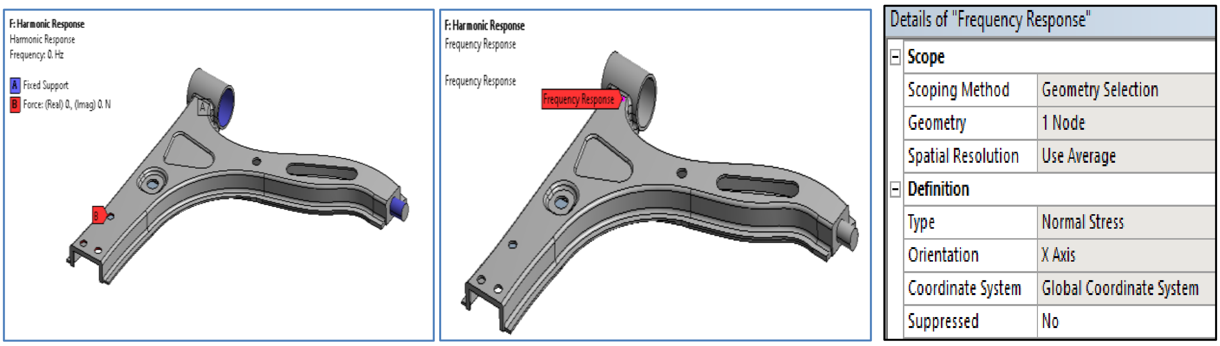


Fig.19 Total deformation results

Harmonic response for optimized lower control arm



Details of "Frequency Response"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Node
Spatial Resolution	Use Average
Definition	
Type	Normal Stress
Orientation	X Axis
Coordinate System	Global Coordinate System
Suppressed	No

Fig.20 Boundary condition for Optimized lower control arm for harmonic response surface along x axis

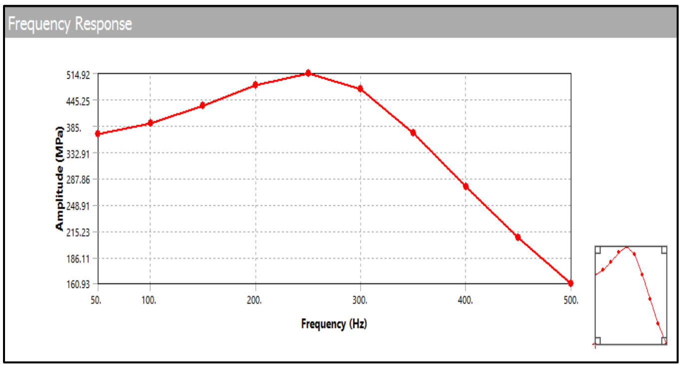
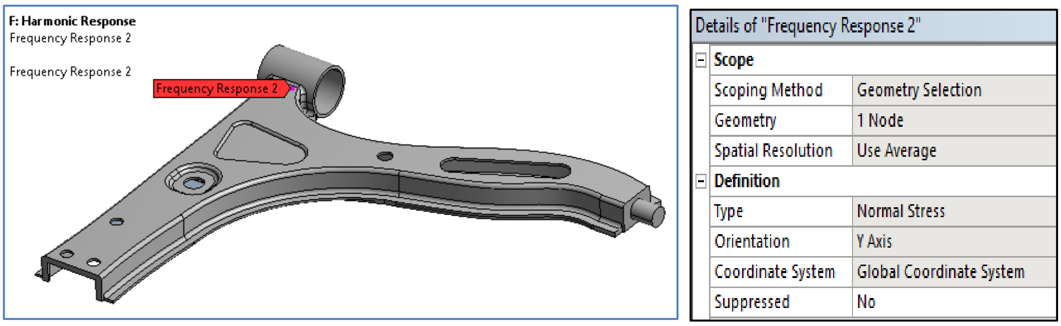


Fig.21 Frequency response result for optimized lower control arm along **x** axis



Details of "Frequency Response 2"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Node
Spatial Resolution	Use Average
Definition	
Type	Normal Stress
Orientation	Y Axis
Coordinate System	Global Coordinate System
Suppressed	No

Fig.22 Boundary condition for Optimized lower control arm for harmonic response surface along **y** axis

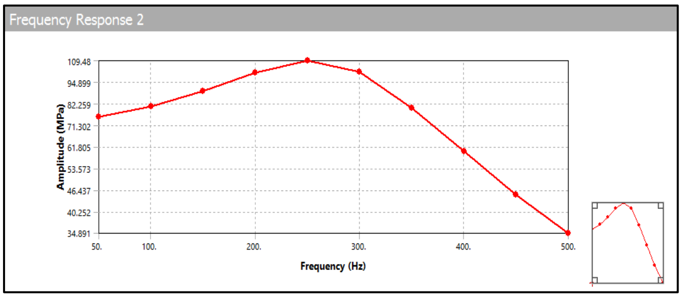
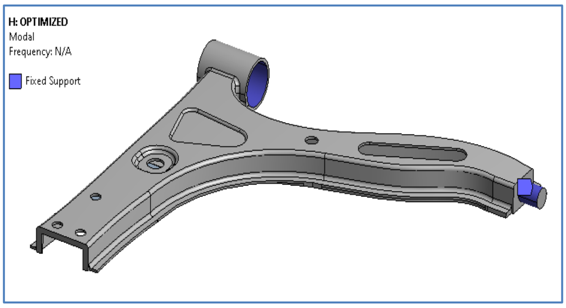
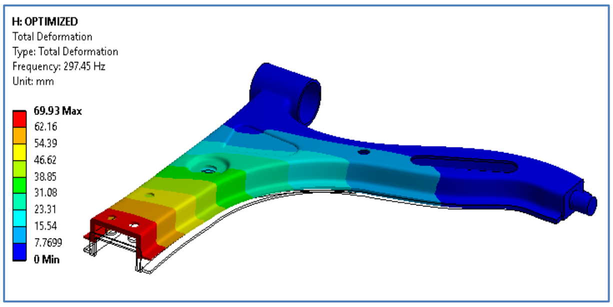


Fig.23 Frequency response result for optimized lower control arm along **y** axis

It is observed from harmonic analysis that amplitude along **x** and **y** axis in optimized design has decreased by 4% as compared to original design.

Modal analysis of optimized lower control arm





Total Deformation
Type: Total Deformation
Frequency: 297.45 Hz
Unit: mm

Fig.24 Boundary condition for modal analysis

Fig.25 Mode shape 1

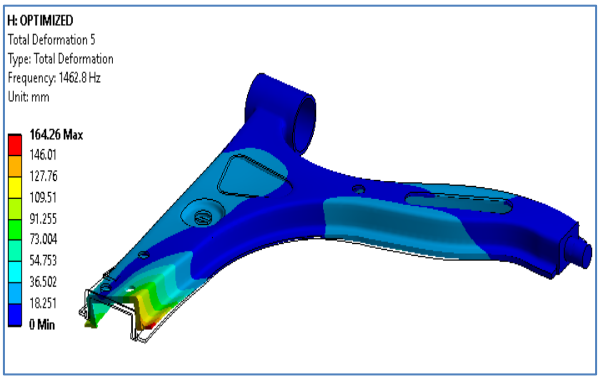
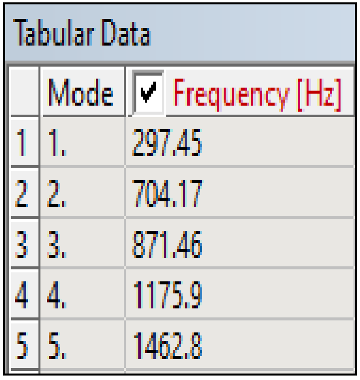


Fig.26 Mode shape 5

Table 3 . Tabular data of natural frequency with respective mode shapes


Tabular Data

	Mode	☑ Frequency [Hz]
1	1.	297.45
2	2.	704.17
3	3.	871.46
4	4.	1175.9
5	5.	1462.8

EXPERIMENTAL TESTING

Fast Fourier Transform

FFTs were first discussed by Cooley and Tukey (1965), although Gauss had actually described the critical factorization step as early as 1805 (Bergland 1969, Strang 1993). A discrete Fourier transform can be computed using an FFT by means of the Danielson-Lanczos lemma if the number of points N is a power of two. If the number of points N is not a power of two, a transform can be performed on sets of points corresponding to the prime factors of N which is slightly degraded in speed. An efficient real Fourier transform algorithm or a fast Hartley transform (Bracewell 1999) gives a further increase in speed by approximately a factor of two. Base-4 and base-8 fast Fourier transforms use optimized code, and can be 20-30% faster than base-2 fast Fourier transforms. Prime factorization is slow when the factors are large, but discrete Fourier transforms can be made fast for N=2, 3, 4, 5, 7, 8, 11, 13, and 16 using the Winograd transform algorithm.

The experimental validation is done by using FFT (Fast Fourier Transform) analyzer. The FFT spectrum analyzer samples the input signal, computes the magnitude of its sine and cosine components, and displays the spectrum of these measured frequency components. The advantage of this technique is its speed. Because FFT spectrum analyzers measure all frequency components at the same time, the technique offers the possibility of being hundreds of times faster than traditional analog spectrum analyzers.

Fourier analysis of a periodic function refers to the extraction of the series of sines and cosines which when superimposed will reproduce the function. This analysis can be expressed as a Fourier series. The fast Fourier transform is a mathematical method for transforming a function of time into a function of frequency. Sometimes it is described as transforming from the time domain to the frequency domain. It is very useful for analysis of time-dependent phenomena.

Impact Hammer Test

Impact excitation is one of the most common methods used for experimental modal testing. Hammer impacts produce a broad banded excitation signal ideal for modal testing with a minimal amount of equipment and set up. Furthermore, it is versatile, mobile and produces reliable results. Although it has limitations with respect to precise positioning and force level control, overall its advantages greatly outweigh its disadvantages making it extremely attractive and effective for many modal testing situations.

The use of impulse testing with FFT signal processing methods presents data acquisition conditions which must be considered to ensure that accurate spectral functions are estimated. Problems stem from the availability of only a finite duration sample of the input and output signals. When a structure is lightly damped the response to the hammer impact may be sufficiently long that it is impractical to capture the entire signal. The truncation effect manifests itself in terms of a spectral bias error having the potential to adversely affect the estimated spectra. The signal truncation problem is further compounded in practice by the computational and hardware constraints of the FFT processing equipment. Typically the equipment has a limited number of data capture lengths or frequency ranges which are available for an operator to select. Normally a user is more concerned with useable analysis frequencies and less with the data capture length. Therefore, it is conceivable that an inappropriate data capture duration could be used which truncates the vibration signal and introduces errors in the estimated spectra. To suppress the truncation a common practice is to artificially force: it to decay within the data capture window [1,2,3]. This artificial reduction is obtained by multiplying the slowly decaying vibration signal by an exponential function. However, the application of the exponential window must be considered carefully since it may also adversely affect the estimated spectra.

A phenomenon commonly encountered during impact testing is the so called "double hit". The "double hit" applies two impulses to the structure, one initially and one time delayed. Both the temporal and spectral characteristics of the "double hit" input and output are significantly different than a "single hit". The input force spectrum for the "double hit" no longer has the wide band constant type characteristics of a single hit. The purpose of this paper is to examine the use of impact vibration testing in relation to the constraints imposed by typical FFT' signal processing techniques. The characteristics of the impact testing procedure are examined with analytical time and spectral functions developed for an idealized test: a single degree of-freedom system excited by a half sine impact force. Once an understanding of the fundamental characteristics is developed it is applied to examine the specific situations encountered in structural impact testing. The relationship of the system's parameters with respect to data capture requirements is evaluated. The effects of exponential windowing are developed to examine the effects on the estimated spectra and modal parameters. Finally, the "double hit" phenomena is examined by combining the results from the single degree-of-freedom system excited by two impulses, one of which is time delayed. The results from these related studies are combined to provide insight into data acquisition guidelines for structural impact testing.

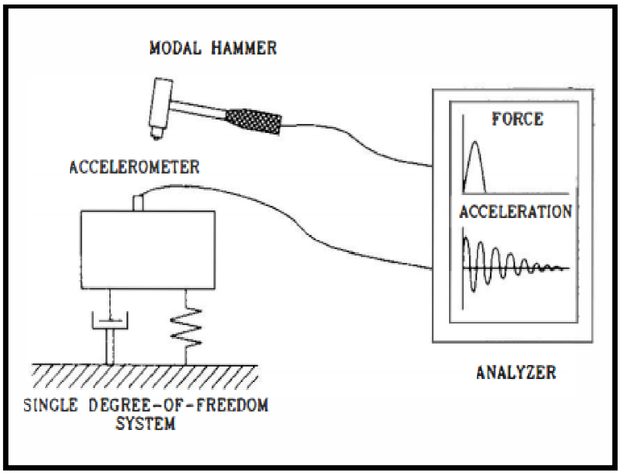
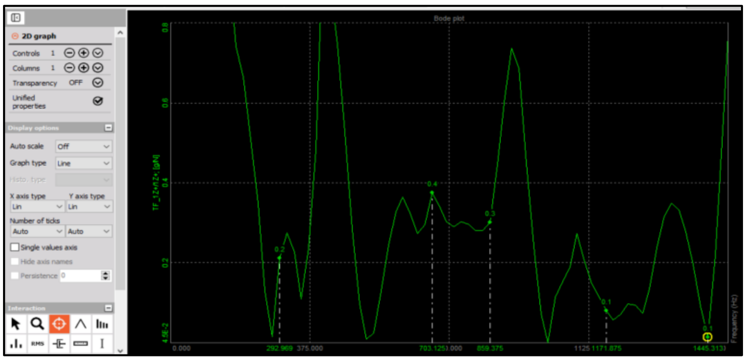


Fig 27: FFT construction

Fig.28 Experimental setup of FFT

EXPERMENTAL PROCEDURE

- Initially fixture is designed according to existing boundary condition as per FEA results.
- FFT analyzer consists of impact hammer, accelerometer, data acquisition system in which each supply is applied to DAS and laptop with DEWSOFT software to view FFT plot.
- Accelerometer is mounted at surface as per high deformation observed in FEA results along with initial impact of hammer are placed for certain excitation to determine frequency of respective mode shapes.
- After impact FFT plot are observed on laptop and comparison of FEA and experimental results are analyzed.
- Five sets of experiments are carried out with the help of new modified lower arm system and results are compared. For sample demonstration a single frequency response on FFT analyzer is shown below.



Controls 1
Columns 1
Transparency OFF
Unified properties
Auto scale Off
Graph type Line
X axis type Lin
Y axis type Lin
Number of ticks Auto Auto

Fig.29 FFT plot of optimized lower control arm

Table 4 Comparison of modified muffler FEA and FFT results

Natural frequency (Hz) MODE SHAPE	**FEA**	**Experimental**	**Percentage error in Experimental and theoretical analysis (%)**
1	297.45	292.96	1.51
2	704.17	703.12	0.15
3	871.46	859.37	1.39
4	1175.9	1171.87	0.34
5	1462.8	1445.30	1.20
Average percentage error in Experimental and theoretical analysis			**0.92**

Hence it is confirmed that the modified design of the lower control arm is safe and has added advantage that it is reduced in the weight by 8% than the existing lower are system used in the market.

CONCLUSION

- In present research existing lower control is redesigned with the help of topology optimization algorithm in ANSYS.
- Weight optimization of 7.8 % is observed as initial weight is around 1.79 kg and reduced weight is around 1.65 kg.
- It is observed that in harmonic analysis frequency response along x and y axis has been reduced in optimized design compared to existing design.
- Experimental FFT analysis natural frequencies are nearly identical with numerically obtained analysis with an average error of 0.92%.
- Hence it is confirmed that modified design of the lower control arm is safe and validated with experimental results.

References

1. 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 (JAMT), ISSN: 1985-3157 Vol. 12 No. 1 January - June 2018
2. 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
3. Do-Hyoung Kim a, Dong-Hoon Choi a, Hak-Sung Kim, “Design optimization of a carbon fiber reinforced composite automotive lower arm “, Composites: Part B 58 (2014) 400–407
4. Xue Guan Song, Ji Hoon Jung, Hwan Jung Son, Joon Hong Park, Kwon Hee Lee, Young Chul Park, “Metamodel-based optimization of a control arm considering strength and durability performance “ , Computers and Mathematics with Applications 60 (2010) 976_980

5. Swapnil S. Khode, Prof. Amol N. Patil, Prof. Amol B. Gaikwad, “Design Optimization of a Lower Control Arm of Suspension System in a LCV by using Topological Approach”, International Journal of Innovative Research in Science, Engineering and Technology Vol. 6, Issue 6, June 2017
6. 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 (2013) 278–302
7. Mohd Viqaruddin, D. Ramana Reddy, “Structural optimization of control arm for weight reduction and improved performance” , Materials Today: Proceedings 4 (2017) 9230–9236
8. Liang Tang, JieWu, Jinhao Liu, Cuicui Jiang, and Wen-Bin Shangguan, “Topology Optimization and Performance Calculation for Control Arms of a Suspension”, Hindawi Publishing Corporation Advances in Mechanical Engineering, Volume 2014, Article ID 734568