Static Structural Analysis &Vibration Optimization of Concept Automotive Lower Control Arm

Rupali G. Dhore, PG Scholar, RMDSSOE, Prof. M. L. Thorat, Asst. Professor, RMDSSOE

***Abstract*— The A 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. Lower control arm performance an 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 a control arm. ANSYS 19.2 software was also used into 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.**

Keywords—Static Structural Analysis, Optimization, FFT analyzer, Impact Hammer Test

I. Introduction

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. 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. Modeling 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 way to improve product quality and reliability is to integrate them in the design and manufacturing process.

Arm is one of the main components in the suspension systems. It can be seen in various types of the suspensions like wishbone or double wishbone suspensions [5]. Most of the times it is called as A-type control arm. It joins the wheel hub to the vehicle frame allowing for a full range of motion while maintaining proper suspension alignment. Uneven tyre wear, suspension noise or misalignment, steering wheel shimmy or vibrations are the main causes of the failure of the A suspension arm. Most of the cases the failures are catastrophic in nature. So the structural integrity of the suspension arm is crucial from design point of view both in static and dynamic conditions. As the Finite Element Method (FEM) gives better visualization of this kind of the failures so FEM analysis of the stress distributions around typical failure initiations sites is essential. Therefore in this dissertation work it is proposed to carry out the structural analysis of a suspension arm of light commercial vehicle using FEM.

In the automotive industry, the riding comfort and handling qualities of an automobile are greatly affected by the suspension system, in which the suspended portion of the vehicle is attached to the wheels by elastic members in order to cushion the impact of road irregularities. Suspension arm is one of the main components in the suspension systems. It can be seen in various types of the suspensions like wishbone or double wishbone suspensions. Most of the times it is called as A-type control arm. It joins the wheel hub to the vehicle frame allowing for a full range of motion while maintaining proper suspension alignment. Uneven tyre wear, suspension noise or misalignment, steering wheel vibrations are the main causes of the failure of a suspension arm. Most of the cases the failures are catastrophic in nature. Hence, it is reported that the structural integrity of the suspension arm is crucial from design point of view both in static and dynamic conditions.

Optimization procedure for optimal design of flexible multi-body systems using the floating frame of reference approach is analyzed. The optimization is based on fully dynamical simulations and equivalent static loads [1].

Weight reduction has become a primary concern in automotive industry. In fact, safety standards and emission regulations impose conflicting performance targets that need to be satisfied at the same time. While the respect of the safety standards pushes the automotive design process towards heavy weight solutions, environmental issues and handling call for a resolute vehicle weight reduction. Topology optimization is defined as finding out the best possible solution of problem by considering the given sets of objective and number of constraints. For solving any topology optimization problem it has to specify three parameters that is design variables (material density), design objective (weight reduction) and design constraints (volume). Hence in order to accomplish the objective of weight reduction over existing design, finite element analysis method is used. While maintaining a critical factor of safety, FEA is used to remove the maximum amount of material from suspension system [2].

McPherson strut required the introduction of unibody construction, because it needs a substantial vertical space and a strong top mount, which unibodies can provide, while benefiting them by distributing stresses. The strut will usually carry both the coil spring on which the body is suspended and the shock absorber, which is usually in the form of a cartridge mounted within the strut. The strut also usually has a steering arm built into the lower inner portion. The whole assembly is very simple and can be preassembled into a unit; also by eliminating the upper control arm, it allows for more width in the engine bay, which is useful for smaller cars, particularly with transverse- mounted engines such as most front wheel drive vehicles have. It can be further simplified, if needed, by substituting an anti- roll bar (torsion bar) for the radius arm. For those reasons it has become almost ubiquitous with low cost manufacturers [3].

II. LITERATURE REVIEW

Robert Seifried et al. The main goal is to reduce the mass of flexible members without deteriorating the accuracy of the system. In this paper structural optimization based on topology optimization of members of flexible multi-body system is introduced and the effects of uncertainty in the optimization process are investigated. Two sources of uncertainty, namely the model uncertainty and the un-certainty in usage are addressed. As an application example, a two-arm manipulator is 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].

Mohd. Viqaruddina et al. 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 [2].

M. Sridharan et al. The control arm is the most vital component in a suspension system. There are two control arms, Acontrol arm and upper control arm. A 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 A ball joint. Significant amount 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. The existing design has been modified, by reducing the thickness of the existing profile and the reinforcement plate has been proposed. The optimization of a control arm is done by applying the DOE method. The Parameters are identified. The FEA is done on LCA and the buckling load has been compared with the existing component [3].

Dattatray Kothawale et al. This paper deals with finite element analysis for MacPherson type suspension system Acontrol arm (LCA) of 4W suspension system. The main function of the control 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. In this project we have prepared CAD Model using PRO-E Software & finite element analysis using Ansys software [4].

Prashant Gunjan et al. The A-arm (called a Volvo control arm) attaches the suspension to the chassis of the vehicle. There may be as many as three or four control arms when coil springs are used in both the front and rear suspension systems. Upper control arms carry driving and braking torque, while the control arms pivot, providing up-and-down movement for wheels. A-arms can be used in different configurations and numbers. Two A-arms per wheel makes up a suspension system called a double wishbone suspension, or an independent suspension. The control arms of a vehicle connect a vehicle's steering rack to the wheels of the car, and they hold the wheels to the car's frame. Control arms allow the wheels to move and manage the motion of the wheels by pivoting. They assist in the wheels to response to varying road conditions by allowing the wheels to lift and descend as the wheels encounter bumps, dips, or other obstructions in the road. In addition to allowing for movement, control arms also assist the wheels in maintaining straight lines in relation to the road [5].

Balasaheb Gadade et al. In this project work mainly focused on the finite element based stress analysis of A – Type suspension arm. The main objective of this study is to calculate working life of the component under static loading. The A – Type a suspension arm was developed by using CAD software. Actual model was manufacture as per Design by using AISI 1040 material. The finite element modeling and analysis was performed by using HYPERMESH software. Mesh was created with 10 node tetrahedral element. A simple design approach was used to calculate effect of stresses on A – Type. The suspension arm element under static loading condition. After manual calculations a modern computational approach based on FEA for integrated durability assessment in an automotive. The suspension arm component is presented. The experimental work includes validation of the FEA results with actual testing of the model under stress. This is carried out with computerized universal testing machine (UTM) of 25 ton capacity [6].

Prof. Amol B. Gaikwad et al. This study deals with Finite Element Analysis of the A control arm of Mac-pherson suspension system and its optimization under static loading condition. The existing design of a control arm from one of the light commercial vehicle is selected for the study. In order to determine the deformation and stress distribution in the current design, the finite element analysis is carried out. The main aim of this paper is to optimize the A control arm of Mac-Pherson suspension system under the current boundary conditions for weight reduction [7].

S. Arjun Kumar et al. The main objective of this paper is project is to model and to perform structural analysis of Control arm used in the front suspension system, which is a sheet metal component. A 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 [8].

S. C. Shilwant et al. These analysis were carried using Altair Hyperworks and solver used is Abacus. In order to reduce stresses and to improve structural strength Topography optimization approach is carried out in Hyperworks in which a design region for a given part is defined and a pattern of shape variable-based reinforcements within that region is generated to increase Stiffness [9].

Hardial Singh et al. This paper deals with the finite element analysis for Macpherson type suspension system front Control arm (LCA) of the 4 wheeler suspension system. Finite element analysis (FEA) methods are used to predict the structural performance of the design .The structural safety of an automobile's suspension is crucial because it plays an important role in the vehicle's stability and handling performance. The front a control arm consists of more than 50 surfaces. These components are made of hot rolled steel (JSH 590B).The analysis has been carried out with actual design considerations and loading conditions [10].

Siddharth Saha et al. The static structural analysis was done to find out the stress, deformation and safety factor of component. The model was meshed using 10-noded tetrahedral elements. Result obtained from the analysis were studied to check whether the design is safe or not. In some cases the stresses becomes more than safe limit. In that case optimization approach is carried out to increase the structural strength of the component [11].

III. 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 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.

IV. 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 model analysis of the A type control arm using ANSYS Software.
5. Testing and validation of Automotive A type control arm with the help of FFT analyzer and Impact Hammer Test.

V. METHOLOGY

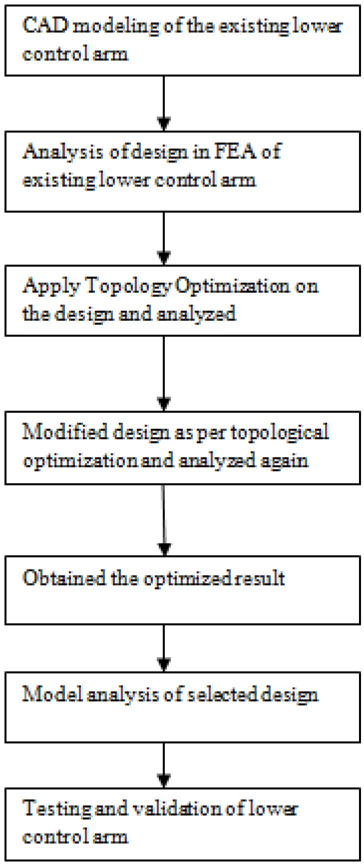


Fig.No.1 Flowchart for the Methodology

STATIC ANALYSIS

1. Material Properties:

1.	Steel	
2.	Modulus of Elasticity	: 200GPa
3.	Poisson's ratio	: 0.30
4.	Density	: 7.85e-6 kg/mm^3
5.	Yield Strength	: 520 MPa
6.	Mass of the body	: 2.6042kg

2. Finite Element Analysis:
Design of existing suspension control Arm is done by using CAD package CATIA V5 as per following;

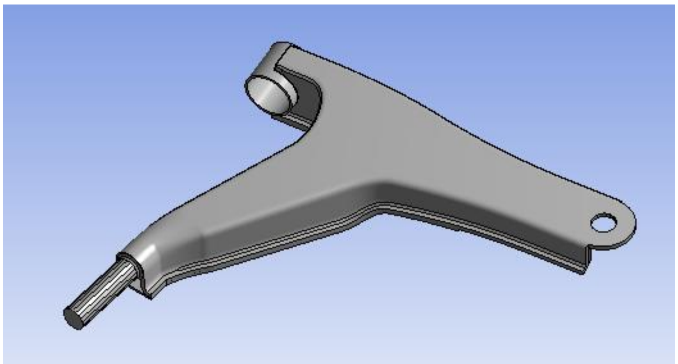


Fig.No.2 Original Model

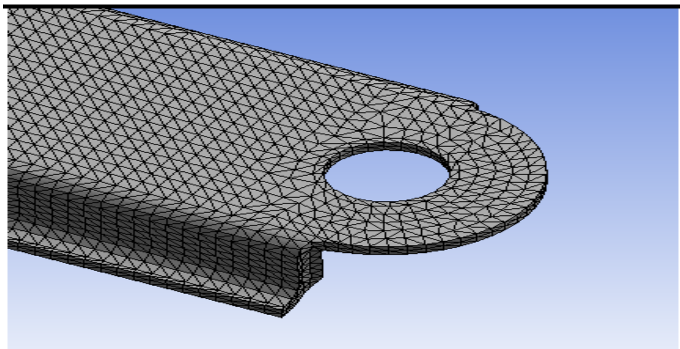


Fig.No.3Solid Mesh

Elements Details:
1. Element Type: Tetrahedron
2. Element Order: second Order
3. Mesh Method: Solid
4. Node Population count: 154821
5. Element Population count: 80483

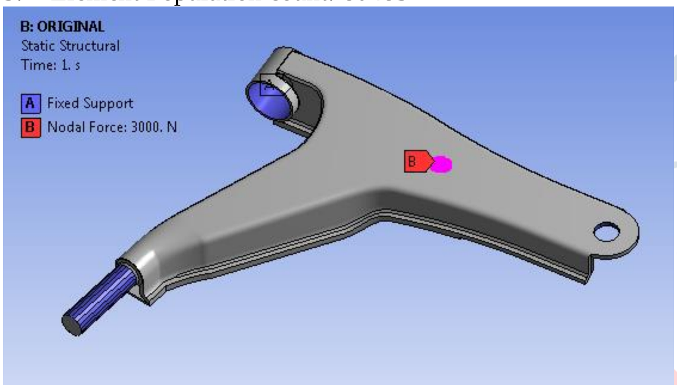


Fig.No.4 Boundary Conditions Original Model

1. Total weight of Indica = 1000kg=10000N (Approx)
2. Around 60 per load acts on front axle= 6000N
3. Load Acting on one ARM = 6000/2=3000N

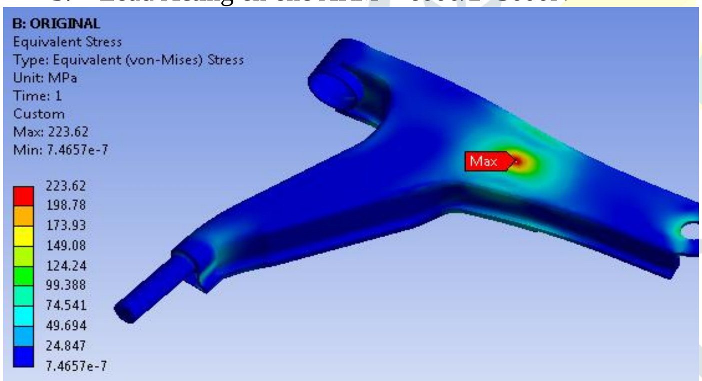


Fig.No.5 Von-Mises Stress Original Model

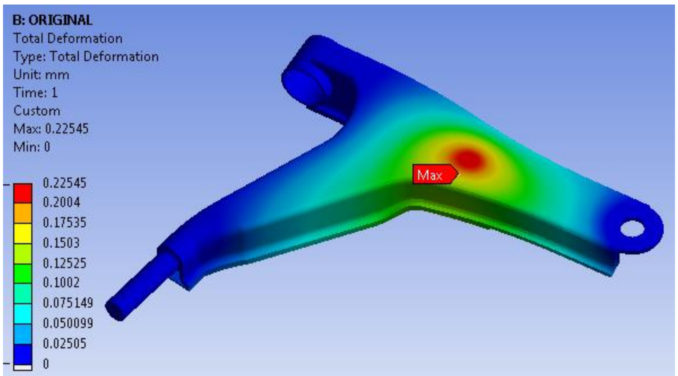


Fig.No.6 Deformation Original Model

OPTIMIZATION

Optimization is a process of determining the best design from a given design and material by applying loads and boundary conditions.

Mass of the body = 2.433kg

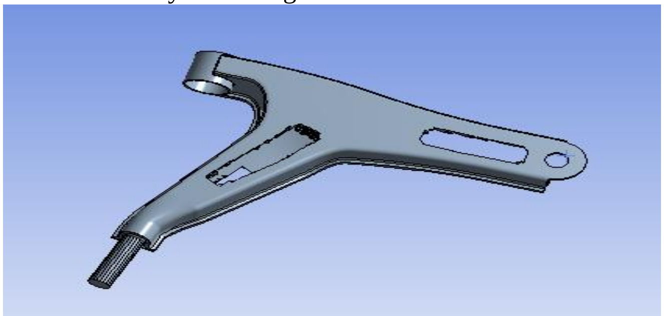


Fig.No.7 Optimized Model

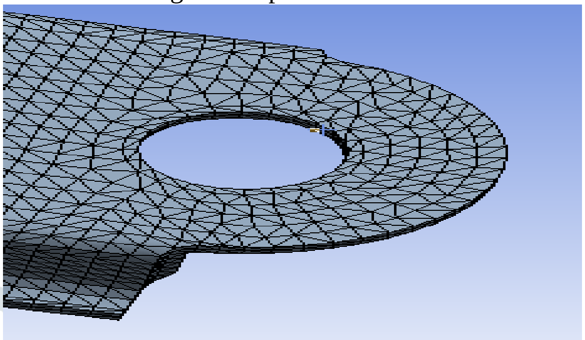


Fig.No.8 Solid Mesh

Elements Details:
- Element Type: Tetrahedron
- Element Order: Second Order
- Mesh Method: Solid
- Node Population count: 1288
- Element Population count: 696

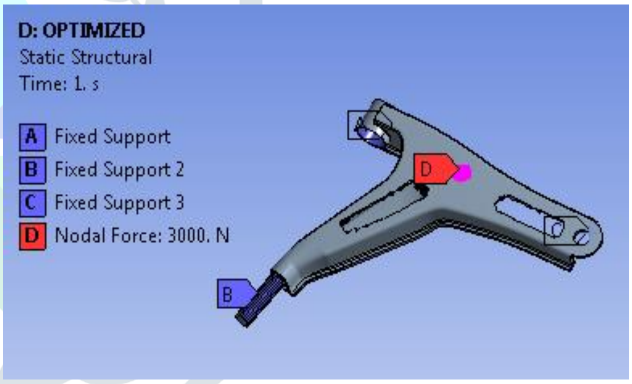


Fig.No.9 Boundary Conditions Optimized Model

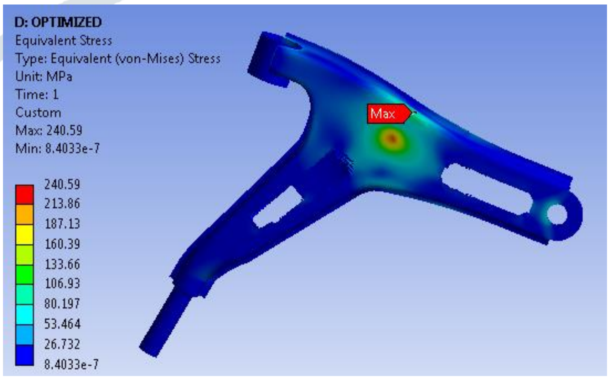


Fig.No.10 Von-Mises Stress Optimized Model

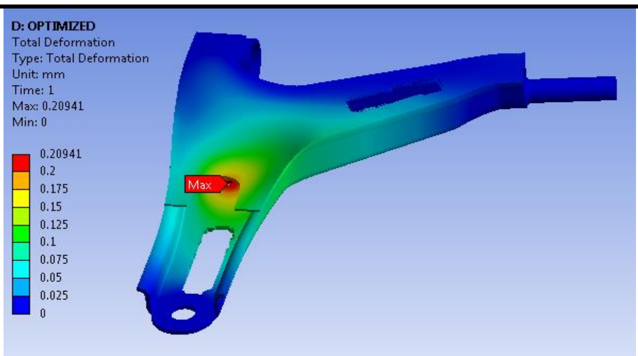


Fig.No.11 Deformation Optimized Model

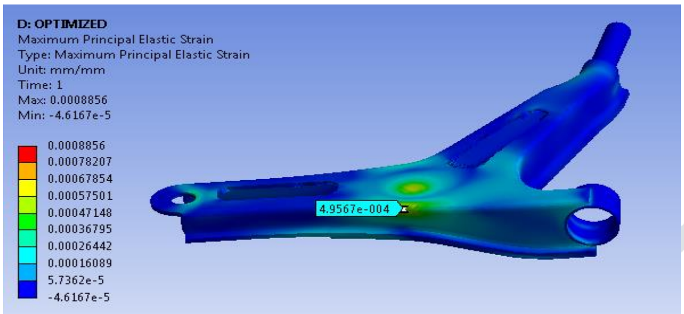


Fig.No.12 Maximum Principal Strain Optimized Model

MODAL ANALYSIS

Every object has an internal frequency (or resonant frequency) at which the object can naturally vibrate. It is also the frequency where the object will allow a transfer of energy from one form to another with minimal loss. As the frequency increases towards the “resonant frequency,” the amplitude of response asymptotically increases to infinity. In other words, the results of the modal analysis are these frequencies at which the amplitude increases to infinity.

Every system can be described in terms of a stiffness matrix that connects the displacements and forces. These frequencies are known as natural frequencies of the system and are provided by the eigenvectors of the stiffness matrix. These frequencies are also known as the resonant frequencies.

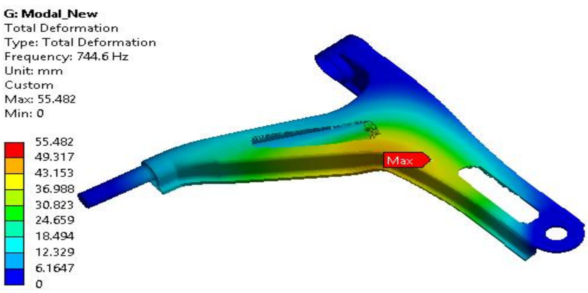


Fig.No.13 model analysis mode 1

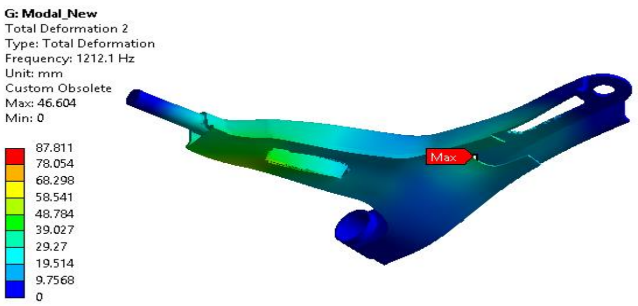


Fig.No.14 model analysis mode 2

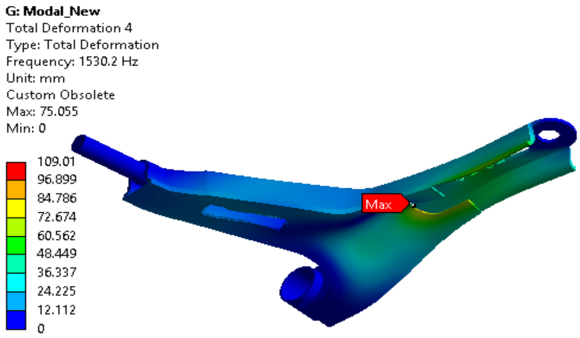


Fig.No.15 model analysis mode 3

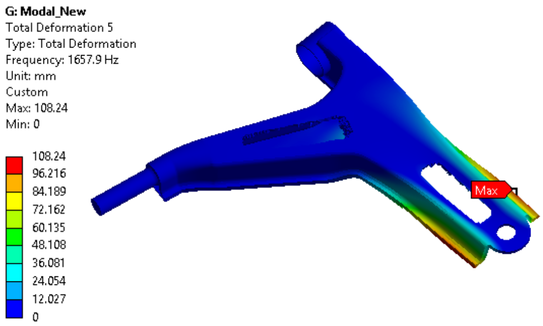


Fig.No.16 model analysis mode 3

VI. EXPERIMENTAL RESULTS

FFT analysis

FFT is one main property in any sequence being used in general. To find this property of FFT for any given sequence, many transforms are being used. The major issues to be noticed in finding this property are the time and memory management. Two different algorithms are written for calculating FFT and Autocorrelation of any given sequence. Comparison is done between the two algorithms with respect to the memory and time managements and the better one is pointed. Comparison is between the two algorithms written, considering the time and memory as the only main constraints. Time taken by the two transforms in finding the fundamental frequency is taken. At the same time the memory consumed while using the two algorithms is also checked. Based on these aspects it is decided which algorithm is to be used for better results

DEWE-43 Universal Data Acquisition Instrument

When connected to the high speed USB 2.0 interface of any computer the DEWE-43 becomes a powerful measurement instrument for analog, digital, counter and CAN-bus data capture. Eight simultaneous analog inputs sample data at up to 204.8 kS/s and in combination with DEWETRON Modular Smart Interface modules (MSI) a wide range of sensors are supported Voltage Acceleration Pressure Force Temperature Sound Position RPM Torque Frequency Velocity And more The included DEWESoft application software adds powerful measurement and analysis capability, turning the DEWE-43 into a dedicated recorder, scope or FFT analyzer.

suspension control Arm	Existing	Optimized	Weight reduction	% Weight reduction
Weight	2.60kg	2.43kg	0.17kg	6.53%

Fig.No.17 Experimental setup of FFT

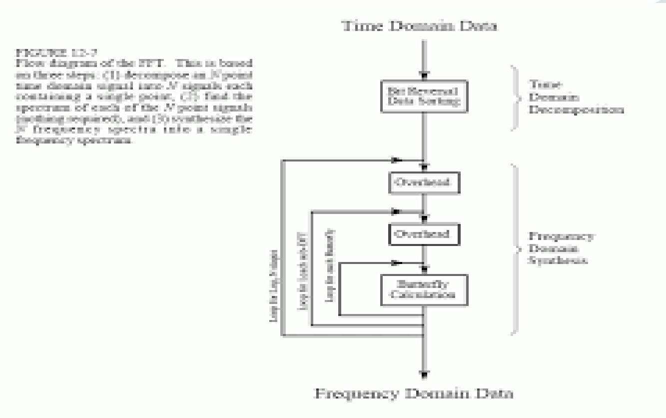


Fig.No.18 Block diagram of FFT

TEST FFT RESULTS

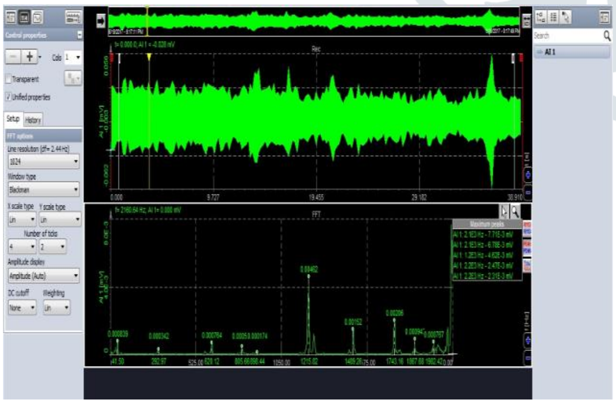


Fig.No.19 Testing result

VII. RESULTS & DISCUSSIONS

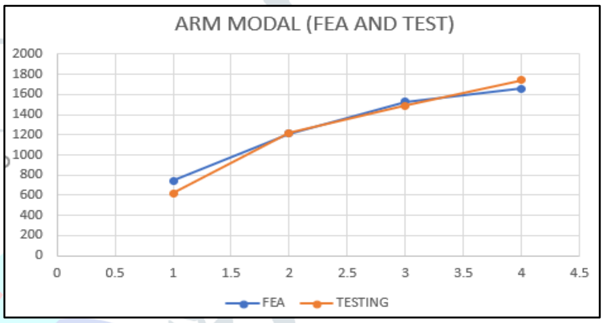
After, Experimental testing is carried for static loading condition on suspension control Arm following Results obtained:

Sr. No.	suspension control Arm	Existing	Optimized
1.	Von-Mises Stress	223.59MPa	240.62MPa
2.	Deformation	0.2254mm	0.2094mm

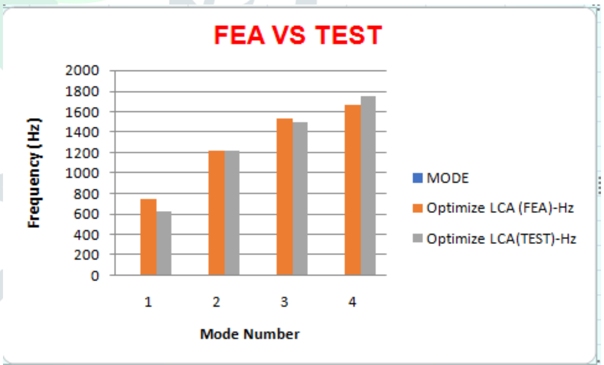
Weight Reduction of suspension control Arm:

COMPARATIVE PLOT OF FEA VS FFT

MODE	FEA	TESTING
1	744.6	620
2	1212.1	1215.82
3	1530.2	1489.26
4	1657.9	1743



Graph 1 COMPARATIVE PLOT OF FEA VS FFT



VII. CONCLUSION

1. The suspension control Arm had the potential for optimization and weight reduction.
2. The optimized suspension control Arm is 6.53% lighter than the existing suspension control Arm. Both design produced stresses are within the yield limit 520 MPa of the material.
3. To model analysis of the A type control arm has done.
4. Natural Frequencies from model analysis and FFT testing are in good relationship hence project is validated.

VIII. SCOPES FOR FUTURE WORK

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

1. Implementing new design changes in the existing model can be done.
2. Identifying more materials for the proposed design and new design increasing the number of levels of the response.

REFERANCES

1. Robert Seifried, "Analysis of Design Uncertainties in Structurally Optimized Lightweight Machines", Procedia IUTAM 13, 2015, PP 71 – 81.
2. Mohd. Viqaruddina, "Structural optimization of control arm for weight reduction and improved performance", Materials Today Proceedings 4, 2017, PP 9230–9236.
3. M. Sridharan, "Design and Analysis of AControl ARM" International Journal of Innovative Research in Science, Engineering and Technology, vol. 5, Issue 4, April 2016, PP 2347-6710.
4. Dattatray Kothawade, "Analysis of a Control Arm in Front Suspension System Using F.E.A. Approach" International Journal of Engineering Research and Development, Vol. 5, Issue 12, Feb. 2013, PP 18-23.
5. Prashant Gunjan, "Optimization Approach of Front Wishbone Suspension Arm" IJARIIE, Vol. 4, Issue 2, 2018, PP 2395-4396.
6. Mr. Balasaheb Gadade, "Design, analysis of A-type front A suspension arm in" International Research Journal of Engineering and Technology (IRJET), Vol. 2, Issue 7, Oct 2015, PP 2395-0072.
7. Prof. Amol B. Gaikwad, "Design Optimization of a A 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, PP 2347-6710.
8. S. Arjun Kumar, "ANALYSIS AND OPTIMIZATION OF ACONTROL ARM IN FRONT SUSPENSION SYSTEM "Int. J. Chem. Sci., Vol. 14, 2016, PP1092-1098.
9. S. C. Shilwant, "Finite Element Analysis and Topography Optimization of A Arm of Double Wishbone Suspension Using Abacus and Optistruct", Journal of Engineering Research and Applications, Vol. 4, Issue 7, July 2014, pp.112-117.
10. Hardial Singh, "Finite Element Analysis of a Front A Control Arm of LCV Using Radioss Linear", Research gate, April 2012, PP 121-125.
11. Siddharth Saha, "STATIC STRUCTURAL ANALYSIS OF SUSPENSION ARM USING FINITE ELEMENT METHOD" International Journal of Research in Engineering and Technology, PP 402-406.