Dynamic Analysis of an Lower Control Arm Using Harmonic Excitation for Investigation Dynamic Behaviour

Sudarshan Bhalshankar

June 27, 2021

DYNAMIC ANALYSIS OF AN LOWER CONTROL ARM USING HARMONIC EXCITATION FOR INVESTIGATION DYNAMIC BEHAVIOUR

Er. SudaRshan D. Bhalshankar

M.B.A [HR.]*, M.E. [CAD/CAM] , B.E [Mechanical Engineering],

D.A.E [Diploma in Automobile Engineering].

Sudarshanbhalshankar111@gmail.com

PradnyaSurya Engineering Works Pvt Ltd.

pradnyasuryaengineerinworks@gmail.com

ABSTRACT

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 tire wear, suspension noise or misalignment, steering wheel vibrations are the main causes of the failure of the lower 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. Finite Element Method (FEM) gives better visualization of this kind of the failures.

Keywords :- FFT , FEA , LCA , Impact, Harmonic, Vibrations,

INTRODUCTION

Suspension is the system of tires, tire air, springs, shock absorbers and linkages that connects a vehicle to its wheels and allows relative motion between the two. Suspension systems must support both road holding/handling and ride quality, which are at odds with each other. The tuning of suspensions involves finding the right compromise. It is important for the suspension to keep the road wheel in contact with the road surface as much as possible, because all the road or ground forces acting on the vehicle do so through the contact patches of the tires. The suspension also protects the vehicle itself and any cargo or luggage from damage and wear. The design of front and rear suspension of a car may be different.

This system means that the suspension is set-up in such a way that allows the wheel on the left and right side of the vehicle to move vertically independent up and down while driving on uneven surface. Force acting on the single wheel does not affect the other as there is no mechanical linkage present between the two hubs of the same vehicle. In most of the vehicle it is employed in front wheels. These types of suspension usually offer better ride quality and handling due to less unsprung weight. The main advantage of independent suspension are that they require less space, they provide easier steer ability, low weight etc. Examples of Independent suspension are, Double Wishbones .

LITERATURE SURVEY

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.Sridharanet al.The A control arm is the most vital component in a suspension system. There are two control arms, A control 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 A control arm (LCA) of 4W suspension system. The main function of the A 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. We have studied to calculate various dynamic loads like road bump, kerb strike, braking, cornering & acceleration load case. By applying all this forces in X, Y and Z directions perform non-linear static analysis using Ansys software [4]

Prashant Gunjanet 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 A 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 Gadadeet al.In this project work mainly focused on the finite element based stress analysis of A – Type A 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 modelling 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 A suspension arm element under static loading condition. After manual calculations a modern computational approach based on FEA for integrated durability assessment in an automotive A 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].

DESCRIPTION OF THE PROBLEM

***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 dynamic analysis of an arm using harmonic excitation for investigation dynamic behaviour.

OBJECTIVES:

1. To determine the problem associated due to vibration on lower control and design of existing lower control arm in CATIA software.

2. Modelling 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.

SCOPE:

The suspension system consists of lower control arm which undergoes failure due to different parameters like strength, natural frequency of lower control arm etc. The scope of the project includes the FEA of existing and optimized lower control arm.

METHODOLOGY:

Step 1: - I started the work of this project with literature survey. I gathered many research papers which are relevant to this topic.
Step2: - After that the components which are required for our project are decided.
Step 3: - After deciding the components, the 3D Model and drafting will be done with the help of CATIA software.
Step 4: - The Analysis of the components will be done with the help of ANSYS using FEA.

Step 5: - The Experimental Testing will be carried out.

Step 6: - Comparative analysis between the experimental & analysis result & then the result & conclusion will be drawn.

DESIGN OF LOWER CONTROL ARM

DESIGN:

CAD: -

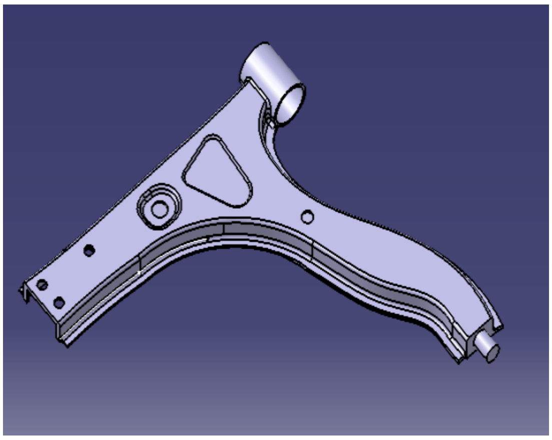


Fig. 4 CATIA model

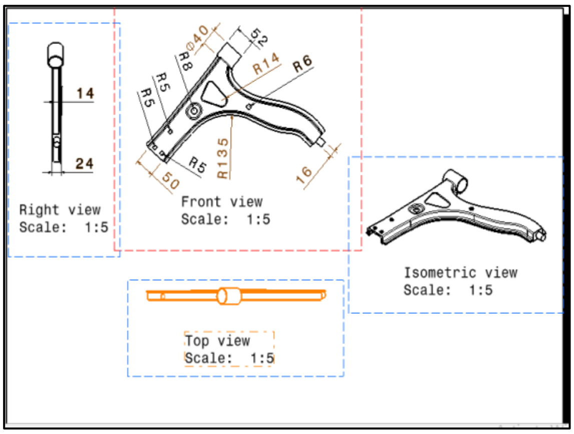


Fig 5 Drafting of existing model

Computer-aided design (**CAD**) is the use of computer systems (or workstations) to aid in the creation, modification, analysis, or optimization of a design. CAD software is used to increase the productivity of the designer, improve the quality of design, improve communications through documentation, and to create a database for manufacturing. CAD output is often in the form of electronic files for print, machining, or other manufacturing operations. The term **CADD** (for *Computer Aided Design and Drafting*) is also used.

Its use in designing electronic systems is known as electronic design automation (**EDA**). In mechanical design it is known as mechanical design automation (**MDA**) or **computer-aided drafting** (**CAD**), which includes the process of creating a technical drawing with the use of computer software.

CAD software for mechanical design uses either vector-based graphics to depict the objects of traditional drafting, or may also produce raster graphics showing the overall appearance of designed objects. However, it involves more than just shapes. As in the manual drafting of technical and engineering drawings, the output of CAD must convey information, such as materials, processes, dimensions, and tolerances, according to application-specific conventions.

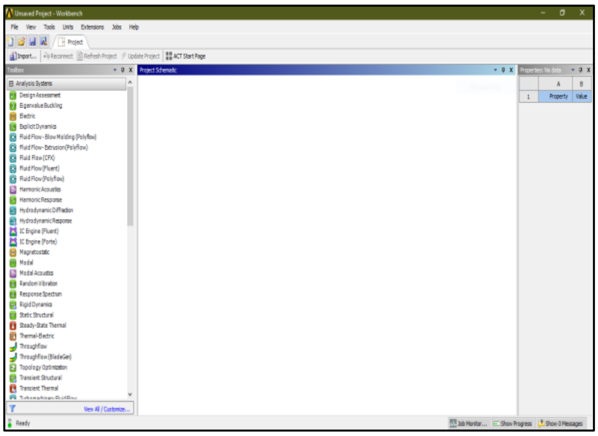
FEA OF LOWER CONTROL ARM

FEA (FINITE ELEMENT ANALYSIS)

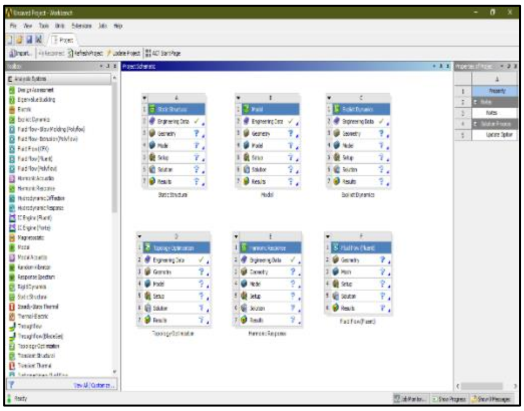
The finite element method (FEM), is a numerical method for solving problems of engineering and mathematical physics. Typical problem areas of interest include structural analysis, heat transfer, fluid flow, mass transport, and electromagnetic potential. The analytical solution of these problems generally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

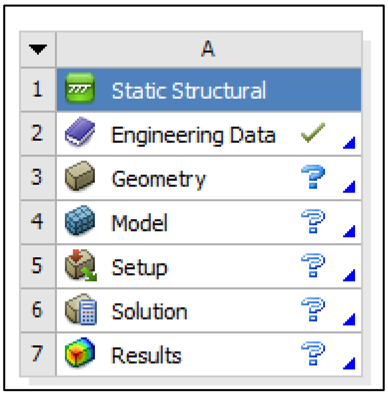
In the first step, the element equations are simple equations that locally approximate the original complex equations to be studied, where the original equations are often partial differential equations (PDE). The process, in mathematical language, is to construct an integral of the inner product of the residual and the weight functions and set the integral to zero. In simple terms, it is a procedure that minimizes the error of approximation by fitting trial functions into the PDE. The residual is the error caused by the trial functions, and the weight functions are polynomial approximation functions that project the residual. The process eliminates all the spatial derivatives from the PDE, thus approximating the PDE locally with

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



Workbench contain analysis of different types namely static, modal, harmonic, explicit dynamics, CFD, ACP tool post, CFX, topology optimization etc. as per problem defined.





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 tria 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.

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

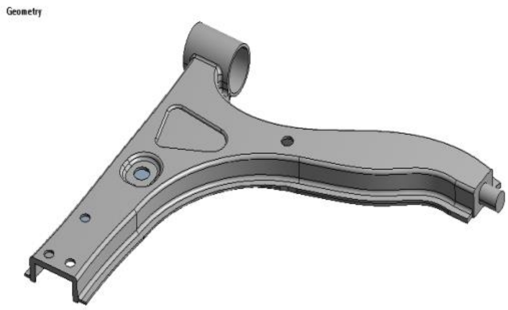


Fig. Existing design of lower control arm

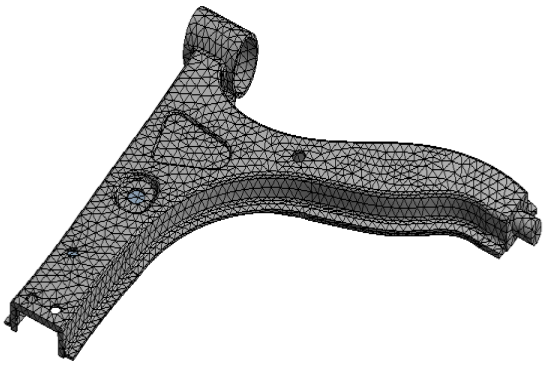


Fig. 6 Meshing of model

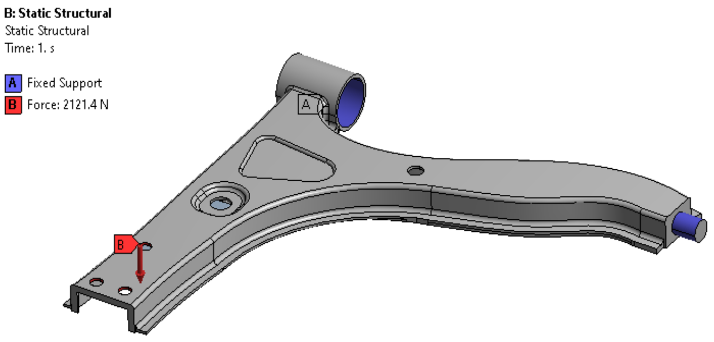
Details of "Body Sizing" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Type	Element Size
Element Size	4.0 mm

Statistics	
Nodes	28547
Elements	14197

Elements Details:

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. 7 Boundary condition of model

Total Deformation

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

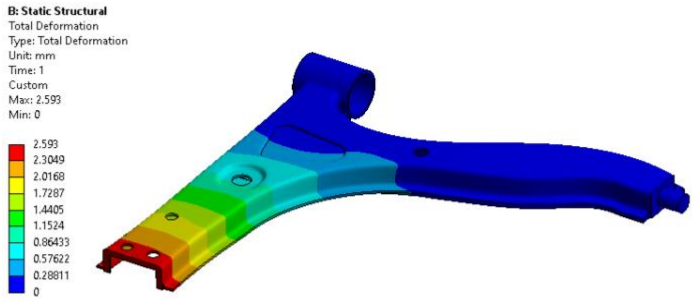


Fig. 8 Total Deformation of model

Equivalent Stress

Equivalent stress is related to the principal stresses by the equation:

$$\sigma_e = \left[\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}{2} \right]^{1/2}$$

Equivalent stress (also called *von Mises stress*) is often used in design work because it allows any arbitrary three-dimensional stress state to be represented as a single positive stress value. Equivalent stress is part of the maximum equivalent stress failure theory used to predict yielding in a ductile material.

The von Mises or equivalent strain ε_e is computed as:

$$\varepsilon_e = \frac{1}{1+\nu'}\left(\frac{1}{2}\left[(\varepsilon_1-\varepsilon_2)^2+(\varepsilon_2-\varepsilon_3)^2+(\varepsilon_3-\varepsilon_1)^2\right]\right)^{\frac{1}{2}}$$

Where:

ν' = effective Poisson's ratio

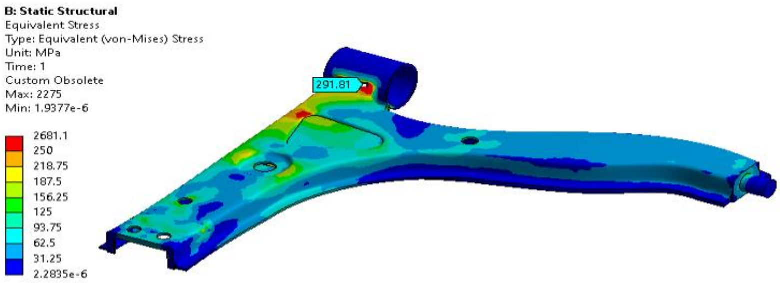
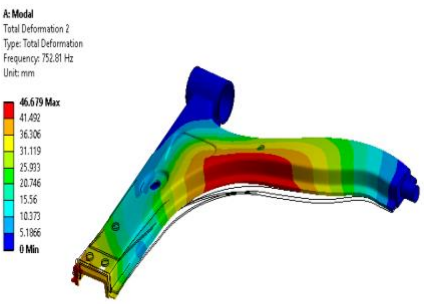


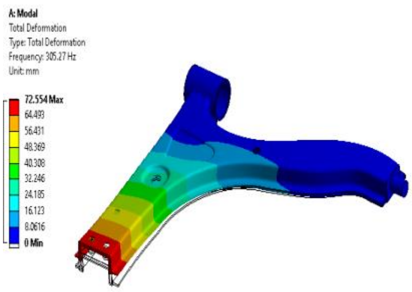
Fig. 9 Equivalent stress of model

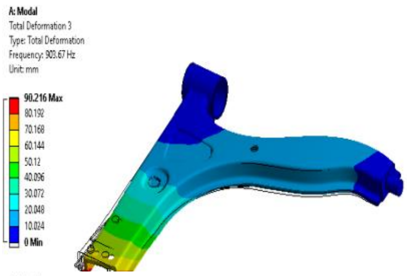
MODEL ANALYSIS

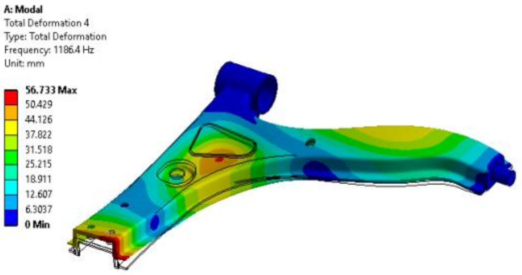
Every object has an internal frequency (or resonant frequency) at which the object can naturally vibrate. It is also the frequency where the object will allow a transfer of energy from one form to another with minimal loss. As the frequency increases towards the "resonant frequency," the amplitude of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

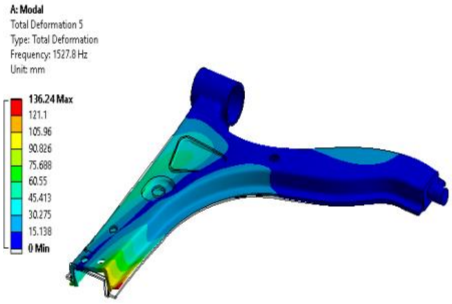
Every system can be described in terms of a stiffness matrix that connects the displacements and forces. These frequencies are known as natural frequencies of the system and are provided by the eigenvectors of the stiffness matrix. These frequencies are also known as the resonant frequencies.











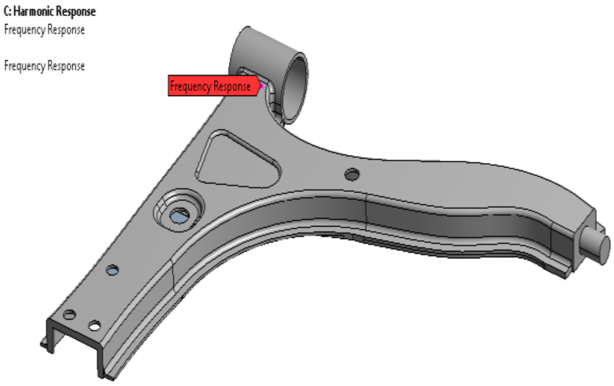
Tabular Data

	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

Fig. 5 Set of Mode Shape of Existing LCA.

Table. Tabular data of natural frequency of respective mode shape

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

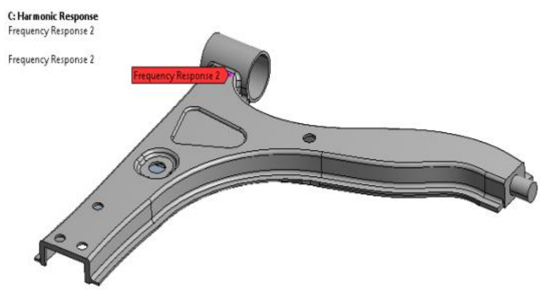


Fig. Boundary condition for harmonic frequency response

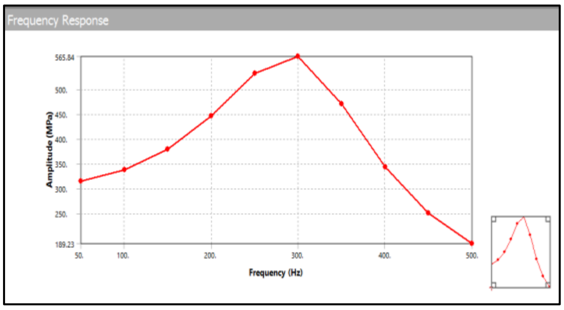


Fig. frequency response result along x axis

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. Boundary condition for harmonic frequency response

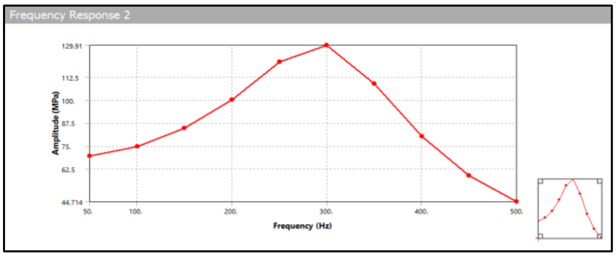


Fig. frequency response result along y axis

TOPOLOGY OPTIMIZATION OF EXISTING COMPONENT

TOPOLOGY OPTIMIZATION

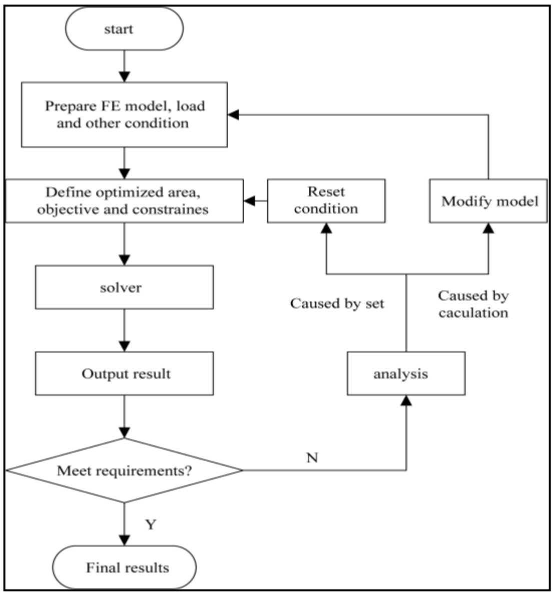
Topology optimization is a mathematical approach that optimizes material layout within a given design space, for a given set of loads and boundary conditions such that the resulting layout meets a prescribed set of performance targets.

Process of Topology Optimization

Based on Hyper Works platform topology optimization holder, first, according to the engine mounting position, we establish the three-dimensional geometric model of engine bracket, and then pre-treated in HyperMesh, define design area, objective function and constraints under the optimization panel, finally operate topology optimization which design process is as Figure 2.

Analysis

To establish the geometry model by CATIA, then input the geometry to the ANSY to carry out pre-treatment operations like geometry clean up, meshing, loads, constraints, etc.



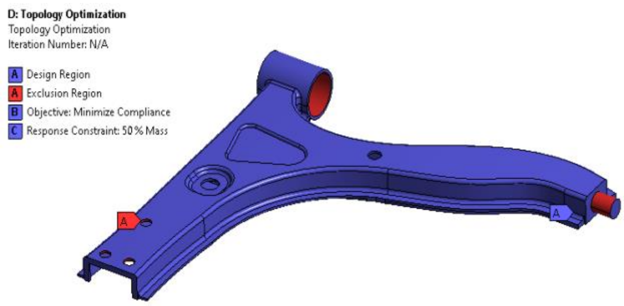


Fig. Boundary condition for topology optimization region

Fig.17: Process of Topology Optimization

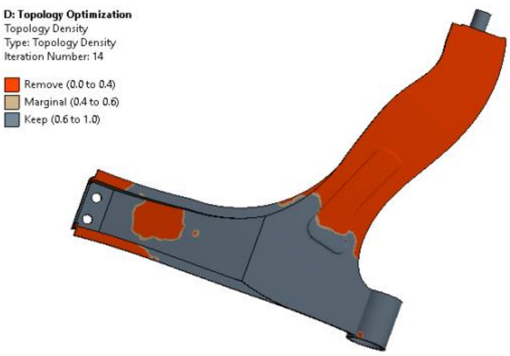
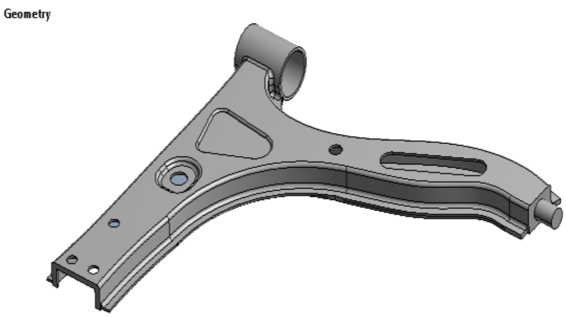


Fig. Topology optimized results

- Red region indicates material removal area region.



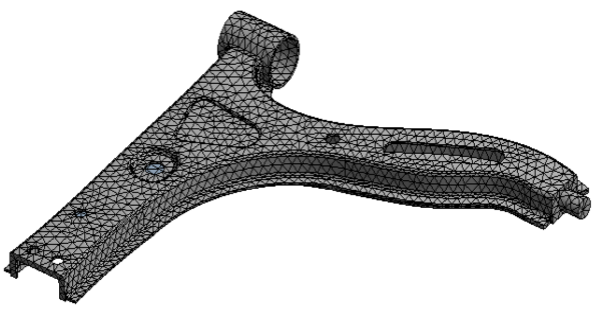


Fig. Optimized lower control arm

Statistics	
Nodes	27382
Elements	13451

Fig. Details of optimized lower control arm

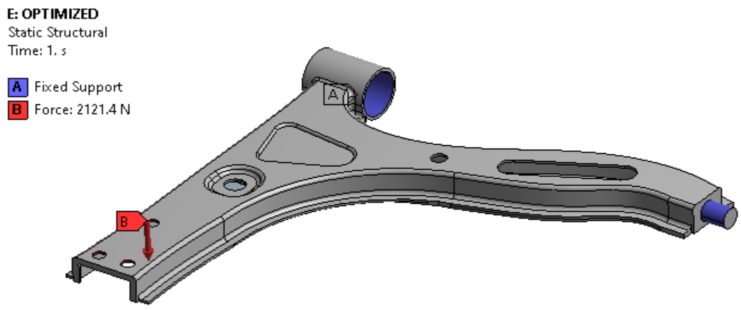
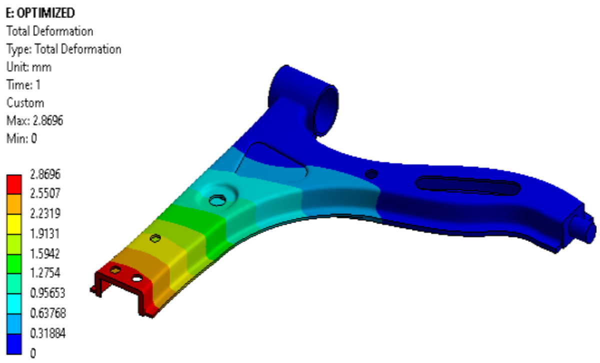


Fig. Boundary condition for optimized lower control arm



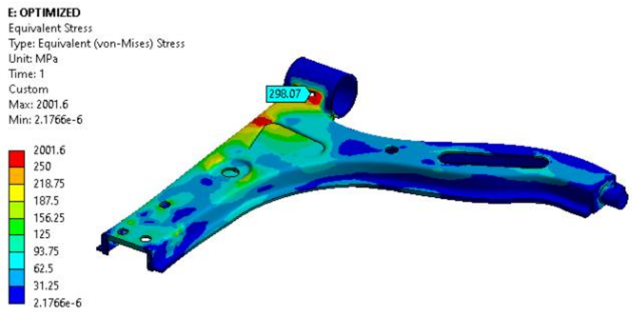
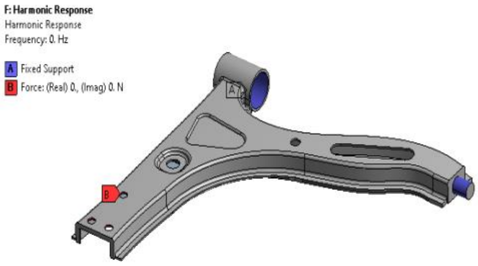
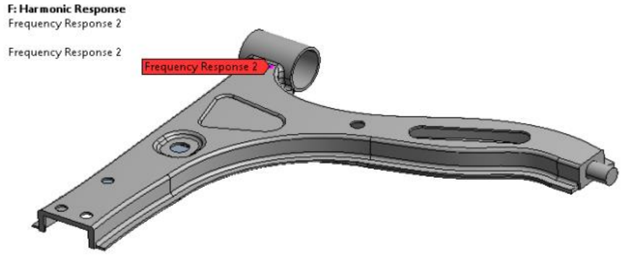


Fig. Optimized lower control arm deformation result

Fig. Optimized lower control arm equivalent stress result

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

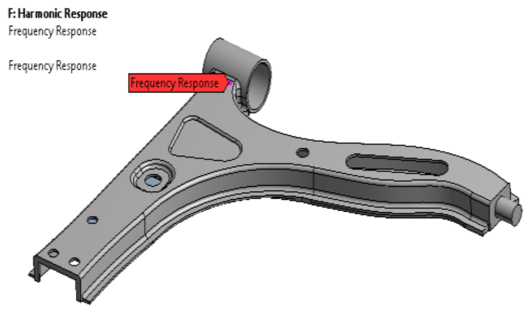


Fig. Boundary condition for Optimized lower control arm for harmonic response surface along x axis

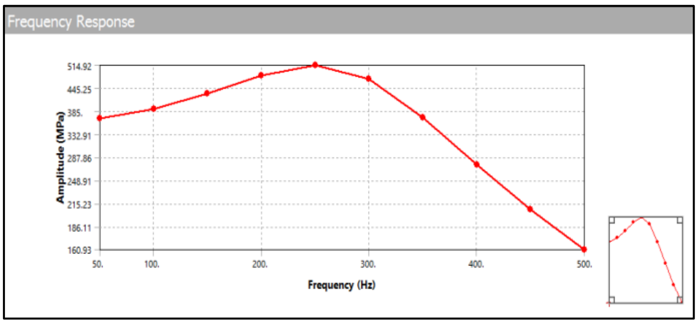


Fig. Frequency response result for optimized lower control arm along x axis

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

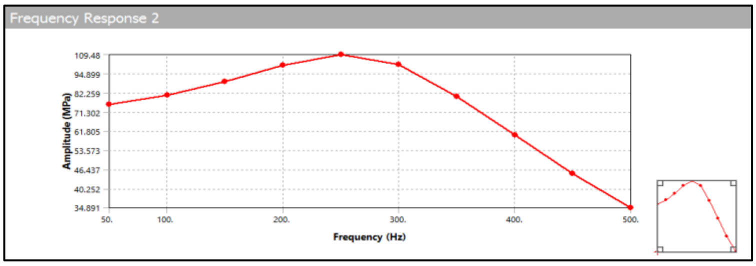


Fig. Frequency response result for optimized lower control arm along y axis

- It is observed from harmonic analysis that amplitude along x and y axis in optimized design have decreased compared to original design.

Model analysis of optimized lower control arm

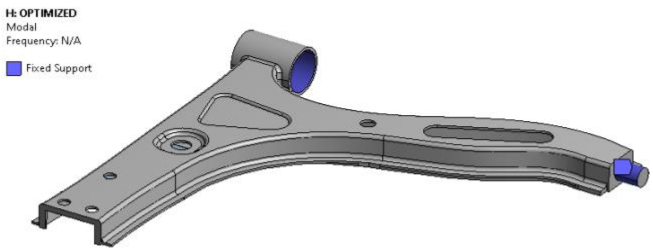
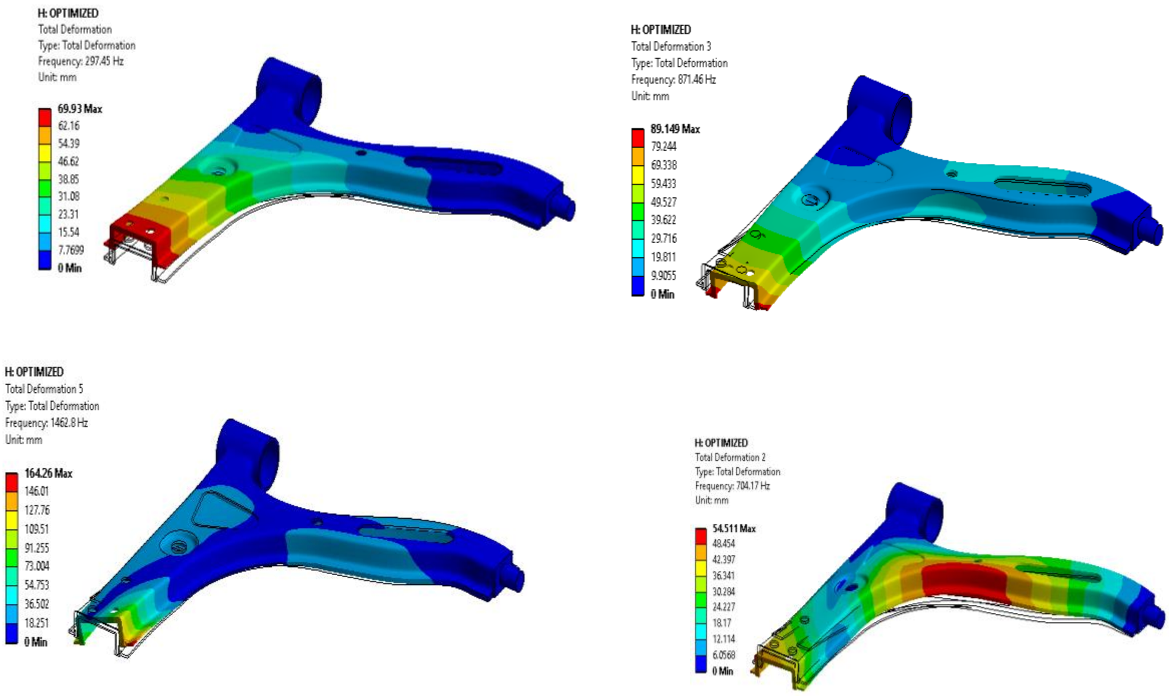


Fig. Boundary condition for modal analysis

Fixed support is applied as per existing boundary condition to determine natural frequency and mode shape.



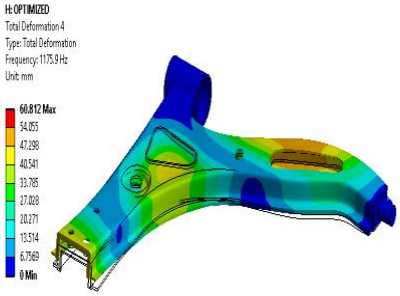


Fig. 5 Set of Mode Shape

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

Table. Tabular data of natural frequency with respective mode shapes

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.

A phenomena 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.

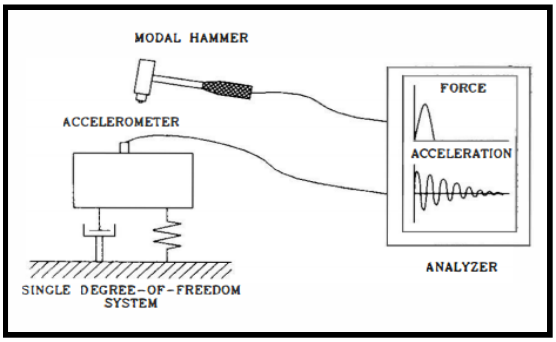
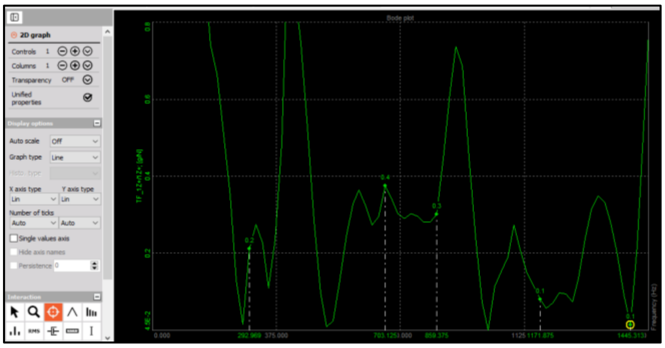


Fig 15: FFT construction

Fig. Experimental setup of FFT

EXPERMENTAL PROCEDURE

- Initially fixture is designed according to existing boundary condition as per FEA results.
- FFT 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 is 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.



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

Fig. FFT plot of optimized lower control arm

Table. Comparison of modified muffler FEA and FFT results

NATURAL FREQUENCY (Hz) MODE SHAPE	**FEA**	**EXPERIMENTAL**
1	297.45	292.96
2	704.17	703.12
3	871.46	859.37
4	1175.9	1171.87
5	1462.8	1445.30

CONCLUSION

- In present research existing lower control is optimized 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 optimized weight is around 1.65 kg.
- It is observed that in harmonic analysis frequency response along x and y axis have been reduced in optimized design compared to existing design.
- Experimental FFT analysis natural frequency are nearly identical with numerically obtained analysis.

REFERENCES

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. DattatrayKothawade, "Analysis of AControl 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. PrashantGunjan, "Optimization Approach of Front AWishbone Suspension Arm" IJARIIE, Vol. 4, Issue 2, 2018, PP 2395-4396.

6. Mr. BalasahebGadade, "Design, analysis of A-type front Asuspension 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 AControl 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 AArm 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 AControl Arm of LCV Using Radioss Linear", Research gate, April 2012, PP 121-125. SiddharthSaha, "STATIC STRUCTURAL ANALYSIS OF SUSPENSION ARM USING FINITE ELEMENT METHOD" International Journal of Research in Engineering and Technology, PP 402-406.

Author Details

1	***Name***	***SudaRshan Dilip Bhalshankar***
2	***Educational Qualification***	***1. M.B.A [HR.]* 2. M.E. [CAD/CAM] 3. B.E. [Mechanical Engineering] 4. D.A.E [Diploma in Automobile Engineering]***
3	***Mail***	***Sudarshanbhalshankar111@gmail.com***
4	***Address***	***Solapur, Maharashtra, India.***
5	***Pin code***	***413005***
6	***Founder Of***	***PradnyaSurya Engineering Works. Pvt Ltd.***
7	***Company Mail***	***pradnyasuryaengineerinworks@gmail.com***