# Design And Failure Analysis of Front lower suspension Arm System of Car

**Tilottama Chaudhari[1], Navneet K. Patil [2]**
Department of Mechanical Engineering
[1]PG student, S.S.B.T's, C.O.E.T., Bambhori, Jalgaon
[2]Associate Professor , S.S.B.T's, C.O.E.T., Bambhori, Jalgaon

***Abstract****-Front lower suspension arm is major component in Automobile Industries. The major function of arm is to maintain smooth suspension condition. Two types of lower arm are use upper and lower. The loads will be acting more on lower arm than upper arm because of its position. In this thesis the 3D model was generated in Catia software and Analysis conducted by the ansys software.*

***Keywords***-Solar energy, solar tracker, solar still, Distillation, re-newable energy.

## I. INTRODUCTION

The primary function of the suspension system of the vehicle should fulfil pretentious requirements about stability, safety and manoeuvrability. The suspension system of the vehicle performs multiple tasks such as maintaining the contact between tires and road surface, providing the vehicle stability, protecting the vehicle chassis of the shocks excited from the uneven road surfaces, etc. 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. The general function of control arms is to keep the wheels of a motor vehicle from uncontrollably swerving when the roadconditions are not smooth. The control arm suspension normally consists of upper and lower arms. The upper andlower control arms have different structures based on the model and purpose of the vehicle.Uneven tyre wear, suspension noise or misalignment, steering wheel shimmy or vibrations are the main causes of the failure of the lower 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 lower suspension arm of light commercial vehicle using FEM.

## II. LITERATURE REVIEW

Mohd Khairil Azirul Bin Khairolazar "This project presents the development of robust design of lower suspension arm using stochastic optimization". The strength of the design analyse by finite element software. The structural model of the lower suspension arm was mode by using the solid works. The finite element model and analysis were performed utilizing the finite element analysis code. The linear elastic analysis was performed using NASTRAN codes. TET10 and TET4 mesh has been used in the stress analysis and the highest Von Mises stress of TET10 has been selected for the robust design parameter. The development of robust design was carried out using the Monte Carlo approach, which all the optimization parameter for the design has been optimized in robust design software. The improvements from the Stochastic Design Improvement (SDI) are obtained. The design capability to endure more pressure with lower predicted stress is identified through the SDI process. A lower density and modulus of elasticity of material can be reconsidered in order to optimize the design.The area of the design that can be altered for the optimization and modification is identified through the stress analysis result. As a conclusion, the robust design by using stochastic optimization was capable to optimize the lower arm suspension. Thus, all the result from this project can be used as guideline before developing the prototype [3].

Veloso, H.S. Magalhães, G.I. Bicalho, E.S.Palma, in this study, a failure analysis of a longitudinal stringer of a prototype vehicle has been carried out. Failure took place at the bumpers fixation points of the vehicle suspension during durability tests. Crack was created and has grown causing fracture of the component. Stress analysis was performed using finite element method. A reinforcement model to solve the problem was proposed. Experimental quasi-static and durability tests were carried out and failures were no longer observed.

Mr. Sushilkumar P. Taksande, Dr. A.V. Vanalkar " Design and Analysis of car Front Lower Suspension Arm" This paper presents design and modelling of car front suspension lower arm to study the stress condition and to increase thickness of lower plate of the front suspension lower arm. The main objectives of this study to determine critical locations and strain distributions of the component. The paper aims to complete Finite Element Analysis of the front suspension lower arm which consist the stress optimization loadings and analysis for deformation.

Eshan Ayyar, Issac de souza, Aditya pravin, Sanket Tambe, Aqeem Siddiqui & Nitin Gurav "Selection Modification Analysis of Suspension System for All Terrain Vehicles" The real pleasure of driving for an off-road enthusiast can be described as the thrill of the terrain coupled with a capable machine to handle the terrain. However, this pleasure can be derived only when the comfort level of the driver is maintained. Thus, it is concluded that the suspension system (which is responsible for providing a comfortable ride quality to the driver) is one of the most important sub-systems to be designed. This paper aims at selecting, modifying, analysing and fabricating a suspension system capable of handling rough terrains while maintaining the ride quality [21].

Dattatray Kothawale, Dr. Y. R. Kharde "Analysis of Lower Control Arm in Front Suspension System Using F.E.A. Approach" This paper deals with finite element analysis for MacPherson type suspension system lower control arm (LCA) of 4W suspension system. The main function of the lower control arm is to manage the motion of the wheels & keep it relative to the body of the vehicle. The main significance of the analysis is to check the structural strength of LCA using dynamic forces. It will going to save the testing as well as validation cost. Also, validating final finite element analysis results through the physical testing of the component. This paper will show the validation of finite element analysis results with actual physical sample testing [16].

Jagwinder Singh, Siddhartha Saha "Static Structural Analysis of Suspension Arm Using Finite Element Method" In this study control arm was reverse engineered. Reverse engineering refers to the process of obtaining a CAD model from an existing physical part. CAD model was prepared using CATIA v5 software and finite element analysis was done using ANSYS 14.5 software by importing the parasolid file to ANSYS. The model is subjected to loading and boundary conditions and then analyzed using the FEA techniques. 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. In this case maximum von-misses stress is 211 MPa which is below the yield strength of the material [22].

## III. SYSTEM DEVELOPMENT

3.1 Mathematical modelling:

We can define displacement in three dimensions as,

$$u = [u \quad v \quad w]^T \qquad \text{Eq. 1}$$

Where $u$, $v$ and $w$ are displacements in x, y and z directions respectively. Thus stresses and strains are given by,

$$\sigma = [\sigma_x \quad \sigma_y \quad \sigma_z \tau_{yz} \quad \tau_{xz} \quad \tau_{xy}]^T \qquad \text{Eq. 2}$$

$$\in = [\in_x \quad \in_y \quad \in_z \gamma_{yz} \quad \gamma_{xz} \quad \gamma_{xy}]^T \qquad \text{Eq. 3}$$

Where,

$\sigma_i$= Normal Stresses in respective directions.

$\tau_{ij}$ = Shear stresses in plane made by $i$ and $j$ axes.

$\in_i$ = Normal Strains in respective directions.

$\gamma_{ij}$ = Shear Strains in plane made by $i$ and $j$ axes.

The stress-strain relationship is given by,

$$\sigma = D\epsilon \qquad \text{Eq. 4}$$

Where D is (6×6) symmetric matrix. For isotropic materials, D is given by,

$$D = \frac{E}{(1+v)(1-2v)}\begin{bmatrix} 1-v & v & v & 0 & 0 & 0 \\ v & 1-v & v & 0 & 0 & 0 \\ v & v & 1-v & 0 & 0 & 0 \\ 0 & 0 & 0 & 0.5-v & 0 & 0 \\ 0 & 0 & 0 & 0 & 0.5-v & 0 \\ 0 & 0 & 0 & 0 & 0 & 0.5-v \end{bmatrix} \qquad \text{Eq. 5}$$

The strain displacement relation is given by,

$$\in = \left[\frac{\partial u}{\partial x}, \frac{\partial v}{\partial y}, \frac{\partial w}{\partial z}\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}, \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}, \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right]^T \qquad \text{Eq. 6}$$

By considering tetrahedral element the nodal displacements q can be found out using potential energy approach, thus equating with finite element relations the elemental stresses can be found out as,

The force vectors are given by,

$$F = [F_x \quad F_y \quad F_z]^T \qquad \text{Eq. 7}$$

The invariants of the stress tensor are,

$$I_1 = \sigma_x + \sigma_y + \sigma_z$$

$$I_2 = \sigma_x\sigma_y + \sigma_y\sigma_z + \sigma_x\sigma_z - {\tau_{yz}}^2 - {\tau_{xz}}^2 - {\tau_{xy}}^2$$

$$I_3 = \sigma_x\sigma_y\sigma_z + 2\tau_{yz}\tau_{xz}\tau_{xy} - \sigma_x{\tau_{yz}}^2 - \sigma_y{\tau_{xz}}^2 - \sigma_z{\tau_{xy}}^2 \qquad \text{Eq. 8}$$

We define,

$$a = \frac{{I_1}^2}{3} - I_2$$

$$b = -2\left(\frac{I_1}{3}\right)^3 + \frac{I_1 I_2}{3} - I_3$$

$$c = 2\sqrt{\frac{a}{3}}$$

$$\theta = \frac{1}{3}\cos^{-1}\left(-\frac{3b}{ac}\right) \qquad \text{Eq. 9}$$

Thus principle stresses are given by,

$$\sigma_1 = \frac{I_1}{3} + c\cos\theta$$

$$\sigma_2 = \frac{I_1}{3} + c\cos\left(\theta + \frac{2\pi}{3}\right)$$

$$\sigma_2 = \frac{I_1}{3} + c\cos\left(\theta + \frac{4\pi}{3}\right) \quad \text{Eq. 10}$$

Assuming 0.003 mm strain in all directions then stress become from eq. 4 we get,

$$\sigma = \begin{bmatrix} 1500 \\ 1500 \\ 1500 \\ 230.8 \\ 230.8 \\ 230.8 \end{bmatrix} \text{N/mm}^2 \qquad \text{Eq. 11}$$

So, working stress is given by,

$$Working\ Stress = \frac{Yield\ Stress}{Factor\ of\ Safety} \text{Eq. 12}$$

$$working\ Stress = \frac{1500}{2.5} = 600 \text{ MPa.}$$

Thus for the safety of arm von miss stress induces in arm should be less than 600 MPa.

3.2 Design of Suspension Arm : Thework presented here is the design and analysis of suspension arm. For this purpose, the models are created in CATIA. The various commands stated in above sections are used to generate the design. Firstly the arm is taken from market survey. The various dimensions are measured by using standard gauge vernier and micrometer gauges. The dimensions are put in CATIA software to generate the 3D model of arm. Here three designs are adopted called as models. Model 1 is general model used in today's automobiles. While the model 2, 3 and 4 are slightly change in design by adopting fillets in critical areas. The models are shown in fig.

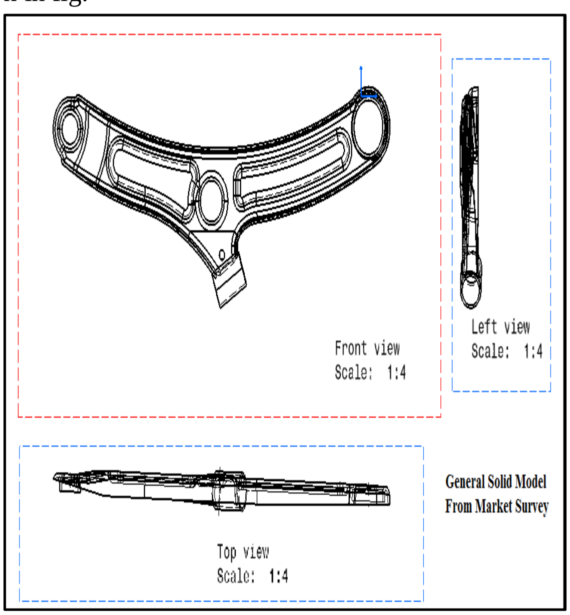


Fig. 3.2.1 Design of Solid Model 1 (General model from market survey)

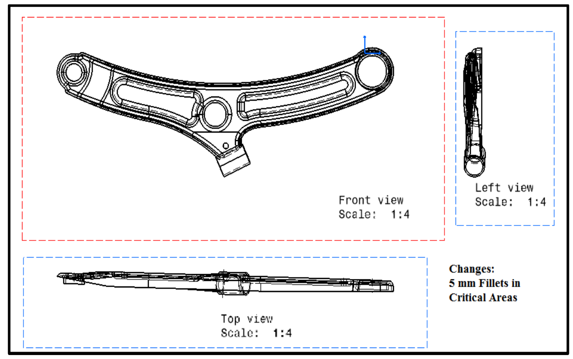


Fig. 3.2.2 Design of Solid Model 2 (5 mm Fillets in Critical Areas)

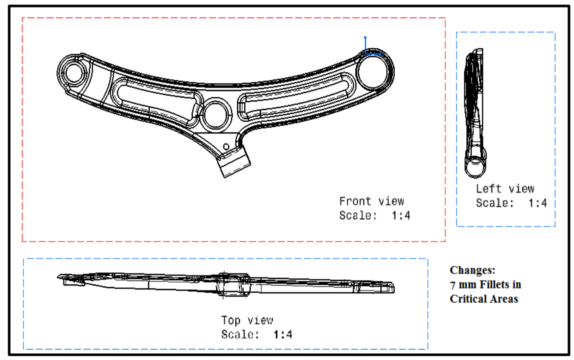


Fig. 3.2.3 Design of Solid Model 3 (7 mm Fillets in Critical Areas)

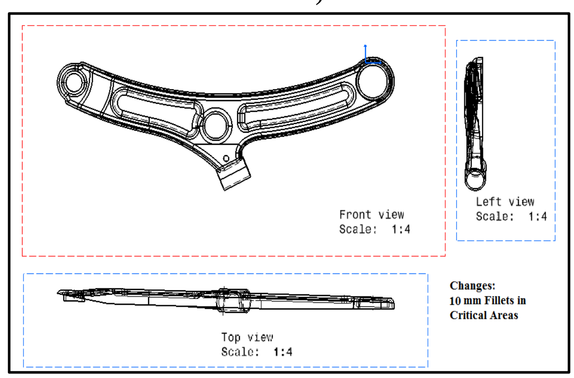


Fig. 3.2.4 Design of Solid Model 4 (10 mm Fillets in Critical Areas)

## IV. STATIC AND FATIGUE ANALYSIS OF ARM

Steps for generating FEA of arm are as follows.

1. Firstly convert the model in IGES or STP format.
2. Select static structure analysis from analysis menu.
3. Apply materials and properties of materials.
4. Import the model in IGES or STP format.
5. Generate the mesh of tetrahedral (elements around 31000 and nodes around 52000).
6. Apply boundary conditions (fixed at two supports and Load of 1000 N at one support)
7. Check ANSYS settings for static structure and fatigue.
8. Solve the model for solution.

9. Insert the fatigue tool in ANSYS setting. (Select the fatigue strength factor=1, type of loading is fully reversed and theory of fatigue is mean stress theory)
10. Evaluate all the results. (Stress, strain, deformation, stress-life curve, etc.)

All the models are tested in ANSYS 15.0 workbench. The results from the ANSYS are shown in Fig. below:

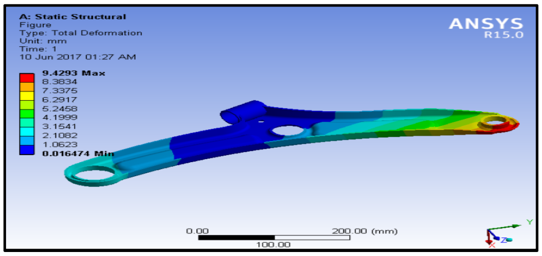

Fig. 4.1 Total Deformation of Model 1

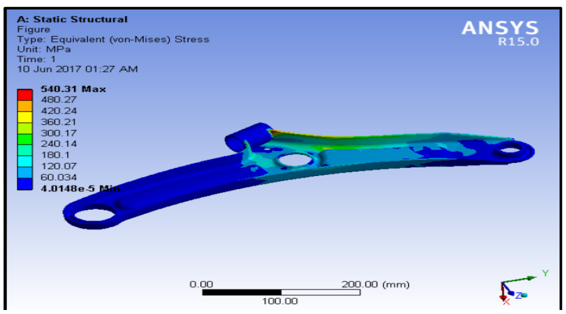

Fig. 4.2 Von Miss Stress for Model 1

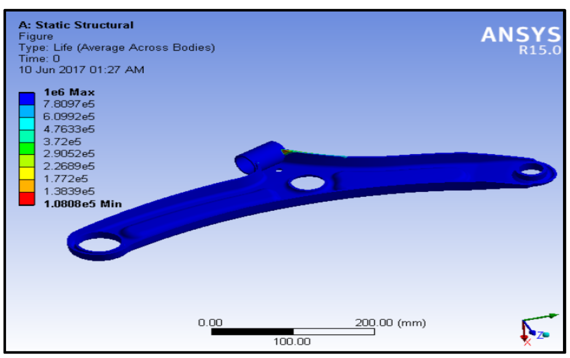

Fig. 4.5 Life of Model 1

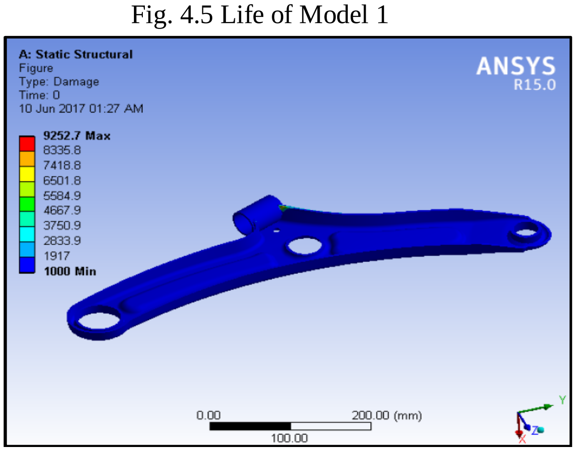

Fig. 4.6 Damage for Model 1

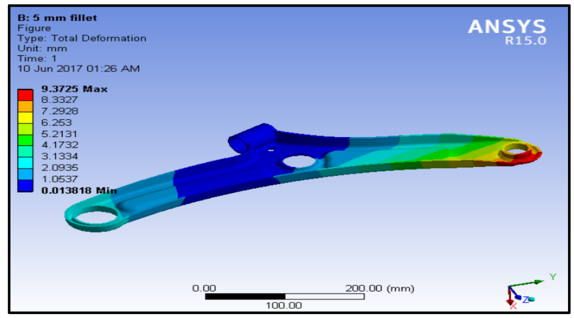

Fig. 4.7 Total Deformation for Model 2

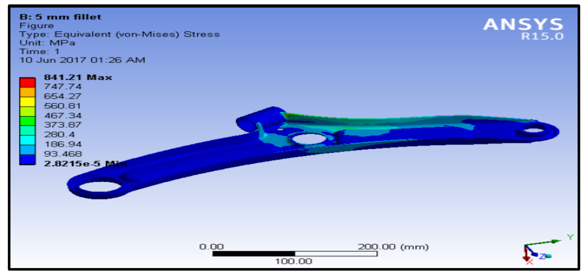

Fig. 4.8 Von Miss Stress for Model 2

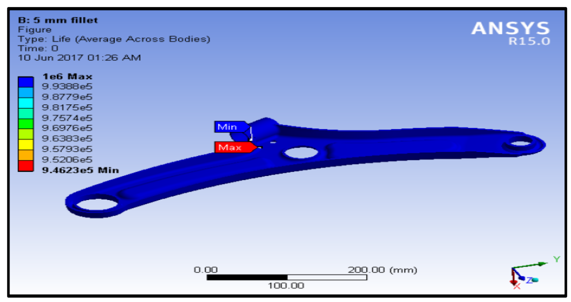

Fig. 4.9 Life of Model 2

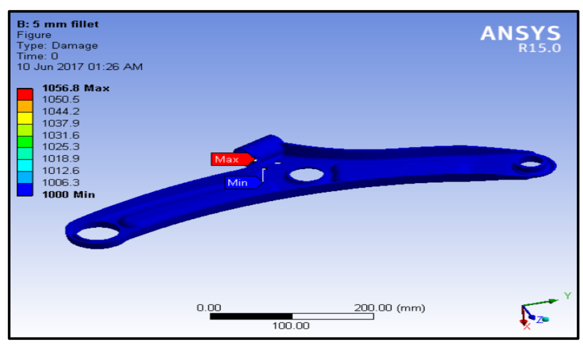

Fig. 4.10 Damage for Model 2

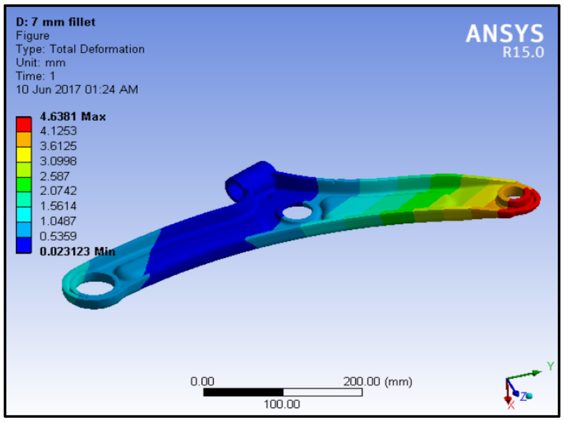

Fig. 4.11 Total Deformation for Model 3

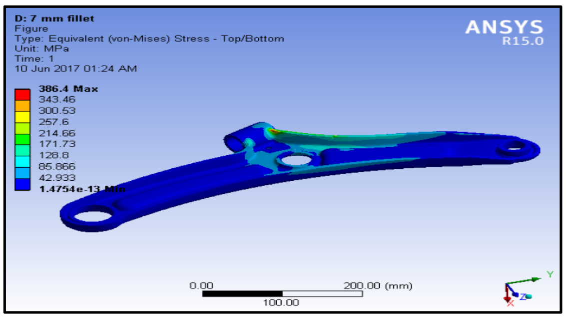

Fig. 4.12 Von Miss Stress for Model 3

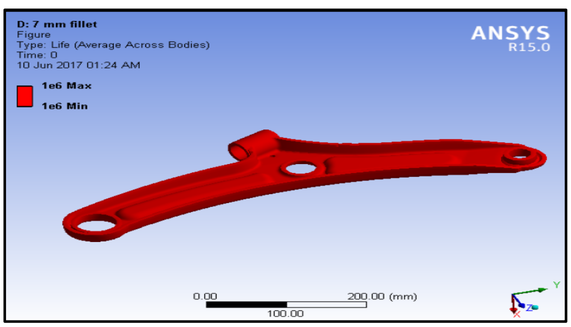

Fig. 4.13 Life of Model 3

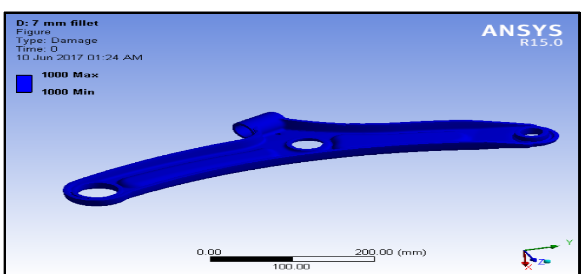

Fig. 4.14 Damage for Model 3

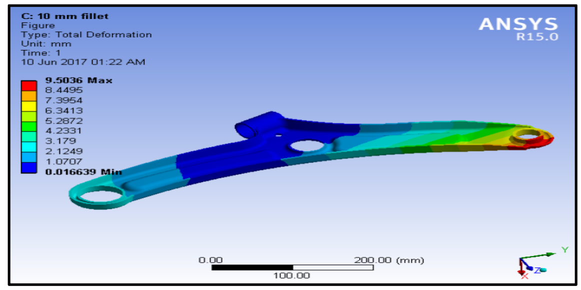

Fig. 4.15 Total Deformation for Model 4

## V. RESULTS AND DISCUSSION

The presented study shows that the static as well as fatigue analysis is important for automobile parts. Since the static study shows the design of arm under the load and fatigue study shows the life of the arm under the repeating stresses due to given load. The comparison has been made between the various model types by considering various parameters. The fig. 5.1 shows the comparison of total deformation for various models. The study shows the model 1, 2 and 4 having total deformation around 9 mm analytically as well as by ANSYS. The model 3 has different deformation among these models. So, as per the deformation point of view model 3 is not optimum model. The stress is also one of the important parameter for the design. In 3D element, generally von miss stress is used for analysis. The fig. 5.2 shows the model 1 has nearly equal value of stress by analytical and ANSYS (540MPa). Thus model 1 is optimum for stress. The fig. 5.3 and 5.4 shows the fatigue tool results of all the types of models. The average fatigue life of all the models is 1000000 cycles. The damage is the property which shows the failure of the arm before design life. Thus, from the study of these models the model 1 is the best optimum for all the parameters. The model 1 having deformation 9 mm, stress 540 MPa, fatigue life 1000000 cycles and damage 9250 units.

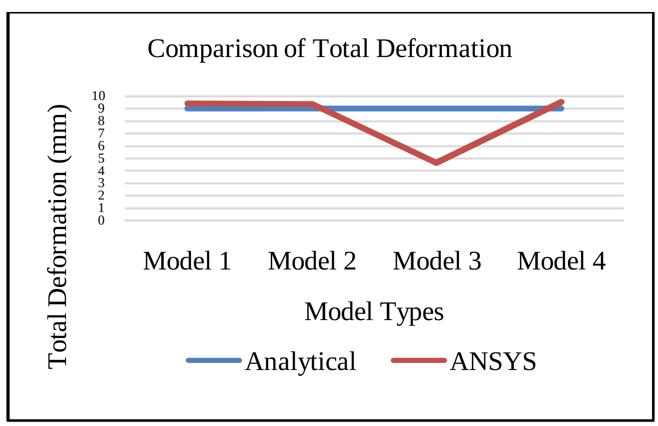

Fig. 5.1 Comparison of Total Deformation for Model Types

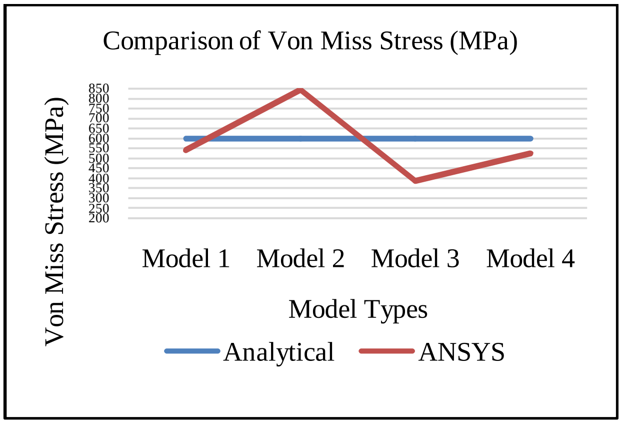


Fig. 5.2 Comparison of Von Miss Stress for Model Types

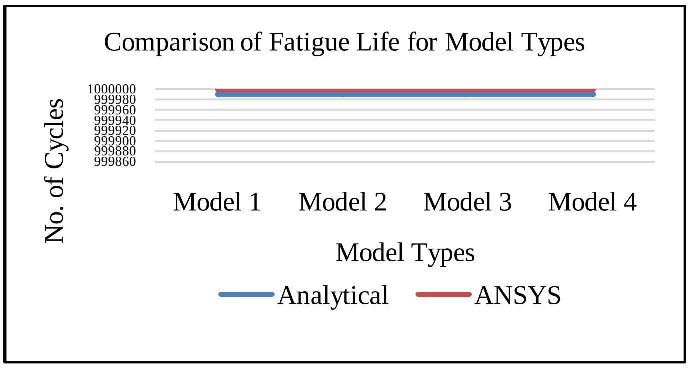


Fig. 5.3 Comparison of Fatigue Life for Model Types

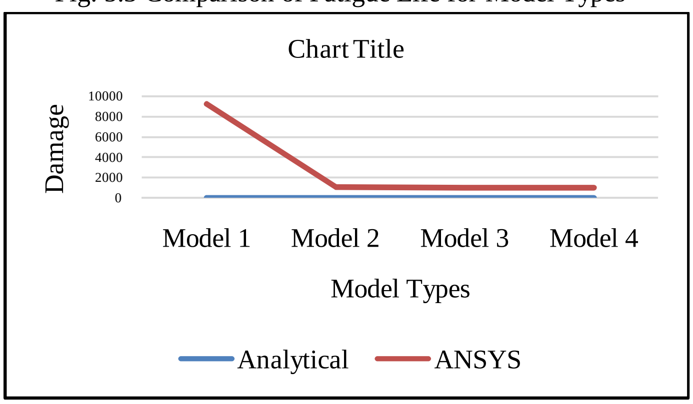


Fig. 5.4 Comparison of Damage for Model Types

## V. CONCLUSION

The work presented here is to optimize the design of a lower suspension arm. The design of arm is subjected to various parameters like deformation, von miss stress, fatigue life and damage etc. The literature has been studied for the design of arm and these parameters has found out by using existed methods of analytical and finite element analysis. The study draws following conclusions.

1. The design of lower suspension arm system has been studied successfully.
2. The literature on the suspension arm, finite element analysis has been done successfully.
3. The model has been made in CATIA and analysed in ANSYS for static as well as fatigue life.
4. The results are nearly equal by analytical as well as by ANSYS.
5. The model 1 is optimum model for design from the study.

## ACKNOWLEGMENT

I would like to thank you my respected guide Er. N.K.Patil Sir for providing the guidance, and support & valuable time to me for preparation of this paper work.

## REFERENCES

[1] Hemin M.Mohyaldeen University Malaysia Pahang
[2] Lihui Zhao College Of Mechanical Engineering, University Of Shanghai For Science And Technology, Shanghai 200093, Pr China.
[3] Mohd Khairil Azirul Bin Khairolazar Faculty of Mechanical Engineering University Malaysia Pahang
[4] Kyrre, S.A. 2006. Fatigue life prediction of an alluminium alloy automotive component using finite element analysis of surface topography. PhD Thesis Dissertation. Norwegian University of Science and Technology Department of Structural Engineering.
[5] Kyrre, S.A., Skallerud, B., Tveiten, W.T. and Holme, B. 2005. Fatigue life prediction of machined components using finite element analysis of surface topography, International Journal of Fatigue. 27: 1590- 1596.
[6] Nadot, Y. and Denier, V. 2003. Fatigue failure of suspension arm: experimental analysis and multiaxial criterion. International journal of Fatigue, 11 (4): 485 – 499.
[7] Smith, R. A., 1999, Fatigue in transport: Background, Solutions and Problems, Fatigue '99, Proceedings of the Seventh International Fatigue Congress, edited by Wu, X. R., and Wang, Z. G., Beijing, P.R. China, pp 2583-2590.
[8] Rill, G., 2006, "Vehicle Modeling by Subsystems", Journal of the Brazilian Society of Mechanical Sciences and Engineering, Vol. 28, No. 4.
[9] Shirahatt, A., Prasad, P.S.S., Panzade, P., Kulkarni, M.M., 2008, "Optimal Design of Passenger Car Suspension for Ride and Road Holding". Journal of the Brazilian Society of Mechanical Sciences and Engineering, Vol. 30, No. 1.
[10] Wong, J.Y., 2001, "Theory of Ground Vehicles", John Wiley & Sons Inc., New York, USA.
[11] Goldberg D.E., "Genetic Algorithms in Search, Optimization, and Mechanical Learning", Addison Wesley, 1989.
[12] Myers R. H., Montgomery D. C., "Response Surface Methodology, Process and Product Optimization Using

Designed Experiments", Wiley Series in Probability and Statistics, 1995.

[13] Finite element analysis of surface topography, International Journal of Fatigue. 27: 1590- 1596.

[14] Sarvadnya Ajinkya Thakare, Prasad C Antapurkar, Divyaj S Shah, P R Dhamangaonkar and S N Sapali "Design and Analysis of Modified Front Double Wishbone Suspension for a Three Wheel Hybrid Vehicle." Proceedings of the World Congress on Engineering 2015 Vol II WCE 2015, July 1 - 3, 2015.

[15] Mr. Prashanthasamy R.M.T, Dr. Sathisha, Mr. Imran Ali M.R & Mr. Jnanesh. K "Design and Modal Analysis of Lower Wishbone Suspension Arm Using FE Approach" Imperial Journal of Interdisciplinary Research (IJIR) Vol-2, Issue-9, 2016.

[16] Dattatray Kothawale, Dr. Y. R. Kharde "Analysis of Lower Control Arm in Front Suspension System Using F.E.A. Approach" International Journal of Engineering Research and Development Volume 5, Issue 12 (February 2013).

[17] Pratik S. Awati, Prof. L.M.Judulkar "Modal and Stress Analysis of Lower Wishbone Arm Along With Topology" International Journal of Application or Innovation in Engineering & Management (IJAIEM) Volume 3, Issue 5, May 2014.

[18] Y. Nadota, V.Denierb, "Fatigue failure of suspension arm: experimental analysis and multiaxial criterion, UMR CNRS No. 6617, LMPM.

[19] "Kumar, Nithin, RK, Veeresha "Analysis of Front Suspension Lower Control Arm of an Automobile Vehicle" Visvesvaraya Technological University, Department of Mechanical Engineering, NMAMIT, Nitte 547110.

[20] Mr.Sushilkumar P. Taksande, Dr.A.V.Vanalkar "Design and Modelling of car frond Lower Suspension Arm" IJIRST –International Journal for Innovative Research in Science & Technology| Volume 2 | Issue 02 | July 2015.

[21] Eshan Ayyar,Issac de souza, Aditya pravin, Sanket Tambe, Aqeem Siddiqui &Nitin Gurav " Selection Modification Analysis of Suspension System for All Terrain Vehical" ISSN : 2319 – 3182, Volume-2, Issue-4, 2013.

[22] Jagwinder Singh, Siddhartha Saha "Static Structural Analysis of Suspension Arm Using Finite Element Method" Ijret: International Journal Of Research In Engineering And Technology.

[23] Mr. Balasaheb Gadade, Dr. R.G.Todka "Design, analysis of A-type front lower suspension arm in Commercial vehicle" International Research Journal of Engineering and Technology (IRJET) Volume: 02 Issue: 07 | Oct-2015