Optimization Approach of Front Lower Wishbone Suspension Arm

Prashant Gunjan[1], Amit Sarda[2]

[12]Department of Mechanical Engineering, Christian College of Engineering and Technology, Bhilai

ABSTRACT

Suspension systems have been widely applied to vehicles, from the horse-drawn carriage with flexible leaf springs fixed in the four corners, to the modern automobile with complex control algorithms. The suspension of a road vehicle is usually designed with two objectives to isolate the vehicle body from road irregularities and to maintain contact of the wheels with the roadway. The suspension link allows wheels to rise and fall on their own without affecting the opposite wheel. In this case, the wheels are either not connected at all or are connected through universal joints with a swing axle. Suspensions with other devices, such as anti-roll bar that link the wheels in some way are still classed as independent suspension link. Suspension arm is one of the main components in the suspension systems. 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. This paper presents design, modelling and analysis of car front suspension lower arm to study the stress condition and to select the suitable materials for 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.

Keywords: *Front lower wishbone arm, Suspension systems, Topological Optimization etc.*

1. 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. This system is the mechanism that physically separates the vehicle body from the wheels of the vehicle. The suspension system will consider ideal if the vehicle body isolate from uneven road and inertial disturbances associated during situation of cornering, braking and acceleration.

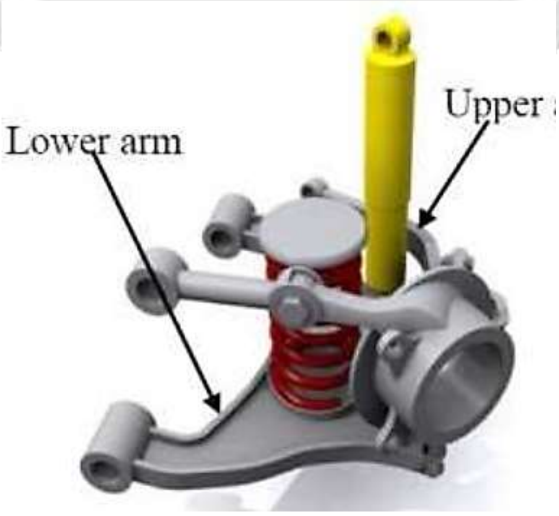

Fig 1. Multi-link suspension arm

Types of the suspension system

The suspension system is always derived by some mechanical way. The designs of the suspension systems are classification in two main groups
(a) Dependent suspension system (solid axle) and
(b) Independent suspension system.

2. LITERATURE REVIEW

In this Chapter focuses on the literatures on suspension system used in vehicles.

Mahishi (2007), this study was a part of a series that focused about the types of failure in materials; this article discussed about metallurgical failures. Failures are always caused by human's errors, according to this, errors were classified into three main categories, errors of knowledge, performance and intent. Finite element analysis is a very powerful tool in analysing mechanical structures because it deals with the whole structure as a huge number of segments on each a mathematical solution is performed in order to know the state of each element of them, then they are summarized together to give the whole status of the structure.

Kang et. al., (2007), Fracture mechanics science is involved in putting the failures due to fracture and cracks into a mathematical frame, so it can be understood and explained; it related the defect size and the crack's principal initiation to the subjected stresses and the fracture toughness of the material. The size of the defect that the material can handle without any further growth can be calculated from the equation: this equation will allow for determination of the allowable flaw size, minimum stress required for sudden failure, applied load when failure occurs, determination of materials used in manufacturing and decide if the design of a certain component was satisfactory or not.

Nawar and Asady (2009), Finite Elements Analysis (FEA) is the mathematical representation of complex engineering problems to obtain a unique solution for each segment of the system; it can be used to obtain structural, heat transfer, static and various other engineering systems solutions. FEA is based on approximating the mechanical and physical behaviour of each tiny segment of the system by a mathematical representation. This paper dealt about simple FEA for a truss (rod).

Sritharan (2005), this study was conducted to compare the results obtained by the FEA and the experimental for an automotive suspension part. The part was a lower suspension arm for a 2000cc sedan car. Variable stress, strain and fatigue tests were performed on the part to obtain the critical point's location, loading and part life prediction. The strain distribution obtained from experimental results was found to be compatible with the FEA relative to the complexity of the geometry of the part, but onthe other hand, the data collected from the road test was very different from that obtained by FEA.

Noor and Rahman (2009), FEA analysis must be compared with experimental data to validate it to ensure its accuracy in order to maximize their advantages in the future, Strain analysis data were reasonably correct by the FE model and helped researchers to identify the exact position where the strain gauges must be fixed, Component (part) data should be collected experimentally, Even though fatigue damage occurred at low cycles range, it was justified by the great number of the cycles.

Roy (2001), This paper dealt about the importance of employing computerized FEA, technology and techniques in the automotive industry. It presented examples of how the results obtained by FEA software were compatible with these collected from expensive, time consuming experiments. The paper also presented a model of a structural design iteration process that is cost effective and can provide a huge useful data.

Kothawale and Kharde (2003), this paper describes analysis of lower suspension arm using F.E.A. Approach. This paper was preparing CAD Model using PRO-E Software & finite element analysis using Ansys software. The important significance of analysis is to check the structural strength of LCA using dynamic forces. The aim of this paper analysis is to show the how finite element analysis is helping in complete product development cycle. Because it going to saves lot of cost, as every vehicle having generally 3-4 stages in complete product development cycle, stages are Proto-I, Aplha-II, Gamma-III & Beta-IV. This paper was show the validation of finite element analysis results with actual physical sample testing. In this Study they concluded that a how much the results by using analysis software as well as physical testing of model are similar or not.

Gadade and Todkar (2015), this paper describes Design, analysis of A-type front lower suspension arm in commercial vehicle. The main objective of this study was to calculate working life of the component under static loading. The A-type lower suspension arm was developed by using CAD software. Actual model was manufacture as per design by using AISI 1040 material. This paper result was this model imported in the hyper mesh. After meshing apply load on hub bush they found the weaker section in the model but it is require validating the FEA results with actual experimental test. To validate the FEA results we create a typical A-type suspension arm of same material of AISI 1040.as per the consideration in FEA we constraint two bushes and load apply on remaining wheel hub bush on universal testing machine. The load goes on increasing and at last they found the stress in material at maximum stressed area.

Prashanthasamy et. al. (2016), this paper describes Design and analysis of lower wishbone suspension arm using FE approach. In this study is made on existing design with aluminium alloy. The 3D model was generated by Hyper Mesh and the Static and dynamic analysis was conducted by Abaques.3d Cad Model of the design created and FE model of the design was created. Analysis under Static and modal condition was done. Based on the result obtained in analysis the design was optimized in different stage. They concluded that the stresses and deformation for the existing design with aluminium alloy is almost maximum compare to AISI 1040.New Design was developed to reduce stress and deformation existing in the current design with aluminium alloy. Finally, they were concluded that from the FE analysis the new design 1 and new design 2 can be replaced with aluminium alloy existing design with AISI 1040 for Wishbone Suspension System.

Patil et. al. (2013), this paper describes Experimental & Finite Element Analysis of Left Side Lower Wishbone Arm of Independent Suspension System. Under the static load conditions deflection and stresses of steel lower wishbone arm and composite lower wishbone arm are found with the great difference. Carbon fibre suspension control arms that meet the same static requirements of the steel ones they replace. Deflection of Composite lower wishbone arm is high as compared to steel lower wishbone arm with the same loading condition. The redesigned suspension arms achieve an average weight saving of 27% with respect to the baseline steel arms. The natural frequency of composite material lower wishbone arm is higher than steel wishbone arm.

3. CONCLUSION

Due to the various functions of control arm is very important parts in suspension. As vehicle passes through bump, speed breaker etc. differing types of forces functioning on the wheels that are transmits to regulate arm via attachments i.e. ball joint assembly etc. to the wheel. These force and force values are mention within the load case.

In this study the existing component design and its function for identifying potential areas for modification and secure geometry and import the same over the pre-processor for Discretization. Upon finding results for structural analysis, use the inputs for pursuing mass optimization Recommend the new design for implementation.

REFERENCES

[1] Dr. J. Mahishi, Nonlinear static and analysis of automotive lower control arm, Ms&M Engineering Inc, Farmington Hills, MI, USA.
[2] Kang, B., Sin, H.-C., & Kim, J., Optimal shape design of lower control arm considering dynamics effect. International Journal of Automotive Technology, Vol. 8, Issue 3, 2007.
[3] Nawar A. Al-Asady, “Automobile body reinforcement by finite element optimization”, Department of mechanical Engineering. UKM, Malasiys, 2009.
[4] Sritharan. G, Deign and mass optimization of independent multi-link suspension for ride performance GM India Pvt, ltd., ITPB, Whitefield Road, Bangalore.
[5] IiM. M. Noor and M. M. Rahman, Fatigue Life Prediction of Lower Suspension Arm Using strain-life approach, © Euro Journals Publishing, Inc., 2009.
[6] Roy, R. K., “Design of Experiments Using the Taguchi Approach”, United States of America, New York, John Wiley & Sons, Inc., 2001.
[7] 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, 2003.

[8] Mr. Balasaheb Gadade, Dr. R.G. Todkar, "Design and Analysis of A- type Front Lower Suspension arm in Commercial vehicle" International Research Journal of engineering and Technology (IRJET) Volume 2 Issue 7oct 2015.
[9] Mr Prashanthasamy R.M.T, Dr. Sathisha, Mr. Jnanesh k. "Design and Model Analysis of Lower Wishbone Suspension Arm Using F.E. Approach" Imperical journal Of Interdisciplinary Research (IJIR) Volume 2 Issue-9, 2016.
[10] Patik S. Awati , Prof. L. M. Judulkar " Model and Analysis of Lower Wishbone Arm Along with Topology" International Journal Application Of Innovation in Engineering & management" Volume-3 Issue-5 May 2014.