International Journal of Engineering Research and Development
e-ISSN: 2278-067X, **p**-ISSN: 2278-800X, www.ijerd.com
Volume 5, Issue 12 (February 2013), PP. 18-23

Analysis of Lower Control Arm in Front Suspension System Using F.E.A. Approach

Dattatray Kothawale[1], Dr. Y. R. Kharde[2]
Student M.E. (Design), Mechanical Engineering Department, Pravara Rural Engineering College, Loni – 13736, Tal – Rahata, Dist. – Ahmednagar.
M.E. (Mech) Phd, Mechanical Engineering Department, Pravara Rural Engineering College, Loni – 413736, Tal – Rahata, Dist. – Ahmednagar.

Abstract:- 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 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.
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.
The aim behind this project 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. By believing on the results of finite element analysis company / organization can skip one or two stages in between proto & final product. This paper will show the validation of finite element analysis results with actual physical sample testing.

Keywords:- Suspension System, Automobile Lower Control Arm, Suspension Bushes, Static & Dynamic FEA Analysis.

I. INTRODUCTION

This Control arm is one of the important components in suspension system. In double wishbone suspension system both upper and lower control arm used. But in McPherson strut suspension system only lower control arm is used. Control arms are located between wheel assembly and chassis of the vehicle. Control arm connect to the chassis by means of bush which are located in pivot points. In control arm front and rear bush are used. Control arm has a substantially U-shaped configuration with bushing apertures formed at the ends of the arm sand a ball joint receptacle formed at the apex of the control arm. The ball joint receptacle is adapted to cooperate with a ball joint assembly and may include a ball joint housing integrally formed with the control arm. Typical modern control arms incorporate a separate ball joint housing which is inserted into the apex of the control arm. The bushing apertures are designed to retain pipe bushings for mating engagement with a pivot bar assembly forming a portion of the vehicle suspension system. The pivot bar typically extends through both bushing apertures allowing the control arm to pivot about the assembly in response to road conditions affecting the vehicle suspension system. For the construction of control arm high strength alloy steel is used. Control arm consists of modulus section which is between the apex and pivot points. Control arm consists of two similar members. During normal loading bushing members keep the members spaced apart. This allows member to bend and flex when subjected to longitudinal forces. As transverse loading occurs two members abut one another. This abutment greatly increases strength of control arm.

Finite element analysis is widely used in numerical solution of many problems in engineering and technology. The Problem includes design of shafts, trusses bridges, buildings heating and ventilation, fluid flow, electric and magnetic fields and so on. The main advantage of using finite element analysis is that many alternatives designs can be tried out for their validity, safety and integrity using the computer, even before the first prototype is build. Finite element analysis uses the idea of dividing a large body into small parts called elements, connected at predefined points called as nodes. Element behaviour is approximated in terms of the nodal variables called degree of freedom. Elements are assembled with due considerations of loading and boundary condition. This results in finite number of equations. Solution of these equations represents the approximate behaviour of the problem.

Control arm first modelled in PRO-E Wildfire Software, it is excellent CAD software, which makes modeling so easy and user friendly. Then model is imported in ANSYS.ANSYS has an option to import models from other CAD software. A part is analyzed in ANSYS in three steps. First is pre-processing which involves modeling geometric clean up, element property definition and meshing. Next comes, solution which involves imposing boundary conditions and applying loads on the model and then solution runs. Next in sequence comes post processing, which involves analyzing the results plotting different parameters like stress, strain, von-mises stress and many.

II. PROBLEM DEFINATION AND METHODLOGY

Problem Definition:

Due to the different functions control arm is important part in components suspension system. As vehicle passes through bump, speed breaker etc different types of forces acting on the wheels which are transmits to control arm via attachments i.e. ball joint assembly etc. to the wheel. These force and torque values are mention in the load case. So in this analysis the main concern is to find out the maximum stress region and stress value in control arm and compare this value with tensile yield strength of material and results of experimental testing of control arm at the same load which are mentioned in load case of control arm.

Methodology:

To solve the problem mention above we have to use the method mentioned in the flow chart below. As shown in the flow chart we have first import the Para solid model to the ANSYS which is analysis software by using which we have to solve the problem. In the next step of analysis we have to edit the geometry which includes removal of areas etc.

In the next step we have to do the meshing by using two different elements for control arm body and bushes of control arm. In this analysis three different assemblies i.e. front and rear bush of the control arm and its body are connect to each other and points where we have to apply the loads are inside as well as outside of the body so to connect these points to their respective bodies and assemblies with each other we have to use contact pair and they are created by using contact manager. In the next step we have to apply loads i.e. force and torque which are mentioned in the load case then after we have to apply the boundary conditions by seeing the actual model i.e. where it is fixed. Then we have to solve the problem by using software.

After completion of the solution by using software we have to do the physical testing of control arm along with its bushes i.e. front and rear. After completion of physical testing we have to compare the result with result of analysis obtained by using analysis software. If results match with each other considering up to 10% error is negotiable then we say that problem is solved otherwise we have to do the changes in meshing, boundary conditions and then results are compared with each other.

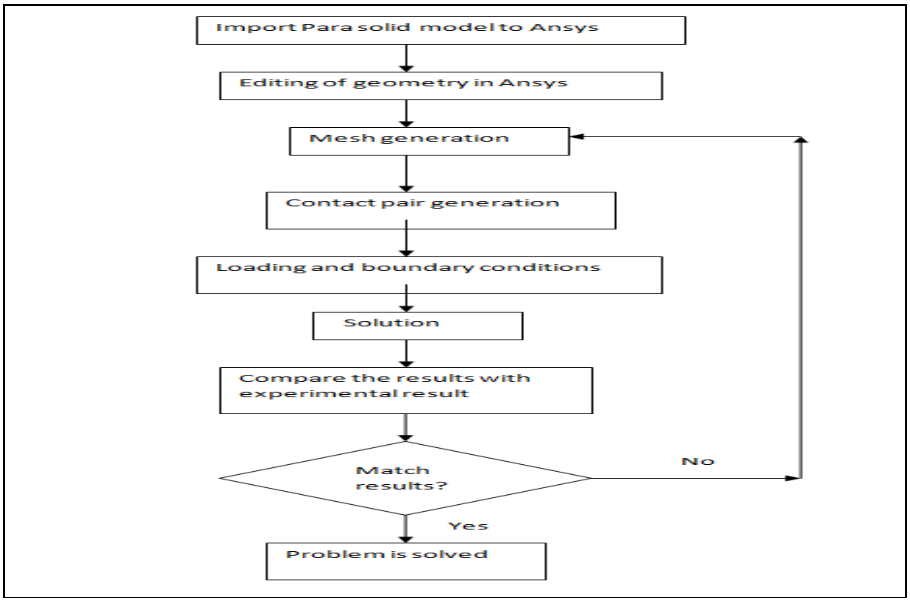

FIG 2.1: FLOW CHART OF PROBLEM METHODOLOGY

III. STATIC ANALYSIS

Introduction:

The Rear Bushing is assembled to the Lower Control Arm by pressing it into an axle, welded to the upper and lower LCA panels and to the Cross-member by pressing the bushing into a Cast Iron Housing.

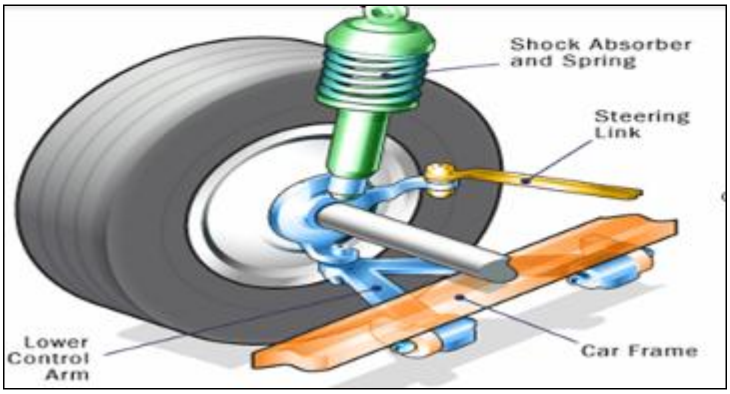

FIG 3.1: General Suspension System

This Housing is assembled to the Cross-member by two bolts. The Front Bushing is pressed into its housing and assembled to the Cross-member by a bolt and nut assembly

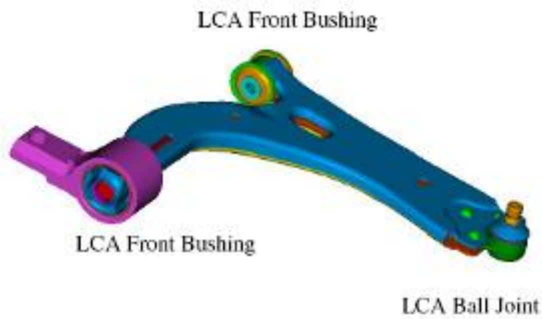

FIG 3.2 : McPherson Suspension Assembly

FIG 3.3 : Lower Control Arm

3D Model of Control Arm & Mesh Generation:

Control arm first modeled in PRO-E Wildfire software is excellent CAD software, which makes modeling so easy and user friendly. Then model is imported in ANSYS.ANSYS has an option to import the model from CAD software.

In this analysis mesh generation is auto mesh generation with element lengths edge is 5 mm. This element size is used for all the volumes except volume number 9 which is in the rear bush. By element size 5 it is not possible to mesh volume number 9 and due to which element size used is 3 mm.

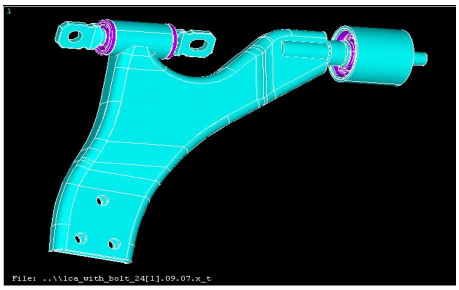

FIG 3.4 : 3D Model of Control Arm

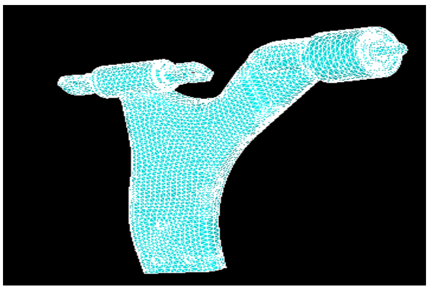

FIG 3.5 : Mesh Model

- NO. OF NODES = 1, 04,579
- NO. OF ELEMENT = 80321
- ELEMENT TYPE = (1) Solid 92 (10 node) (2) Solid 186(20 node)

IV. LOADING AND BOUNDARY CONDITIONS

Loading Data:

Loads are given in the load case which is mentioned above. Loads are in the form of force values and moment. These loads are applied on the points which are mentioned in the load case these points are inside the bushes and one is outside the body of control arm. Forces are applied on x, y & z direction.

Load Cases:

1. Braking moment load & Acceleration Load
2. Vertical Loading Coupled with Lateral Load at Spindle
3. Static Vertical Load Only
4. Curb Impact event, where steering angle of 22.5º

Check Directional deformation, Stress & Displacement for Lower Control Arm

This difference in the stress value occurs mainly due to use of different materials, use of different elements etc. In our analysis we use material such as structural steel FE 420 for control arm body, natural rubber for the bushings i.e. front as well as rear, but in the cedimento test material used is the structural steel FE 420 for control arm body and polymers for bushings. So due to differences in the material changes in results are observed in the results.

In our analysis we use SOLID 92 element for meshing of control arm body and SOLID 186 for meshing of bushings. In cedimento test they used element SHELL 181 for meshing of control arm body and SOLID 186 for the meshing of bush, so this is another reason for the difference in the stress distribution.

There are other reasons for difference in stress distribution are different loading conditions, different locations of points where loads are acting.

Boundary Condition for Model Analysis:

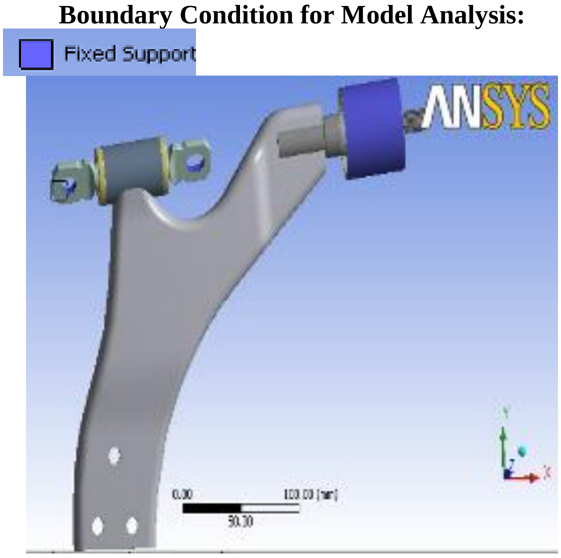

FIG 4.1 : Boundary conditions for modal analysis

V. PHYSICAL TESTING

Introduction:

Actual testing of control arm along with its front and rear bush is done to check whether results obtained by analysis are correct or not. In physical testing of control arm control arm and its bushes are test under same load conditions in which its analysis was done. For the analysis of control arm we use "Star testing machine -200 of BONSAI series".

This star testing machine consist of wide range of pneumatic or mechanical grips, accessories or fixtures, extensometers, environmental chambers and furnaces etc. as per test requirement. Products in this machine are tested by using standards like ASTM, DIN, BS, JIS, IS etc. Output of the machine is directly attached to the computer. As output of the machine is directly attached to the computer so by using computer we can easily enter the force and torque values. With the help of computer we see the resultant stress values as well as we also able to plot the graphs showing results.

Capacity of this machine is from 5 kgf to 5 tons so due to large variation in load capacities this machine is used for small as well as for large load values. This machine is used for the determination of tensile strength, compression, puncture strength, shear strength, adhesion strength, crush resistance, elastic limits, deformation strength, bend strength, ductility, coefficient of friction etc. So due to its variety of applications, such large load range it is widely used for the testing of different components. Machine showed in the below figure.

Fig 5.1: Star testing Machine (BONSAI series) – Ohio City, Maumee, U.S.

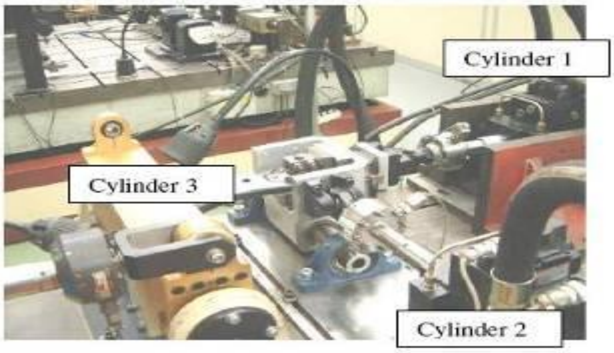

Fig 5.2: Rig test for the Lower Control Arm Assembly – Ohio City, Maumee, U.S**.**

As far as the loads applied at the Lower Control Arm are three orthogonal forces, the rig-test was designed to reproduce the phenomenon. Cylinder 1 (one) applies lateral load. Cylinder 2 (two) applies the longitudinal one. The third cylinder was used to represent the effect of the vertical load by its generated moment.

VI. CONCLUSION

Here in this case study of analysis of lower control arm, its front and rear bush of 4W model for static as well as for contact conditions using ANSYS software. For analysis uses the most popular technique of analysis FEA (Finite Element Analysis). After completion of complete load condition analysis we will review how much the results by using analysis software as well as physical testing of model are similar or not.

ACKNOWLEDGMENT

I would like to thank you my respected **Guide Dr. Y.R. Kharde** Sir & **HOD** (Head of Mechanical Engineering) **Prof. R. R. Kharde** Sir, for providing guidance, support & valuable time to me for preparation of this paper work.

REFERENCES

[1]. Miguel A.Eguia ,Minishiou Huang, and Tay Tyan, (NAT impact safety, Ford motor company), "Crashworthiness Simulation of Lower Control Arm Impact Tests", SAE TECHNICAL PAPER SERIES, 2005-01-0361, Page Nos. 2 - 10.

[2]. S. Hakan OKA, F. Basar Yalciner, Koray Gursu : Figes A.S / TURKEY, Zeki Yalin, Durmus Buyuktuncel: YPS A.S/ TURKEY, "Simulation of Cedimento Test of a Suspension Control Arm by Using Finite Element method" Page Nos. 3 - 12.

[3]. Eric Deschamps, "Fundamental Physics Behind New Suspension Concept for Automobiles" SAE TECHNICAL PAPER SERIES, 2001-01-1647, Page Nos. 1 - 4.

[4]. Dr. J.M. Mahishi, Director Engineering, "STATIC AND MULTI-AXIAL FATIGUE ANALYSIS OF AUTOMOTIVE LOWER CONTROL ARM USING NEiNASTRAN", MS&M Engineering Inc, Farmington Hills, MI, USA, Page Nos. 3-5.

[5]. Craig Lewitzke and Ping Lee (General Motors Corporation), "Application of Elastomeric Components for Noise and Vibration Isolation in the Automotive Industry", SAE TECHNICAL PAPER SERIES, 2001-01-1447, Page Nos. 12 - 20.

[6]. Jing Heng Wen (E-Z-GO, A Textron Company), “FEA Modeling Verification and Validation: Correlating Model with Test Data by Optimization Analysis”, SAE TECHNICAL PAPER SERIES, 2007-01-1745, Page Nos. 2 - 7.

[7]. Murali M. R. Krishna, William J. Kroppe and Scott V. Anderson (DANA Corp.), “Flexibility Effects of Control Arms & Knuckle on Suspension – A Finite Element Vs. Rigid Body Comparative Analysis”, SAE TECHNICAL PAPER SERIES, 2000-01-3446, Page Nos. 2 – 12.

[8]. Paulsen, T.J., Improvement to the Suspension and Modularization of the C2500 Pickup Truck Finite Element Model, Thesis, University of Nebraska – Lincoln, April 2003.

[9]. Tau Tyan, Minishiou Huang & Miguel A.Eguia , (NAE Crash safety, Ford motor company), “Impact Testing of Lower Control Arm for Crashworthiness Simulation”, SAE TECHNICAL PAPER SERIES, 2005-01-0352, Page Nos. 6 - 13.

[10]. Stephen Willam Hope (Thyssen Krupp Automotive AG), Alexandre de Paula Pinto (Ford Motor Company), Dominique Cadiot (Hutchinson), “Lower Control Arm Rear Bushing Development An Integrated Approach”, SAE TECHNICAL PAPER SERIES, 2003-01-3676, Page Nos. 2 – 4.