International Journal of Science and Research (IJSR)
ISSN (Online): 2319-7064
Impact Factor (2012): 3.358

Finite Element Analysis and Topology Optimization of Lower Arm of Double Wishbone Suspensionusing RADIOSS and Optistruct

Vinayak Kulkarni[1], Anil Jadhav[2], P. Basker[3]

[1, 2]PG Scholar, M.Tech (Automotive Engineering), School of Mechanical and Building Sciences, VIT University, Vellore- Tamilnadu India-632014

[3]Assistant Professor, School of Mechanical and Building Sciences, VIT University, Vellore- Tamilnadu India-632014

Abstract*: Stability, road handling and comfort of vehicle depend on optimum design of suspension system. Mostly all passenger cars and light trucks use independent suspension system because of inherent advantages over rigid suspension systems. Double wishbone system which is also called as Short Long Arm system consists of upper and lower wishbone arms. While actual running conditions forces like braking, cornering and vertical loads are taken by lower arm of suspension system, hence probability of failure of lower arm under these forces is more .Also this component is subjected to road irregularities, hence its dynamic behavior need to be understand. This paper deals with calculating the forces acting on lower wishbone arm while vehicle subjected to critical loading conditions (Braking, Cornering and Descending though slope). Suspension geometry and suitable materials for the suspension arm has been identified. Lower arm suspension has been modeled using Pro-Engineer. Von mises stress –strain is carried out in order to find out maximum induced stress and strain, while modal analysis is done for finding out natural frequencies and mode shapes of component. These analysis were carried using Altair Hyperworks and solver used is Radioss. From the analyzed results, design parameters were compared for two different materials and best on was taken out. From result obtained it was found that current design is safe and is somewhat overdesign. So in order to save material and reduce weight of component, Topology optimization analysis is carried out in Hyperworks which yielded in optimized shape.**

Keywords: Suspension, Wishbone, Hyperworks, Topology

1. Introduction

Suspension is the system which consists of dampers, springs and mechanical linkages that connects a vehicle to itswheels [1].Suspension systems have been evolved from early day's horse-drawn carriage with simple leaf springs to modern era vehicles with complex mechanisms [2].

Vehicle suspension system fulfills following purposes: 1) In order to maintain contact between ground and wheel it provides a vertical obedient element between un-sprung and sprung mass, by reducing the sprung mass motion.2) To maintain proper attitude of the vehicle during various operating conditions like braking, cornering, accelerating etc. 3)To maintain road holding and steering characteristics [3].Overall performance of suspension system is limits on maximum suspension travel, transmissibility of forces, road holding, minimum weight and cost [4].

There are three different types of suspensions namely: Dependent (Rigid Axle), independent and semi-independent suspensions [5]. In the independent suspension system, there are no linkages between two hubs of same axle and it allows each wheel to move vertically without affecting the opposite wheel. This system has inherent advantages over dependent system such as more space for engine, better roll resistance, lesser un-sprung weight and better resistance to steering vibration [6]. Dependent suspension or rigid axles provide a solid connection between two wheels of the same axle. Therefore motion of one wheel is transferred to the other wheel while travelling along surface irregularities [7].Semi-rigid suspensions system shows intermediate characteristics between the other two categories [5].

Double wishbone suspension system is type of independent suspension system which is consist of two lateral control arms namely upper and lower arm. The control arm (or wishbone or A arm) is nearly flat and roughly triangular member, that pivots in two places. The broad end of triangle is attached at the frame and pivots on bushing. The narrow end attaches to the steering knuckle and pivots on the ball joint [1].

During running condition lower control arm subjected to loads due to variation in gross weight and impact loads due to fluctuation of road surface and additional forces such as braking and cornering. Because of this complex nature of loads, the chances of bending and hence failing of lower control arm at ball joint are very high which is undesirable, hence it is necessary to carry out static and modal analysis of lower control arm [8].

With fierce competition in automotive world, major automotive OEM's (Original Equipment Manufacturers) are forced to reduce fuel consumption and cost of vehicle while assuring the safety. Hence, minimizing weight is becoming major objective of automotive industry without compromising on reliability and durability [8].

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 have to specify three parameters that is Design Variables (material density), Design objective (Weight reduction) and design constraints (Volume) [9].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 [10]. This paper deals with calculation of various forces acting on lower control arm of independent suspension system and modeling of lower control arm. Also this paper describes structural and modal analysis using different materials. Topology optimization also carried out for lower control arm in order achieve weight reduction.

2. Objective

The key objective of this effort is to carry out static and modal analysis of lower control arm using different materials and also perform topology optimization for achieving weight reduction.

- Determinations of the forces acting on the lower control arm during various running conditions.
- Solid modeling of the lower control arm of suspension system.
- Determination of Von mises stress.
- Determination of total determination.
- Determination of natural frequencies and corresponding mode shapes.
- Comparison of design parameters viz stress, total deformation, natural frequencies and mode shapes for different materials.
- Topology optimization.

3. Methodology

In order to proceed with this study various forces acting on lower control arm were calculated. CAD model of lower control arm designed in Pro-Engineer was imported in Hypermesh for geometric cleanup and meshing. Meshed model of the lower arm essentially consist of 11280nodes and 42822 elements with total 35177 degree of freedoms. Tetra elements give enhanced result as compared to other types of elements, therefore the elements used in this analysis is tetra elements.

Two materials namely Mild Steel and Aluminum Alloy (7075-T6) were used for lower control arm. Calculated forces and boundary conditions were applied on meshed model in HYPERMESH as shown in figure 1. Static and modal analysis was performed by using RADIOSS. Design parameters obtained from above Finite Element Analysis were compared for above stated materials and best one was selected. Topology optimization of the model was carried out using OPTISTRUCT module of HYPERWORKS software.

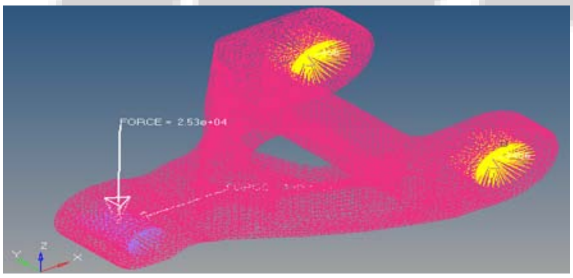

Figure 1: Lower control arm with meshing, loading and boundary condition

3.1 Vehicle specifications

Table 1: Vehicle Parameters [2]

Parameters	*Descriptions*	*Values*
M	Mass of vehicle (kg)	3050
A	Front axle track width (m)	1.67
H	Height of center of gravity (m)	1.16
L	Wheelbase (m)	2.9
C	Distance from front axle to CG (m)	1.276
B	Distance from rear axle to CG (m)	1.624

Table 2: Assumption made for calculations [2]

Parameters	*Description*	*Values*
R	Radius of curvature (m)	100
β	Angle of banking (Degree)	12
θ	Slope (Degree)	11
M	Coefficient of friction between Tires and road	0.6
F	Retardation By braking (m/s^2)	6
V	Velocity of vehicle(kmph)	120

3.2 Design parameters

In case of vehicle in actual running conditions forces acting on it are of dynamic in nature and changes as per driving conditions. Various longitudinal forces are acting due to braking and acceleration while lateral forces acting due to cornering of vehicle. In order to make preliminary analysis steady state operating conditions are assumed. The assumptions made are smooth road conditions, steady state cornering and constant grade.

1) Vertical Loads acting on wheel:
In order to determine forces acting on lower control arm, following critical situations are considered. For above condition, load acting on front outerwheel is given by following formula [11].

a)Vehicle at the instant of braking on downhill grade:
$(W_{fo)\,brak}$=(1/2L) [W (H Sin θ + C Cos θ) + m.a.H] (1)
Where, $(W_{fo)\,brak}$is breaking force on outside wheel.

b)Vehicle at the instant of cornering:
For above condition, load acting on front outer wheel is given by following formula,
$(W_{fo)corn.}$=(W/2a) [(V^2/2*g)(a* Sin β + 2H* Cosβ) + (a.Cosβ– 2H* Sin β)] (2)
Where,$(W_{fo)corn}$ is cornering force on outside wheel.

2) Lateral Loads acting on Wheel
while vehicle taking turn, lateral forces acts on it which is given by
$(L)_{fo}$= μ *$(W)_{total}$ (4)
Where, $(W)_{total}$ = $(W_{fo)\,brak}$+ $(W_{fo)corn}$ (5)
Numerical values of forces acting on wheel are given by table 3.

Table 3: Load acting on front wheel

Sr. No.	*Velocity(kmph)*	*$(W)_{total}$ (N)*	*$(L)_{fo}$(N)*
1	120	16424.69	14012.43

Now, front outer wheel will subjected to maximum load if vehicle is subjected to above two conditions simultaneously i.e. vehicle is subjected to cornering and braking on downhill grade. Therefore, total load acting on front outer wheel can be determined by summing up the loads due to above conditions

3) Forces acting on lower control arm

Let R_x and R_y be the maximum forces at the center of contact patch of front tire as shown in figure 1 and 2.

Let Px, Py and Q_x, $Q_y$be the reaction forces acting on lower control arm as shown in figure 2.

The reaction forces (P_x, P_y and Q_x, $Q_{y.}$) acting on lower control arm was found out by using equilibrium equation of mechanics.

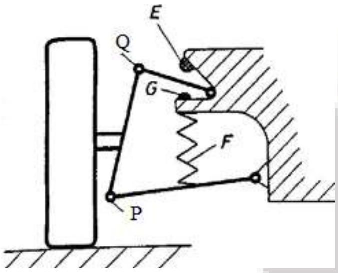

Figure 2: Suspension diagram

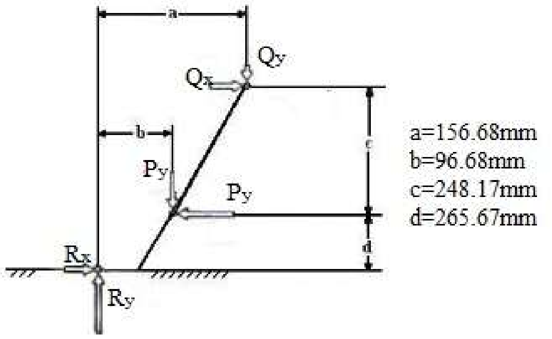

Figure 3: Forces on suspension

Table 4: Reaction Forces acting on the suspension arms

Sr. No.	*Velocity(kmph)*	*Qx (N)*	*Qy(N)*	*Px (N)*	*Py(N)*
1	120	4434	702	18187	25290

Material Properties

Table 5 gives mechanical properties of selected materials

Table 5: Material Properties [12]

Mechanical Properties	*Mild Steel*	*Al-7075 T6*
Young's Modulus (Mpa)	210000	71700
Poisson's Ratio	0.3	0.33
Density (kg/m3)	7850	2890
Yield Stress (Mpa)	350	503

4. Results and Discussions

The behavior of lower control arm with different material has been studied by using design parameters (von misesstress, strain and total deformation, mode shapes and its frequencies) which were obtained from analysis.

a) Static analysis:

From figure4 and 5, it is observed that the maximum stress developed in the component is 72.4 Mpa for Mild Steel and 71.89 Mpa for Aluminum Alloy which is lower than the maximum allowable stresses that is 350 Mpa and 503Mpa respectively for Mild Steel and Aluminum Alloy. Hence, design is safe that is the values of maximum stresses are acceptable as compared to yield strength of respective material. From figure 6 and 7, it is observed that the total deformation in the component is 0.43mm for Mild steel and 1.3mm for aluminum alloy which are less than the thickness of the component and also deformation limit of the material. In order maintain constant attitude of vehicle and also to maintain road holding, component should not get deformed beyond safe limit. For same loads and boundary condition, the maximum stress developed in component is approximately same for both materials whereas deformation is more for Aluminum Alloy than for Mild Steel.

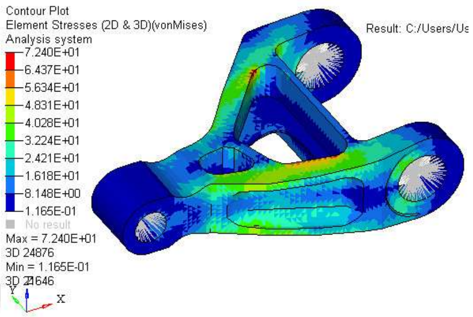

Figure 4: Von mises stress for Mild Steel

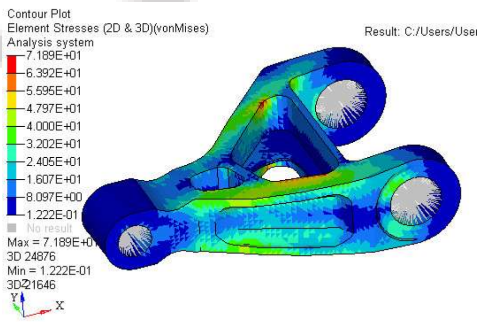

Figure 5: Von mises stress for Aluminum Alloy

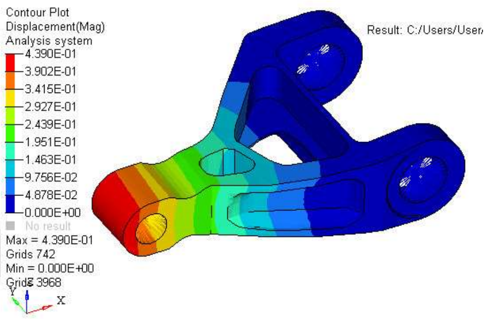

Figure 6: Total deformation for Mild Steel

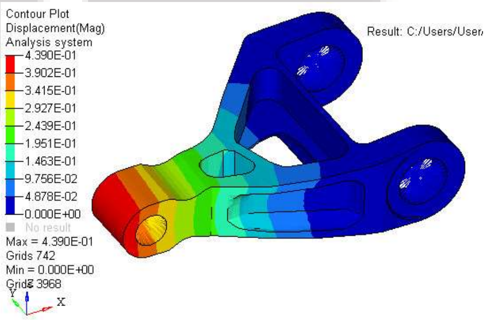

Figure 7: Total deformation for Aluminum Alloy

b) Modal analysis

The objective of Modal Analysis is to determine the natural motions of a system, and the frequencies at which those motions occur. Modal analysis is extremely important in situations where resonance is a potential problem. (Resonance is the tendency of the system to oscillate with greater amplitude at some frequencies than at others). The results of a modal analysis in hypermesh will be the natural frequencies and the mode shapes.

From table 6, it is observed that, the natural frequencies for first ten modes vary from 442Hz to 2729 Hz for Aluminum Alloy and 457 to 2834Hz for Mild Steel. For same mode shape, the natural frequency of component for mild steel is higher than that for .

Table 6: Modal frequencies and deflection for two materials

Mode Shape	*Aluminum Alloy*		*Mild Steel*	
	Frequency (HZ)	*Displacement (mm)*	*Frequency (Hz)*	*Displacement (mm)*
1	442	18.07	457.68	10.94
2	927.47	18.07	958.73	10.63
3	960.5	13.63	987.51	8.39
4	1366.95	16.88	1418.89	10.22
5	1684.6	13.99	1745.35	8.51
6	1831.8	12.05	1892.06	6.96
7	1899.35	35.85	1950.58	21.49
8	2024.9	62.39	2089.97	37.36
9	2551.65	25.64	2623.65	15.54
10	2729.11	25.15	2834.68	15.24

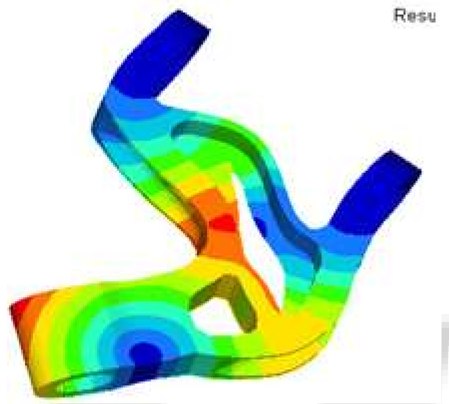

Figure 8: Mode shape for mode number 3

From table 6, also it is observed that at approximately same frequency for Mild Steeland Aluminum Alloy, the total deformation produced is somewhat less in Mild Steel than Aluminum Alloy. As the frequency increases, the number of modes also increases. Modes (or resonances) are inherent properties of a structure. Mass, stiffness, damping ratio and boundary condition determines the resonance. Each mode is defined by a natural (modalor resonant) frequency, modal damping, and a mode shape. Hence, modes are changed by changing the material properties or boundary condition. Figure 8, shows the shape of mode number 3. The shape for both materials is remaining same but frequency is changes and hence resonance condition also changes.

c) Topology optimization

Optimization technique gives an optimum component within given design space. The design space can be defined using shell or solid elements, or both. The classical topology optimization set up solving the minimum compliance problem, as well as the dual formulation with multiple constraints are available. Constraints on von mises stress and buckling factor are available with limitations. Manufacturing constraints can be imposed using a minimum member size constraint, draw direction constraints, extrusion constraints, symmetry planes, pattern grouping, and pattern repetition. A conceptual design can be imported in a CAD system using an iso-surface generated with OSSmooth, which is part of the OptiStruct package.

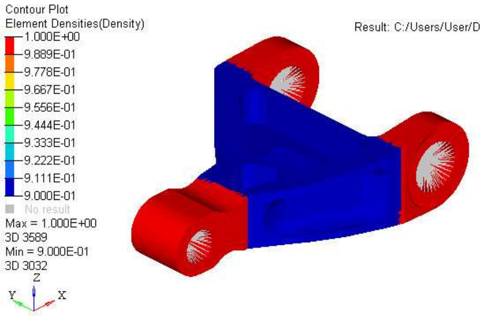

Figure 9: Component with uniform density

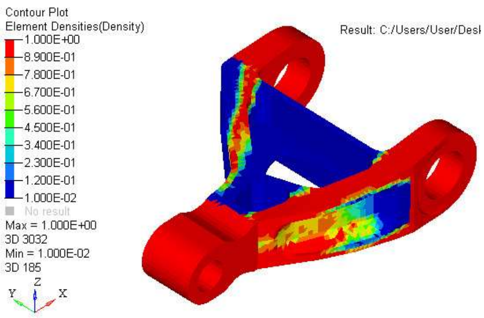

Figure 10: Optimized component

From the table 7, it is found that the factor of safety is almost double of critical factor of safety. Hence it is somewhat overdesign, for that removal of the material from the component which is not heavily stressed. This will leads to reduction of the weight and hence the fuel consumption.

Figure 9 shows lower arm component with uniform density material. Optistruct take this geometry as initial iteration. Optistruct identifies stress distribution pattern throughout the lower arm and remove the material from that region in successive iterations based upon set of objectives and constraints. This material removal is given by varying density of each element from 0 to 1. After number of iterations, when solution converges the density pattern of component is like figure 10. In figure 10, a region with lower density indicate that it can be removed without hampering safety of component. So by removing the material from these design space of component objective of reducing weight of component will be fulfilled without changing volume.

Table 7: Analyzed results for both materials

Design Parameters	Aluminum Alloy	Mild Steel
Max. Stress (Mpa)	71.89	72.4
Max. Displacement(mm)	1.3	0.43
Yield strength(Mpa)	503	350
Factor of safety	6.9	4.8

5. Conclusions

In this project, the forces acting on lower control arm of wishbone has been calculated while vehicle subjected to critical loading. The CAD model of lower control arm has been carried out using Pro-E software packages. The static and modal analysis as well as optimization of lower control arm has been carried out in Hyperworks. From the analyzed results, it is concluded that.

- The values obtained for the maximum von mises stress and maximum displacement are lower than safe limit. On strength basis, aluminum alloy is good material than Mild Steel whereas on strain basis, Mild Steel is good material than aluminum alloy.
- Modes and mode shapes of lower control arm contingent on material properties. Hence change in material leads to change in resonance condition. Modes are used as a simple and efficient means of characterizing resonant vibration.
- The higher factor of safety leads to optimization of component. Topology optimization generates an optimized material distribution for a set of loads and constraints within a given design space. Optimization reduces weight, product design cycle time and cost.

References

[1] Chandrakant M. Chaudhari, Dr. M V. Sulakhe, "Finite element analysis and testing of suspension arm," National Conference on Advances in Mechanical Engineering, 03 - 05January, 2011.

[2] Duygu Guler, "Dynamic analysis of double wishbone suspension system"İzmir Institute of Technology,July 2006.

[3] Ragnar Ledesma,Shan Shih, "Heavy and Medium Duty Vehicle Suspension-Related Performance Issues and Effective Analytical Models for System Design Guide," SAE TECHNICAL PAPER SERIES, 1999-01-3781.

[4] J. Y. Wong, "Theory of Ground Vehicles", John Wiley & Sons, Third Edition.

[5] Giancarlo Genta, Lorenzo Morello, "The automotive chassis,"volume 1: component design, springer science publication.

[6] Thomas D. Gillespie, "Fundamental of vehicle dynamics",society of automotive engineer Inc.

[7] Prof. Dipl.-Ing. Jörnsen Reimpell,Dipl.-Ing. Helmut Stoll,Prof. Dr.-Ing. Jürgen W. Betzler, "The Automotive Chassis:Engineering rinciples," second edition, Butterworth-Heinemann publication.

[8] Li-Hui Zhao,Song-Lin Zheng, Jin-Zhi Feng, "Failure mode analysis of torsion beam rear suspension under service conditions" ScienceDirect, Engineering Failure Analysis 36 (2014) 39–48.

[9] Vivek Zolekar, Dr. L.N. Wankhade, "Finite Element Analysis and Optimization of I.C. Engine Piston Using RADIOSS and OptiStruct," Altair technology conference,2013.

[10] Badih A. Jawad,Brian D. Polega, " Design of Formula SAE Suspension Components,"SAE International, 2002-01-3308.

[11] William F. Milliken, Douglas L. Milliken, "Race car vehicle dynamics,"SAE international publication.

[12] Hemin M. M, Rahman M. M, Omar R.M., "Dynamic Analysis of Vehicle Arm Based on Finite Element Approach"Journal of Advanced Science and Engineering Research 1 (2011) 124-136.