ISSN 2395-1621

Design Optimization of Integrated Steering Knuckle

#1Pooja Chavan, #2Dr. D.N. Kamble

1poojachavan.b@gmail.com
2dnkamble.sae@sinhgad.edu

#1PG Student, Mechanical Engg. Dept, SAOE Kondhwa, Pune-411048
#2Professor, Mechanical Engg. Dept, SAOE Kondhwa, Pune-411048

ABSTRACT

Steering knuckle is a prominent component in car which actually takes the loads from the wheel and transfers these forces to the suspension system. Steering knuckle requires lots of attention in selection because once it is damaged then it has to be replaced by the new one. Structural components such as a steering knuckle might be strong enough to withstand a single applied load but has a chance to fail when subjected to a fatigue load. Mass or weight reduction is becoming important issue in car manufacturing industry. Weight reduction will give substantial impact to fuel efficiency, efforts to reduce emissions and therefore, save environment. Weight can be reduced through several types of technological improvements, such as advances in materials, design and analysis methods, fabrication processes and optimization technique, etc. The paper deals with design of integrated steering knuckle and its weight optimization. Various loading conditions like static and dynamic loadings were carried out and design has been optimized by reducing the weight by using topology optimization while maintaining the strength.

***Keywords* — Integrated Steering Knuckle, FEA, Weight Optimization**

ARTICLE INFO

Article History

Received :18th November 2015

Received in revised form :

19th November 2015

Accepted : 21st November , 2015

Published online :

22nd November 2015

I.INTRODUCTION

Steering knuckle is one of important component of vehicle which is connected to steering, suspension and brake to chassis of vehicle. It undergoes different loading under different conditions [2].The Weight reduction is becoming important issue in car manufacturing industry. Weight reduction will give substantial impact to fuel efficiency, efforts to reduce emissions and therefore, save environment. Weight can be reduced through several types of technological improvements, such as advances in materials, design and analysis methods, fabrication processes and optimization techniques, etc [10].Steering Knuckle links suspension, steering system, wheel hub and brake to the chassis. It undergoes varying loads subjected to different conditions without affecting vehicle steering performance and other desired vehicle characteristics [4] .Optimization process for this work was conducted using OptiStruct solver in order to reduce the weight of the component which will reduce the cost with respect to the weight production process [8][9].

II. MODEL

A. Design parameter of Steering Knuckle

The input parameter for modeled the steering knuckle includes loads in case of loads acting on knuckle in road bump case, braking case and cornering case.

Axial loads: The two major loads acting on the knuckle are Tensile and Compressive loads. The stresses due to these loads can be determined as

Tensile Load (P_t) = Tensile Stress X Area (1)

Compressive Load (P_c) = Compressive Stress X Area

Inertia loads: To calculate the inertia force, first two harmonies are taken into consideration load due to the inertia of the moving partsIt is given by,

Inertia load (F_a) $=\omega^2 R\left(\cos\theta + \frac{R}{L}\cos(2\theta)\right)$ (2)

Bending load: This load is due to the weight of the vehicle which is acting on the knuckle. This force trends to bend the steering knuckle outwards, away from the center line. It is alternating one, and at high speed, it is considerable.

Total inertia bending force is given by,

$F_b = \frac{\rho A_i L^2 \sin(\theta+\emptyset)}{2}$ N (3)

Loads on steering knuckle

- Road bump case
- Braking case
- Cornering case

Car wheel designation: BOLERO (P215/75 R15)

- Turning radius:- 6.35 meter
- Gross vehicle weight (GVW):- 2200kg
- Mass of the vehicle is splitted –

Mass of front wheel = 2200/4= 550kg

B. CAD model

CAD model of steering knuckle has been prepared in CATIA V5. It consist of stub hole, brake caliper mounting points, steering knuckle mounting points, suspension upper and lower arm mounting points. Knuckle design mainly depends on suspension geometry and steering geometry.

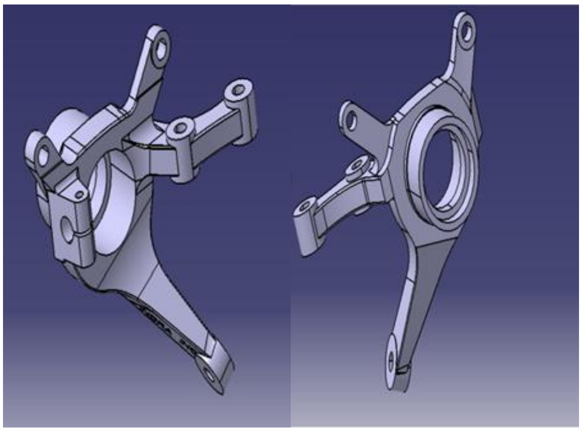

Fig. 1CAD model of simplesteering knuckle in CATIA V5

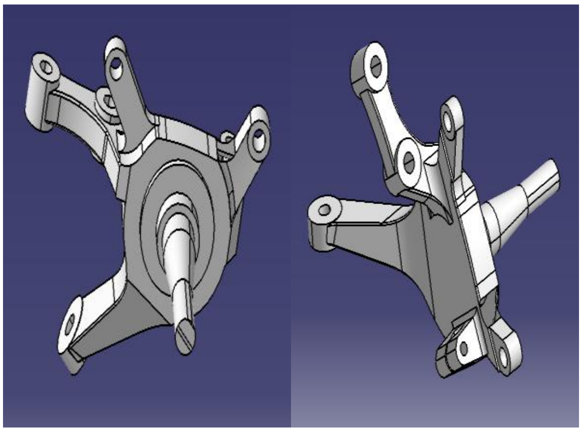

Fig. 2 CAD model of integrated steering knuckle

III. STRUCTURAL ANALYSIS

A general-purpose commercial finite element code, Hyper-Mesh and Ansys is applied to conduct the static simulations. The FEA model of steering knuckle in this study is constructed based on the geometry. A full 3-D solid model is constructed for the static test simulation. The schematic of an FEA model used in static test simulations is shown in figure 3.

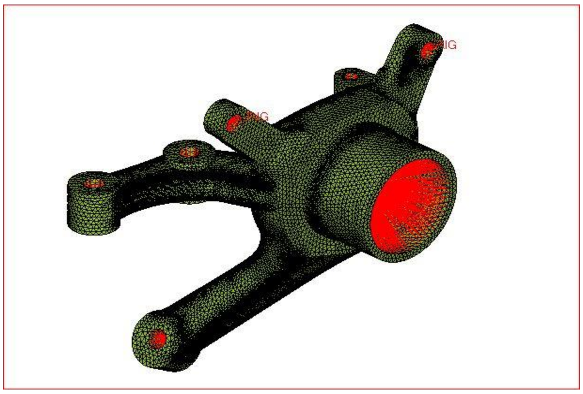

Fig.3Meshed model with rigids

TABLE I

MATERIAL PROPERTIES: FORGED STEEL

Property	***Value***
Young's Modulus, E	***205 GPa***
Poisson's Ratio ,v	***0.29***
Density, ρ	***7850 kg/m^3***
Yield Stress, σ_{yield}	***390 MPa***
Ultimate Tensile Stress, σ_{uts}	***520 MPa***

A. Applied Boundary Conditions

The suspension joint portion and lower arm portion is made fixed (as shown in figure 4) and then various loads are applied and analysis was done.

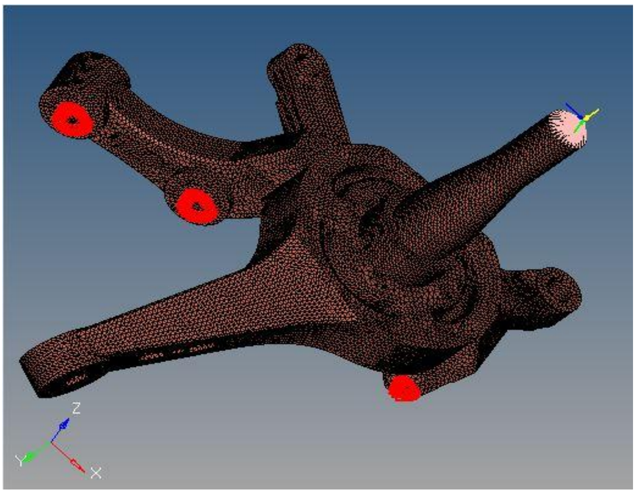

Fig. 4 Boundary conditions applied on model

TABLE II

BOUNDARY CONDITIONS

Load	***Value***
Road Bump Case	***26510 N***
Braking Case	***2697.5 N***
Cornering Case	***1700.25 N***

Meshing Details

No. of element =189421
No. of nodes =41966

B. Analysis Results

Simple Steering Knuckle

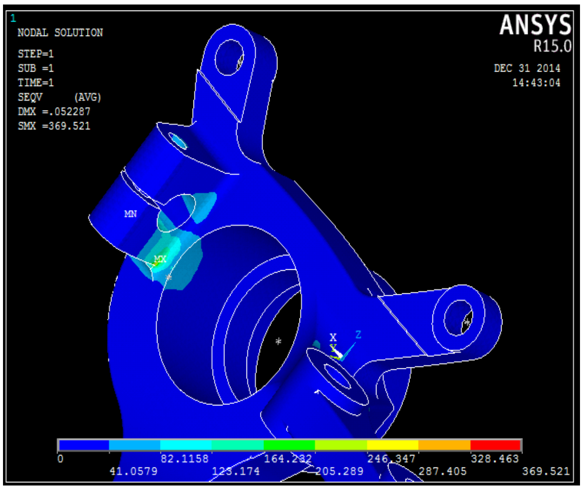

Fig. 5Stress analysis of simple component

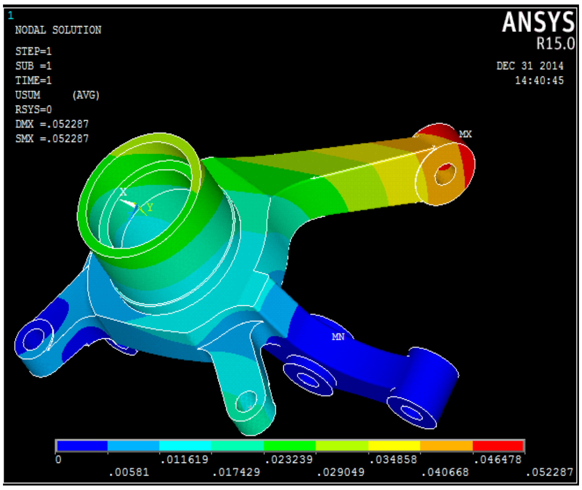

Fig. 6Displacement analysis of existing component

Integrated Steering Knuckle

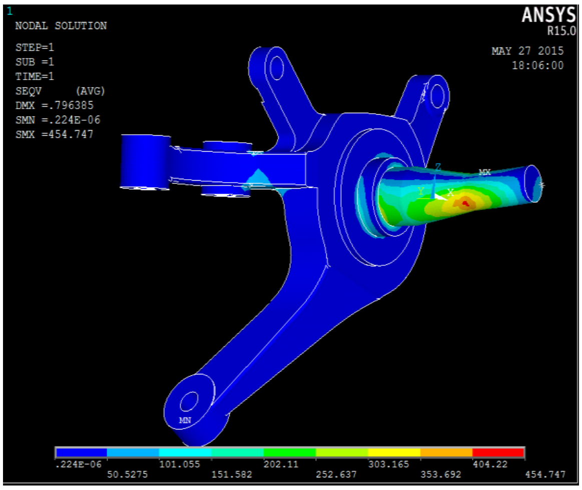

Fig. 7 Stress analysis of integrated component

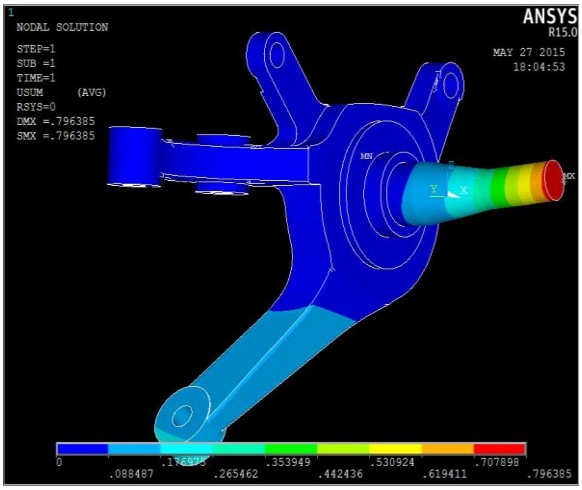

Fig. 8 Displacement analysis of integrated component

TABLE III

SIMMULATION RESULT

S.no	Type of knuckle	Max. STRESS	Max. displacement
1.	***Simple Steering Knuckle***	***369.521Mpa***	***0.052287mm***
2.	***Integrated Steering Knuckle***	***454.747Mpa***	***0.796385mm***

C. Optimization

Topology optimization

Optimum material layout for a given design space which takes into any number of design constraints.By defining a design space that applying boundary conditions such as predefined loads and fixture positions, topology optimization can suggest the ideal layout of material to meet defined performance targets. Topology optimization can be used at the concept level of the design process to arrive at a conceptual design proposal that is then fine tuned for performance, weight and manufacturability. This process replaces time consuming and costly design iterations and hence reduces design development time and overall cost while improving design performance

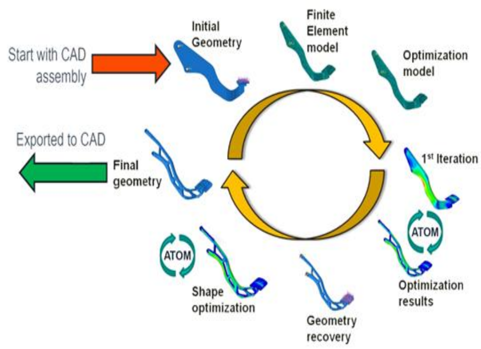

Fig. 9 Loop of topology optimization

OptiStruct

It uses highly advanced optimization algorithms; OptiStruct can solve the most complex optimizationproblems with thousands of design variables in a short period of time. OptiStruct advancedoptimization engine allows users to combine topology, topography, size and shape optimizationmethods to create better and more alternative design proposals leading to structurally sound andlightweight design. Manufacturing requirements can also be defined as input to the simulation tocreate design proposals that are easier to interpret and to manufacture.

TABLE IV

TOPOLOGY OPTIMIZATION METHODOLOGY

Design Constraint	***Von- Mises stress < 390 Mpa***
Objective Function	***Volume reduced***
Design Variable	***Density of the element***

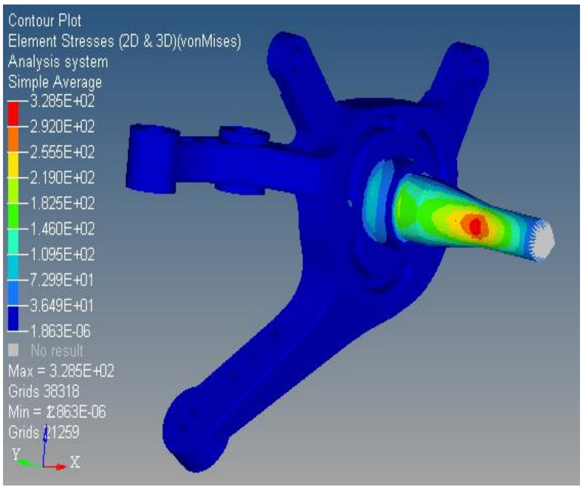

Fig. 10 Stress analysis of optimized geometry

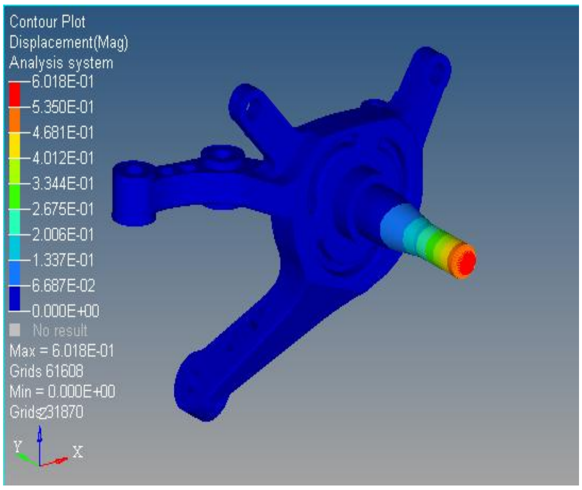

Fig. 11 Displacement analysis of optimized geometry

From the static analysis of the optimized geometry it is observed that the maximum magnitude of thedisplacement is 0.6018 mm and the maximum stress 328.5 Mpa observed as shown in fig 10 and 11.

IV. PROPOSED EXPERIMENTATION AND VALIDATION

A prototype for experimentation is produced for testing. The input conditions will be recreated in the lab while the component testing. The loading and the boundary conditions will be matching the practical workingconditions in which the vehicle is expected to perform. For simplicity, a Universal Testing Machinealong with a suitable fixture for the component shall be engaged for testing purposes. The measured values will be compared with the results from software and will be validated accordingly.

The proposed setup is expected to be performing better with a satisfying amount of weight reduction. The weight reduction will hence lead to better fuel efficiency of the vehicle.

Fig. 12 Prototype produced

V. CONCLUSION

Three CAD models with three different loading cases of the steering knuckle are generated in CATIA and these modelsare imported to ANSYS for analysis. The results obtained from structural analysis ANSYS are compared with existing steering knuckle and it is observed that the stress values are less than their permissible yield stress values. So the design is safe. The model has been optimized by using OptiStruct solver.

A prototype of integrated steering knuckle has been prepared for experimentation and is expected to perform better with satisfying amount of weight reduction.

REFERENCES

1. Thomas D.Gillespie, "Fundamentals of Vehicle Dynamics", Society of Automotive Engineers, USA, 1992.
2. Mahesh P. Sharma, Denish S. Mevawala, Harsh Joshi, Devendra A. Patel, "Static Analysis of Steering Knuckle and Its Shape Optimization", IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE).
3. B.Babu, M.Prabhu, P.Dharmaraj, R.Sampath, "Stress Analysis of Steering Knuckle of Automobile Steering System", International Journal of Research in Engineering and Technology, March 2014.
4. Sharad Kumar Chandrakar, DheerajLalSoni and ShohelGardia, "FEA of A Steering Knuckle for Life Prediction'', International Journal of Engineering Research and Technology.
5. VirajRajendraKulkarni, AmeyGangaramTambe, "Optimization and Finite Element Analysis of

Steering Knuckle", Altair Technology Conference, Dec 2013.

6. Wan Mansor Wan Muhamad, EndraSujatmika, Hisham Hamid, &FarisTarlochan, "Design Improvement of Steering Knuckle Component Using Shape Optimization", International Journal of Advanced Computer Science, Feb. 2012.
7. . MehrdadZoroufi and Ali Fatemi, "Fatigue Life Comparisons of Competing Manufacturing Processes: A Study of Steering Knuckle", The University of Toledo.
8. Chang Yong Songa, Jongsoo Lee, "Reliability based design optimization of knuckle component using conservative method of moving least square meta-models.", Science Direct, 2010.
9. KamleshLalasahebChavan,S R Deodas, S.S.Kulkarni, "Mass reduction for steering knuckle arm in a suspension system throught topology optimization in CAE".
10. PurushottamDumbre, A. K. Mishra, V. S. Aher"Structural analysis of steering knuckle for weight reduction"

