ISSN 2395-1621

Topology Optimization of Automotive Steering Knuckle using Finite Element Analysis

[#1]S.R.Gore

[#1]Mechanical Engineering Department , Siddhant College of Engineering, Savitribai Fule University, Pune, India

ABSTRACT

This thesis aims towards the developing strategies in the area of structural optimization and to implement these strategies in design process. The steering knuckle has been chosen as a study component in this thesis. The objective of this project study is to reduce the mass of an existing steering knuckle component of a racing car model by applying topology optimization technique. A finite element software Hyper work which contains several modules is used to achieve objective. Hypermesh is used to prepare the finite element model while Optistruct is utilized to carry out static analysis and topology optimization. Static analysis is carried out for four load cases considered simultaneously for extreme condition analysis. Optimization method used in this study have succeeded in reducing the mass of an existing knuckle component by 12.5%.

Keywords— Optimization, Steering knuckle, FEA, Topology

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 was used as component for study. Main design and functionality of steering knuckle depends on type of suspension implemented. Additional factors like brake caliper used, mounting of tie rod of steering sub-system also effects knuckle design. Suspension system in any vehicle uses different types of links, arms, and joints to let the wheels move freely; front suspensions also have to allow the front wheels to turn. On constraining the knuckle, combined load of brake torque on the caliper mounting, longitudinal loads due to traction, vertical reactions due to weight and steering reaction, the finite element model was solved using RADIOSS solver. The stress levels and deformation was checked using HyperView for static as well as dynamic conditions. FEA results were verified by analytical calculations. OptiStruct solver was used for performing Topology Optimization to minimize the amount of material to be used and setting geometric parameters as design variables. Considering the results obtained from optimization, geometric model was modified and iterated until satisfactory results were achieved.

II. PROBLEM DEFINATION

This research aims to provide solution to a car designer in optimizing the component structure against the types of failure modes. Here the component is Steering Knuckle. Designing a light weight component for fuel efficiency is likely to increase the fatigue stress distribution. This study will identify the potential failure modes of crack initiation using topology and shape optimization approach. To study the failure modes of component that are subjected to weight reduction, in term of shape and dimension. Designing a light weight component for fuel efficiency is likely to increase the fatigue stress distribution.

2.1 Objectives

1. Optimization using Finite Element Method.
2. Stress value under loading constraints.
3. The failure modes of parts will be analyzed.
4. To reduce potential failures.

2.1 Methodology

Topology and Shape optimization was applied to reduce volume or weight of knuckle component in a local car model. Modeling, simulation and optimization processes used software modules included in Hyper Works, HyperMesh. Solid model was imported for finite element modeling where loads and constraints were applied. Shape optimization process requires shape definition for design variables and HyperMorph was used to conduct such purpose.

III. LITERATURE REVIEW

Mahesh P. Sharma et. al. "Static Analysis of Steering Knuckle and Its Shape Optimization" focus on approaches to automating the manual optimization process and the challenges that it presents to the engineering community. The study identifies scalability as the major challenge for design optimization techniques. Large-scale optimization will require more research in topology design, computational power and efficient optimization algorithms.

Wan Muhamad et.al. "Failure modes of a vehicle component designed for fuel efficiency" uses the different material than regular material for optimization of steering knuckle. They use metal matrix composites as it has potential to meet demanded design requirements of the automotive industry, compared with conventional materials. Structural analysis of steering knuckle made of alternate material Al-10% tin was performed using commercial code. It is found from the analysis; the knuckle strut region has maximum stress and deflection during its life time.

Prof. Purushottam Dumbre et. al. "structural analysis of steering knuckle for weight reduction" accesses fatigue life and compares fatigue performance of steering knuckle made from three materials of different manufacturing processes . These include forged steel, cast aluminum and cast iron knuckles. Finite element models of the steering knuckles were also analyzed to obtain stress distributions in each component. Based on the results of component testing and finite element analysis, fatigue behaviors of the three materials and manufacturing processes are then compared.

Faris Tarlochan et. al. " Modeling, Simulation and Optimization Analysis on Steering Knuckle Component For Purpose of Weight Reduction" discuss an integrated design and manufacturing approach that supports the shape optimization. The main contribution of the work is incorporating manufacturing in the design process, where manufacturing cost is considered for design.

Viraj Rajendra Kulkarni and Amey Gangaram Tambe "Optimization and Finite Element Analysis of Steering Knuckle" carry out the topology optimization of clamp cylinder t using CAE tools to reduce weight with the constraints of standard operating condition. the new optimized design of configuration is proposed.fea of optimized cylinder is also carried out and compared with acceptance criterion. the optimized model is equally strong and light in weight compared to existing model.

Hisham Hamid et. al. "Design Improvement of Steering Knuckle Component Using Shape Optimization" discuss the process of designing a light weight knuckle from scratch. The design space is identified for the knuckle and subsequently a design volume satisfying the packaging requirements is created from it. Using opti struct, topology optimization is performed on the design volume to derive the optimal load path required for the major load cases.

Pe´riaux J et. al. "multiple objective optimization strategies for electromagnetic backscattering" investigate system in any vehicle uses different types of links, arms, and joints to let the wheels move freely; front suspensions also have to allow the front wheels to turn. Steering knuckle/spindle assembly, which might be two separate parts attached together or one complete part, is one of these links.

Viraj Rajendra Kulkarni "Steering control and optimization through Finite Element Analysis" shows Steering Knuckle is one of the critical components of vehicle which links suspension, steering system, wheel hub and brake to the chassis. It undergoes varying loads subjected to different conditions, while not affecting vehicle steering performance and other desired vehicle characteristics.

Alex Richardson et. al. "Finite Element analysis of Steering Knuckle for weight reduction" explains the details of the problem, experimental set up, data collection, analysis, interpretation, and conclusions were explained. The main idea behind this experiment was to discover the reason behind the "rasping noise" heard in Suspension system when an automobile abruptly accelerates when it is subjected to cold conditions. The most important objective of steering nuckle is to reduce the optimization and noise coming from engine.

IV. PROJECT OVERVIEW

To find the failure modes of components that are subjected to weight reduction, in term of shape and dimension. The selected part is a steering knuckle. This is a safety part which is linked to brake disc and steering linkages. The steering knuckle is assembled on the brake disc housing. The research explores the design optimization using Finite Element Method to evaluate the stress values under loading constraints. The failure modes of parts will be analyzed.

To achieve the objectives following three different methodologies are used and validation with their result.

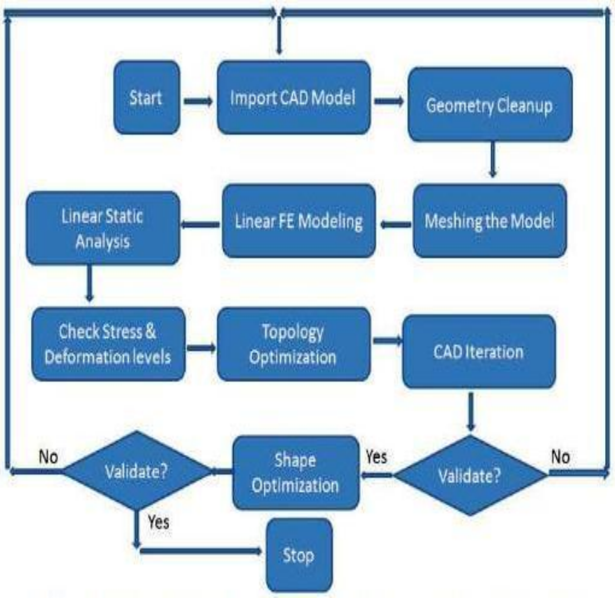
1. Finite Element Analysis 2. Experimentation.

4.1 Finite Element Analysis

Finite Element Analysis will be carried out by following steps.

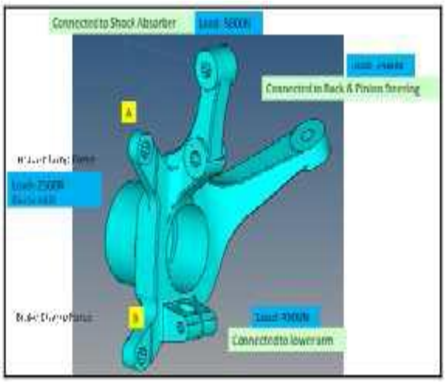
Step I - Creating Model Geometry

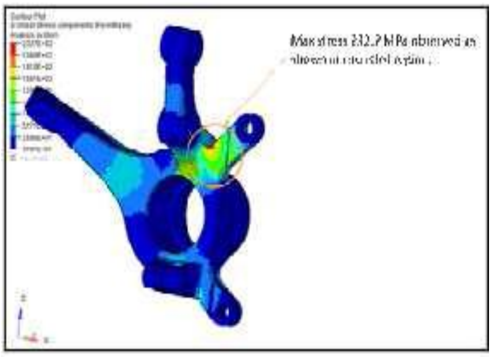
The main objective of this step is Creating the model geometry. There are two methods to create the finite element model is to generate Finite Element model by solid modeling and by direct generation. Solid modeling will be done in Pro-E wild fire 5.0 the geometric boundaries are described of model and then instructed the ANSYS 14.5

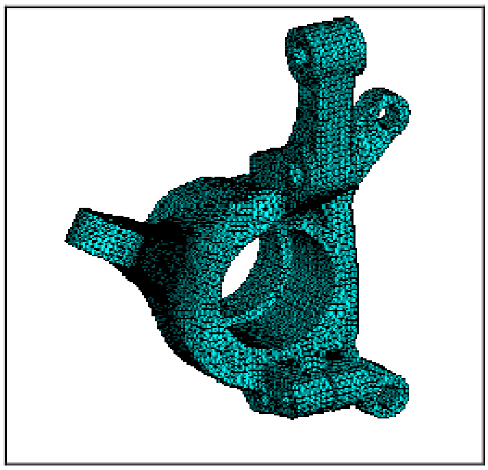


Step II - Apply load and boundary conditions.
In this step we are applying the forces on the instructed model and then apply boundary conditions on support of the vessel.

Step III - Solution
In this step we are used to review results. We can obtain contour displays, deformed shapes and tabular listings to review and interpret the results of the analysis.
MESHING:
CAD model of knuckle converted into STEP file. This model is imported into Abaqus Workbench simulation. Geometry cleanup was performed prior to meshing of model. Optistruct is used as solver. For better quality of mesh fine element size is selected. The base geometry is simulated for the actual loading conditions. The load conditions are obtained from data acquisition system. The loadings are as follows.







After the analysis of base model the maximum displacement is found to be 0.2 mm in the brake clamping holding area as shown in fig.3 The maximum stress is found in junction area brake clamping and shock absorb connecting arm as shown in fig.4
The material density distribution is observed for finding the low stress area where we can modify the geometry. Fig.5 shows the low stress area where we can remove material that not bear the load. For removal of material we have to consider the manufacturing aspect and some functional constraint. The feasibility of tooling required for modified geometry should check at the time of material removal.

4.3 Design Variables for Shape Optimization
The vector of nodal coordinates (x) is used to define the shape of steering knuckle structure in finite elements model. Changes of the boundary in model structure will translate to interior of mesh to avoid distortion of the elements when shapes change. During shape optimization, there are two approaches that can be used to account for mesh changes; those are the basis vector approach and the perturbation approach.
The basis vectors define nodal locations.
$x = \Sigma\, DVi \,.\, BVi$ (1)
Where,
x is the vector of nodal coordinates.
BVi is the basis vector associated to the design variable DVi.

Using the perturbation vector approach, the structural shape change is defined as a linear combination of perturbation vectors. The perturbation vectors define changes of nodal locations with respect to the original finite element mesh.
$x = xo + \Sigma\, DVi \,.\, PVi$ (2)
Where x is the vector of nodal coordinates, xo is the vector of nodal coordinates of the initial design, PVi is the perturbation vector associated to the design variable DVi.

4.4 Shape Optimization Parameters
An general optimization or a mathematical programming problem can be stated as follows .
Find (X) = which minimize f(X) subject to the constraints

$gj\ (X) \leq 0,\ j = 1, 2, \ldots, m$
$lj\ (X) = 0,\ j = 1, 2, \ldots, p$
where,

X is an n-dimensional vector called the design vector.

f (X) is termed the objective function.

gj (X) and lj (X) are known as inequality and equality constraints, respe.

In this paper, the objective function of shape optimization problem is as implicit function that is to minimize volume and subject to maximum stress of the elements as constraint.

4.2 Experimental Setup

The following are the component of experimental set up.

1) Steering Knuckle.
2) Universal Testing Machine (UTM).
3) Punching Machine.
4) Fixture.
5) Strain Guage.

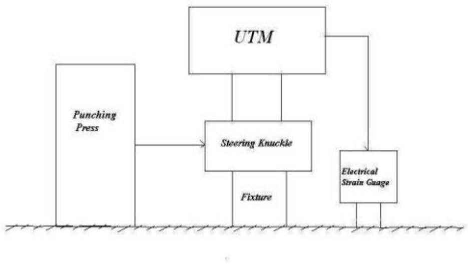


Fig. 4.1 Experimental setup for optimization of Steering Knuckle.

V. RESULT & DISCUSSION

The contour highlights maximum stress value as 71MPa and maximum displacement as 0.079mm. Thus the free run indicates design is safe and optimization is necessary. The rightmost image indicates element density contour. The red region indicates density as 1 and towards blue it lowers density up to 0 where material is not necessary. Now region below 0.3 density are aimed at removing in next CAD iteration.

5.1 **Challenges**

Raw block of alloy should be cut in such a way for mass production that reduces material loss. Also, More efficient system with no limit of number of nodes would increase efficiency of optimization and displaying more accurate results.

5.2 Future Plans

In future work more materials can be tried out like composite materials. Comparison between machined steering knuckle and forged knuckle can be carried out. Fatigue analysis can be aimed.

5.3 Conclusions

Optimization method used in this study in reducing the mass of the existing steering knuckle to 12.22%. This implies the first CAD model was over designed. Even if slighly optimized model would been investigated observing results the reduction of mass would have definitely been over 10%. The maximum stresses and displacement is within control and yielding a factor of safety around 2.8 to 3 necessary for such a crucial part in an automobile. Displacement is under 0.08mm and frequency obtained is at higher range thus eliminating cause of resonance.

REFERENCES

[1] M Mahesh , P. Sharma, S.A. Narayanan. “Static Analysis of Steering Knuckle and Its Shape Optimization” International Journal of Engineering Research and Application (IJERA) ISSN: 2248-9622 Vol. 2, Issue 3, May-Jun 2012, pp. 1493-1497 .

[2] Wan Muhamad, Farhad Nambhani, Askok Patole “Failure modes of a vehicle component designed for fuel efficiency” International Journal of Engineering Research and Application (IJERA) ISSN: 2248-9622 (National Conference on Emerging Trends in Engineering & Technology(VNCET-30 Mar’12).

[3] Prof. Purushottam Dumbre “Structural analysis of steering knuckle for weight reduction” Altair Engineering Inc., (2009), India.

[4] Faris Tarlochan et. al. “ Modeling, Simulation and Optimization Analysis on Steering Knuckle Component For Purpose of Weight Reduction”, MA: Addison-Wesley, 1989, ISBN 0-201- 15767-5.

[5] Viraj Rajendra Kulkarni and Amey Gangaram Tambe “Optimization and Finite Element Analysis of Steering Knuckle” In: Quagliarella D, Pe´riaux J, Poloni C, Winter G, editors. Genetic algorithms and evolution strategies in engineering and computer science. Chichester, England: Wiley; 1997, p. 245–66. ISBN 0-471-97710-1.

[6] Hisham Hamid et. al. “Design Improvement of Steering Knuckle Component Using Shape Optimization” (1997) AIAA Journal, vol. 35, no. 8, pp.1413-1415.

[7] Pe´riaux J, Sefrioui M, & Mantel P. G. multiple objective optimization strategies for electromagnetic backscattering”. In: Quagliarella D, Pe´riaux J, Poloni C, Winter G, editors. Genetic algorithms and evolution strategies in engineering and computer science. 1997, p. 225–43. ISBN 0-471-97710-1.

[8] Viraj Rajendra Kulkarni ”Steering control and optimization through Finite Element Analysis” PhD Thesis. City University, Measurement and Instrumentation Centre, Department of Electrical, Electronic and Information Engineering, School of Engineering. 1995, London (UK).

[9] Alex Richardson et. al. “Finite Element analysis of Steering Knuckle for weight reduction”. PhD Dissertation, Mechanical Engineering Department, Stanford University, 1995.

[10] Sefrioui M & Mantel P. G. “Multiple objective optimization strategies for electromagnetic backscattering”., Vol. 35, pp. 727-737.

[11] **L. Harzheim & G. Graf** "Topshape: an attempt to create design proposals including manufacturing constraints," (2002) International Journal of Vehicle Design, vol. 28, no. 4, pp. 389-409.

[12] Rao, S.S. “Engineering Optimization Theory and Practice”, John Wiley & Sons, Inc., 2009, 4th edition. Zurofi M. "Manufacturing Process Effects on Fatigue Design and Optimization of Automotive Components– An Analytical and Experimental Study", The University of Toledo, Ph.D. thesis, 2004.