Optimization and Finite Element Analysis of Steering Knuckle

Viraj Rajendra Kulkarni
Student
Vishwakarma Institute of Technology, Pune
666, Upper Indira Nagar, Bibwewadi,
Pune 411037, India

Amey Gangaram Tambe
Student
Vishwakarma Institute of Technology, Pune
666, Upper Indira Nagar, Bibwewadi
Pune 411037, India

Abbreviations: FEA: Finite element analysis, CAD: Computer Aided Drafting

Keywords: Optimization, Steering knuckle, FEA, Topology

Abstract

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. This paper focuses on optimization of steering knuckle targeting reducing weight as objective function, while not compromising with required strength, frequency and stiffness. Taking into consideration static and dynamic load conditions, structural analysis and modal analysis were performed. Finite element model was developed in HyperWorks. 10 node tetrahedral elements were used for meshing, providing better results in less time. 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.

Introduction

This paper investigates steering knuckle as a component for study. Geometry of knuckle depends upon type of suspension implemented. Additionally factors like type of brake calliper used, mounting of tie rod of steering sub-system influence the design. Suspension system in any vehicle uses different types of links, arms and joints to let the wheel move freely allowing them to turn. Steering knuckle and spindle assembly which might be two separate parts attached together or one complete part is one of the links.[2]. Reduction of weight has been one the critical aspects of any design. It has substantial impact on vehicle performance, fuel efficiency and in turn reduces the emissions.

Optimization methods are developed for manufacturing lighter vehicle. Optimization can be defined as the automatic process to make a system or component as good as possible based on an objective function and subject to certain design constraints. There are many different methods or algorithms that can be used to optimize a structure, on OptiStruct is implemented some algorithms based on Gradient Method. There are four types of optimization process namely:

i. Topology optimization which provides the optimum material layout according to design space and loading case in which design variables are defined as a fictitious density for each element, and these values are varied from 0 to 1 to optimize the material distribution

ii. Shape optimization which provides optimization of fillets and other outer dimensions.
iii. Size optimization which provides optimum thickness of the component
iv. Topography being an advanced form of shape optimization which generates reinforcements depending upon pattern of shape variable and design region.

Objective of this investigation is reduction of mass of the steering knuckle of a rear driven vehicle assuming double wishbone geometry of suspension including certain additional required parameters. Whereas methodology remains same for other geometries with some minor changes in design. This research focuses on topology and shape optimization. Finite element analysis has been used to implement optimization and maintaining stress and deformation levels and achieving high stiffness.

Process Methodology (details with figures)

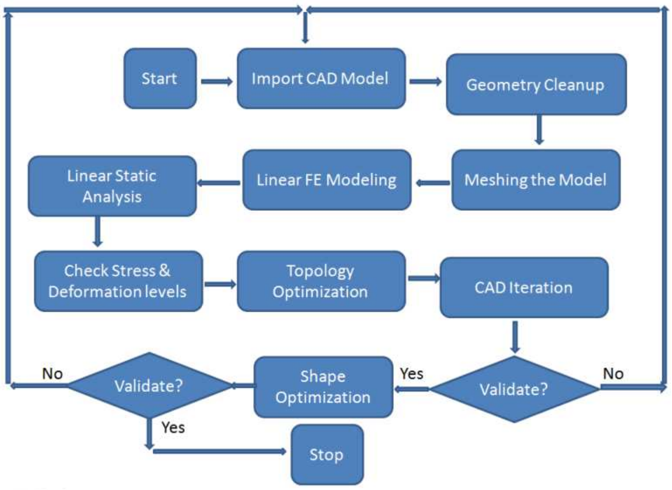

Figure 1: Showing Flowchart for Process Methodology

(A) Material Selection:

As per the material survey the best suited material was Aluminium alloy. The mentioned material was chosen as the material for steering knuckle due to its low density and compatible yield strength. This material was chosen for designing knuckle manufactured by machining comparing its results with different materials as structural steel, EN8, etc. Thus this paper focuses on optimizing aluminum knuckle. Table 1 and 2 indicate the material properties considered.

Silicon (Si)	Iron (Fe)	Copper (Cu)	Lead (Pb)	Bismuth (Bi)	Zinc (Zn)	Others (Total)	Aluminium (Al)
0.40 max	0.70 max	5.00 - 6.0	0.20 - 0.60	0.20 - 0.60	0.30 max	0.15 max	91.2 - 94.6%

Table1: Chemical Properties of Aluminium 2011 T3 Alloy

Young's Modulus	7.1e+4 MPa
Poisson's Ratio	0.33
Density	2770 kg/m3
Ultimate tensile strength	310Mpa
Yield Strength	280Mpa

Table 2: Physical and Mechanical Properties

(B) Designing a CAD model:

Required cad was developed using a 3D modeling software. The CAD geometry had basic requirement for stub hole, brake caliper mounting points, steering mounting point, outer points of upper and lower a-arms. Required mounting points were obtained from suspension geometry and steering geometry.

(C) Meshing:

Geometry clean up was performed prior to meshing the model. Finite element model was developed using HyperWorks. For better quality mesh combination of first and second order tetra elements were used. Surface meshing using triangular elements was performed to achieve better control on the meshing. Further this mesh was converted into a tetra mesh. Selective tetra elements were converted into second order and selective regions were finely meshed using first order elements controlling the number of nodes formed.

(D) Preliminary analysis:

To observe the maximum stresses, the model subjected to extreme condition and static structural analysis was carried out using HyperWorks and RADIOSS solver. Referring to figure 2 and table 3 a combined load of 1.5g braking force and 1.5g lateral acceleration were applied to the model considering the longitudinal load transfer during braking and lateral load transfer during cornering.

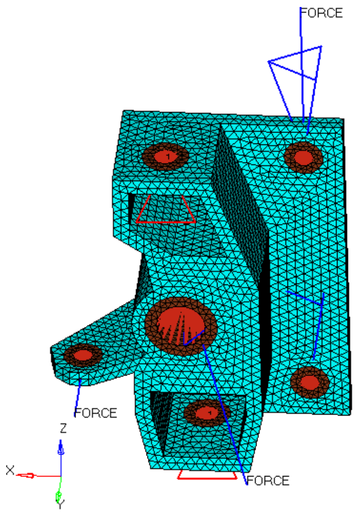

Figure 2: Showing Constraints and forces on meshed model

Mesh Statistics	
No. of nodes	8290
No. of elements	26085
Type of element	Tetrahedral elements Combination of 10 node tet elements and 4 node finer tet elements
Load Conditions	
Braking Force	1.5g
Lateral Force	1.5g
Steering force	Assumed for a steering effort of 40-50N
Self Weight	1g

Table 3 : Showing Mesh Characteristics and Load Conditions

The steering force from tie rod to steering plate on knuckle was analytically calculated using certain approximations and applied to the model with its self weight considered. The knuckle was constrained at the a-arm mountings. Two load steps were used. One for linear static analysis, while other for Normal Mode analysis. The completed FEA model was solved using RADIOSS Solver. Validation of data is crucial while performing analysis. Thus, the stress levels and deformation levels were checked for the worst case scenario. The modal frequencies were checked for six modes aiming comparatively high values to achieve better stiffness and avoid resonance with a given engine, concluding the preliminary analysis or the free run performed.

(E) Topology Optimization:

The model from static analysis step was imported in OptiStruct for optimization.

(E1) Response

Response for OptiStruct is any value or function that is dependent of the Design Variable and is evaluated during the solution. OptiStruct allows the use of numerous structural responses, calculated in a finite element analysis, or combinations of these responses to be used as objective and constraint functions in a structural optimization. Responses are defined using DRESP1 bulk data entries.

Responses considered for topology optimization were:

1. Mass response
2. Static displacement response for caliper mounting points
3. Static displacement response for tie rod mounting point.

(E2) Objective Function:

The Objective function is a model response to be maximized or minimized. There are two ways to specify an objective in OptiStruct. Either a single response can be minimized or maximized or you can choose to minimize the maximum value, or maximize the minimum value, of a number of normalized responses.

(E3) Design Constraint

On every engineering design there are constraints that need to be satisfied. These constraints can be defined as a lower bound or an upper bound on any response that is dependent of the design variable.

Design constraint used for model-

1.Upper bound for static displacement response for caliper mounting points.

2.Upper bound for static displacement response for tie rod mounting points

(E4) OptiStruct:

OptiStruct solves topological optimization problems using either the homogenization or density method. Under topology optimization, the material density of each element should take a value between 0 to 1, defining the element as being either void or solid, respectively. For topology optimization a constraint of max stress of 100 MPa was maintained.

The results obtained from the OptiStruct were interpreted very effectively using HyperView. The results obtained in the form of density change between 0 to 1. The regions having low densities were aimed for CAD iteration. These results helped in changing the CAD model so as to reduce the overall weight of the model.

Notice that there are some local regions where the stresses are still high. This is because topology stress constraints should be interpreted as global stress control or global stress target. The functionality has some ways to filter out the artificial or local stresses caused by point loading or boundary conditions, but those artificial stresses will not be completely removed unless the geometry is changed by shape optimization. Hence these stress hotspots will be addressed in the subsequent design fine tuning stage (shape optimization).

(F) CAD Iteration and Design Check Stage I:

Following the results from Topology Optimization changes were made in CAD model to reduce the weight of the model as well as maintaining the strength and rigidity of the model. Changes were made to remove the portion having relative density less than 0.3. Material was removed targeting %mass reduction from Topology optimization. This design check was carried to test the modified CAD assembly. The modified CAD model was subjected to same loading condition as mentioned in linear static analysis. Same material was taken into consideration. Optimization was carried out again to confirm any scope for further optimization. Aim was to carry out this process again until a scenario arises where regions of material removal are negligible. Results of Design Check were satisfactory.

(G) Shape optimization

Shape Optimization is an automated way to modify the structure shape based on predefined shape variables to find the optimal shape. Free-shape optimization uses a proprietary optimization technique developed by Altair Engineering Inc., wherein the outer boundary of a structure is altered to meet with pre-defined objectives and constraints. The essential idea of free-shape optimization, and where it differs from other shape optimization techniques, is that the allowable movement of the outer boundary is automatically determined, thus relieving users the burden of defining shape perturbations.

Free-shape design regions are defined through the DSHAPE bulk data entry. Design regions are identified by the grids on the outer boundary of the structure (the edge of a shell structure or the surface of a solid structure). These grids are listed on the DSHAPE entry. Free-shape optimization allows these design grids to move in one of two ways:

1. For shell structures; grids move normal to the surface edge in the tangential plane.
2. For solid structures; grids move normal to the surface.

During free-shape optimization, the normal directions change with the change in shape of the structure, thus for each iteration the design grids move along the updated normal. [1]

Shape optimization is used to stress relieving in some local regions where the stresses are still high. Stress was set as response, max stress as design objective reference and objective was to minimize maximum stress. The results of shape optimization were observed from HyperView. The required shape changes to reduce stresses were indicated along with the reduced stresses.

(H) CAD Iteration Stage II:

According to the results obtained from shape optimization the desired changes were made in CAD model to reduce the stresses in the model. Again this design check was carried to test the modified CAD assembly. The modified CAD model was subjected to same loading condition as mentioned in linear static analysis. Same material was taken into consideration. Results of Design Check were excellent and the model was finalized.

Results & Discussions

Fig 3 shows the contours of stress and displacement for preliminary analysis. 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.

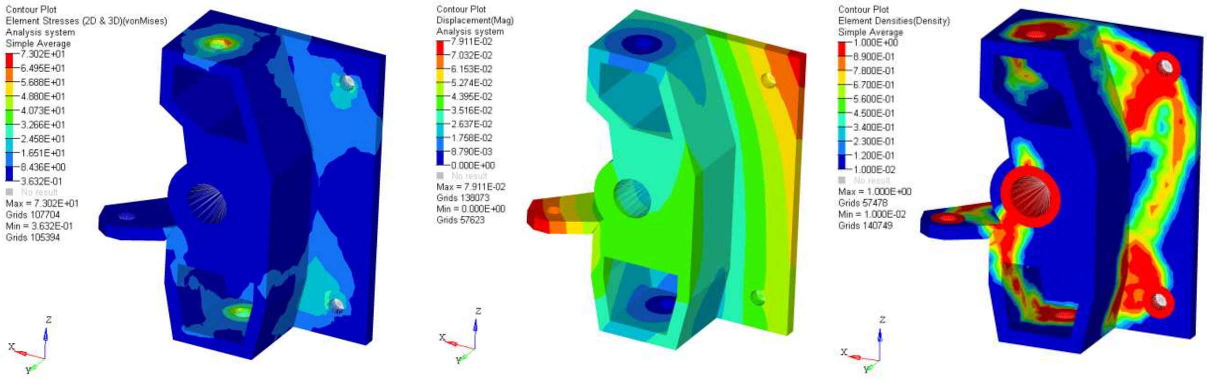

Fig 3: Contours of Stress, Displacement for preliminary analysis and Element densities for optimization from left to right

Fig 4 shows optimization performed per iteration. Initial mass being 1.3 kg (0.0013 tones) can be reduced to about 400 grams as per software. But checking geometric and manufacturing feasibility and stress values, optimization is done manually aiming less than 800 grams.

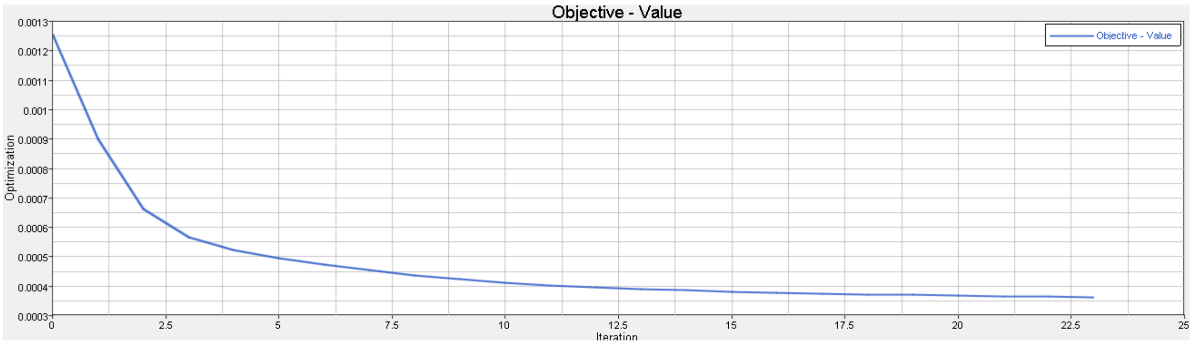

Fig 4: Graph Total mass in tones v/s iteration number

This completes first optimization. Now after CAD iteration stress values and displacement was checked. Referring figure 5 maximum stress is 103MPa, where as maximum displacement is 0.1mm.

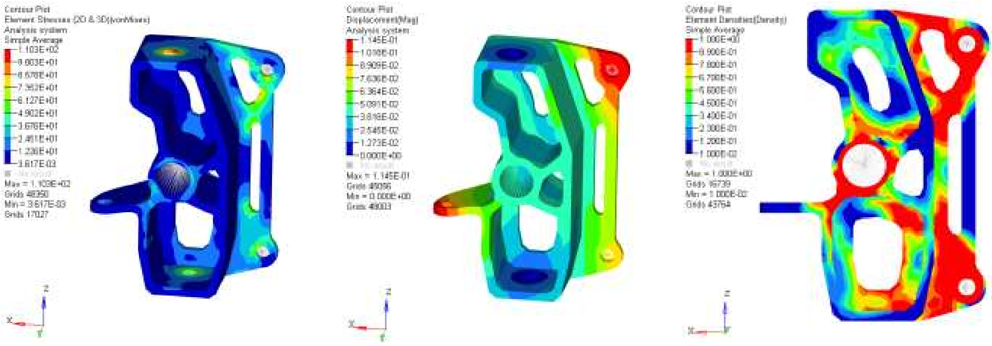

Fig 5: Contours of Stress, Displacement for Design Check after first Optimization and Element densities for next optimization from left to right

Observing the element densities contour in figure 5 indicates topology optimization is satisfactory and model is ready for shape optimization. The caliper mounting area has a vertical member seen with low density, but it has not been changed as majority of force acts along the member and adds to

improving life of the component reducing fatigue chances. Fig 6 shows the leftmost image as shape optimization output. It highlights the required reinforcement of material at the fillets near caliper mounting. Thus now inferring this result the CAD model was again changed and again stress levels and displacement are checked. 2nd image shows maximum stress as 96 MPa and 3rd image shows max displacement as 0.08mm which is excellent. After shape optimization stresses were reduced by an appreciable 7Mpa of existing 103Mpa as local stresses caused by point loading or boundary conditions during topology were removed by shape optimization.

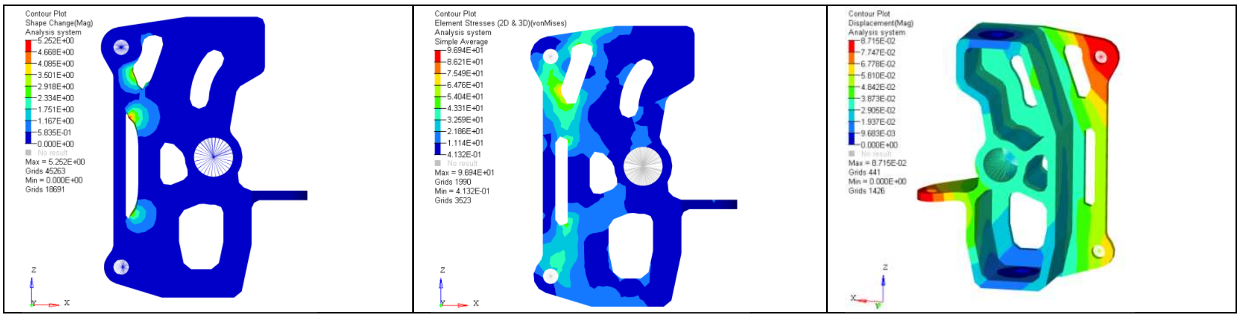

Fig 6: Contours of Shape change for shape optimization, Stress and Displacement of final Component from left to right

Benefits Summary

Resultant knuckle weighs only 730 grams. Thus this will have significant impact on vehicle performance reducing un sprung mass, improving fuel efficiency and reducing emissions.

Stages of Design	Weight (kg)	Stress (MPa)	Displacement	Frequency
I-Preliminary analysis	1.355	71	0.079	1300Hz
II-Analysis after Topology optimization	0.72	103	0.1	1630Hz
III-Analysis after Shape optimization	7.73	96	0.08	1870Hz

Table 4: Showing results of each stage of research

Challenges

Raw block of alluminium 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.

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.

Conclusions

Optimization method used in this study in reducing the mass of the existing steering knuckle to 53.33%. 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] Practical aspects of Finite element simulation-HyperWorks Student guide
[2] Wan Mansor Wan Muhamad, Endra Sujatmika, Hisham Hamid, & Faris Tarlochan,' Design improvement of steering knuckle using shape optimization' 2012
[3] Yannis Tsompanakis, Nikos D. Lagaros, Manolis Papadrakakis, Structural Design Optimization Considering Uncertainties: Structures & Infrastructures Book , Vol. 1, Series, Series Editor: Dan M. Frangopol
[4] Rao, S.S., “Engineering Optimization Theory and Practice”, John Wiley & Sons, Inc., 2009, 4th edition.