Optimization and Fatigue Analysis of Steering Knuckle
Atul Yadav¹ Abhijeet S Kabule²
¹M. E. Scholar ²Assistant Professor
1,2K.J College of Engineering & Management Research, Pune, India

Abstract— Light weight, low fuel consumption and high durability are the primary demands for any vehicle. The steering knuckle is a standoff amongst the most vital parts of a vehicle. The aim of this research is scale down the mass of an existing steering knuckle component of a local car model, using Creo 2.0, and performing its shape optimization, using Hyperworks as pre and post processor and Nastran as a solver, in order to meet the required strength attributes at the cost of minimum weight. The Investigation has been successful in optimizing the steering knuckle satisfying the required attributes and achieving a weight reduction of 8.81% in comparison with its initial model. Fatigue analysis is performed on the model to investigate the load bearing behavior of steering knuckle under when in operation.

Key words: Creo, Fatigue, Linear Static, Nastran, Steering Knuckle, Optimization

I. INTRODUCTION
Steering knuckle or hub carrier is a forging that usually includes the spindle and steering arm, and allows the front wheel to pivot. The knuckle is bestrode within the upper and lower ball joints on a Short Lower Arm (SLA) suspension, and between the strut and lower ball joint on a Macpherson strut suspension. Steering knuckle is the part of front axle assembly which fastens to the spindle and is held in place by the kingpin. This part is located at the front axle of a vehicle. It holds the wheel as it also hold the lower control arm, upper control arm, upper ball joint, and also the lower ball joint. If vehicles pass through a bad surface or cobblestone slalom, the road forces are then transferred into the knuckle. Due to this condition, some cases, these pivot have occurs fatigue effect. [1] The steering knuckle is the connection between the tie rod, stub axle and axle housing.

Fig. 1 Steering knuckle

II. OBJECTIVES
Objective of this investigation is reduction of mass of the steering knuckle of a vehicle 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 which will reduce the cost with respect to the mass production process. Fatigue analysis performed on the finite element mode to study the load-bearing behavior of steering knuckle under operating conditions.

The broad objectives of the project are:
1) To study the stress analysis of the existing steering knuckle.
2) To redesign the steering knuckle and optimize the same to avoid failure.
3) Fatigue life prediction of steering knuckle.

III. LITERATURE REVIEW
Viraj Rajendra Kulkarni his study 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.[2]

Parushottam Dumbre et al performed structural analysis on steering knuckle for static condition found out the stress level generated and used topology optimization to reduce the weight by 11% while meeting the strength requirement.[3]

Ameya Bhusari, Aditya Chavan and Sushant karmarkar shows a revised design which showed a 62.78% reduction in weight and 62.95% reduction in material volume as compared to the initial design; while having a factor of safety of 2.05. The overall weight of the vehicle can be reduced to achieve savings in costs and materials, as well as, improve fuel efficiency and reduce carbon emissions. The unsprung mass of the vehicle is reduced by 6.4kg which will contribute to improved vehicle handling. [4]

IV. METHODOLOGY
This research project investigated weight and opportunities and fatigue analysis that a SG Iron offer. The steering knuckle chosen for this project belonged to a passenger car. In this investigation, first a literature review on several aspects of steering knuckle in the areas of load and stress analysis, durability, manufacturing, economic and cost analysis, and optimization was carried out. First, the steering knuckle was digitized. Load analysis was performed based on the input from the design calculation performed. A linear static FEA at that point performed utilizing the results from load analysis to gain insight on the structural behavior of the steering knuckle and to determine the design loads for optimization. Component fatigue analysis was also performed to predict the durability of the optimized steering knuckle.

V. DESIGN OF STEERING KNUCKLE
While doing the analysis of Knuckle, forces to be considered are – Self-weight, Braking, Steering and
Cornering/Lateral forces. So as to keep up the sturdiness of the design, the model is subjected to extreme conditions as suggested by Sharma [2], considering Generic (G) forces when all the forces are considered to be acting simultaneously. The weight of the vehicle for this investigation is 1185kg. Considering the weight, the magnitude of each of the forces were ascertained and depicted in the table I.

**B. Force**

Wheels load constituents in the X, Y and Z axis. These loads are due to the response because of footing, vertical response because of vehicle weight and directing response. 

\[
X \text{ - axis } = Y \text{ - axis } = 3G = 3 \times 9.81 \times 296.25 = 8781.6377 \text{ N} \\
Z \text{ - axis } = 1G = 1 \times 9.81 \times 296.25 = 2906.2125 \text{ N}
\]

Since all load in X, Y, and Z direction are perpendicular to each other, the resultant of all the forces s given by,

\[
F = \sqrt{X^2 + Y^2 + Z^2} \\
F = \sqrt{8781.6375^2 + 8781.6375^2 + 2906.2125^2} \\
F = 12667.87 \text{ N}
\]

VI. **FINITE ELEMENT ANALYSIS OF STEERING KNUCKLE**

To watch max stress produce into steering knuckle, model is submitted to extreme consideration and analysis is carried out. The loading and boundary condition is shown in figure 3.

### Table 1: Units for Magnetic Properties

<table>
<thead>
<tr>
<th>Load in X-direction</th>
<th>Load in Y-direction</th>
<th>Load in Z-direction</th>
</tr>
</thead>
<tbody>
<tr>
<td>3G</td>
<td>3G</td>
<td>1G</td>
</tr>
</tbody>
</table>

There are two sorts of burden following up on knuckle i.e. force moment. This knuckle is intended for vehicle of 1185 kg weight. Braking following up on it produces moment. By measurement perpendicular distance is 92mm

**A. Moment**

According to the weight distribution of the car the weight acting on each wheel is 296.25kg. Braking following up on it produces moment. By measurement perpendicular distance is 92mm

\[
\text{Braking force} = 1.5G = 1.5 \times 9.81 \times 296.25 = 4359.318 \text{ N} \\
\text{Moment} = \text{braking force} \times \text{perpendicular distance} = 4359.318 \times 92 = 401057.325 \text{ Nmm}
\]

This moment is acting on steering knuckle where break caliper is mounted. Brake calliper is mounted at three locations therefore distributing moment at three points, Therefore, moment at each point is

\[
\text{Moment} = 401057.325 \div 3 = 133685.775 \text{ Nmm}
\]
From the analysis it can be seen that the maximum stress is induced on the brake calliper area and it is well below the yield stress of the material hence the design is safe. Therefore the analysis allows the objective of dissertation as to reduce the weight of the steering knuckle by using structural optimization.

VII. STRUCTURAL OPTIMIZATION

The optimization opted here is shape optimization. Shape optimization is a mechanized approach to change the structure shape in light of predefined shape variables to locate the ideal shape. DVs are utilized to change the geometry shape of the component, on HyperMesh it is used HyperMorph to define this parameter.

In the optimization process step by step iterations are taken to minimize the weight with Von Mises stress as constraints. The areas showed inside the box in fig 6 are the area from where the material has been removed in each iteration. The analysis is done for the different optimized steering knuckle for the stress verification. The loading and boundary condition are kept same as used in initial model i.e. before optimization.

VIII. EXPERIMENTAL VALIDATION

The component produced for the experimentation is made up of SG is used for the testing. The component is fixed using fixture and load is applied.

The Test Report for the component for verifying the results with the Analytical method of analysis are compared it is observed that the results are much closer. Typically, depending on the type of Test and the application, an error margin or about 5 to 10% could be considered close towards validating the proposed design.

The stress determined by FEA is 156.045 Mpa and the stress determined by the test is about 168.88 Mpa. Hence the percentage variation in result is 6.94%.
IX. FATIGUE ANALYSIS

Steering knuckle fatigue life calculated is based on stress-life approach. The fatigue life of 5938164.5 cycles which is predicted in analysis, shows that crack initiation will start after these numbers of cycles.

X. RESULT AND DISCUSSION

The steering knuckle is exposed to its maximum loading condition and a stress of 154.8 MPa is generated which is less than the yield strength of the material, hence we can say that it component is safe under calculated loading. After structural optimization stress reduction of 1.56% is observed in the final model. Displacement in each of the model is observed is relatively same. The weight of the component is one the major domain area; the automobile is working as it influence the automobile economy, pollution and the components manufacturing cost. Hence a virtual optimization task was performed and using shape optimization a reduction of 8.81% is achieved. Fatigue analysis predicted the minimum cycles as 5938164.5 before crack starts.

<table>
<thead>
<tr>
<th>Attributes</th>
<th>Von Mises Stress (MPa)</th>
<th>Deformation (mm)</th>
<th>Weight (kg)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Original</td>
<td>158.480</td>
<td>0.150</td>
<td>2.769</td>
</tr>
<tr>
<td>Iteration 1</td>
<td>367.300</td>
<td>0.257</td>
<td>2.433</td>
</tr>
<tr>
<td>Iteration 2</td>
<td>251.200</td>
<td>0.267</td>
<td>2.489</td>
</tr>
<tr>
<td>Iteration 3</td>
<td>156.045</td>
<td>0.153</td>
<td>2.525</td>
</tr>
</tbody>
</table>

Table 2: Stress and Displacement Induced in Various Modified Steering Knuckle

XI. CONCLUSION

The following conclusions can be drawn from this investigation:
1) The existing and modified design is modeled using modeling software and various attributes are obtained and the results are taken and compared.
2) The working stress developed in the steering knuckle is 158.8MPa which is less than the yield stress which improves the design life of the steering knuckle.
3) To improve performance geometry has been modified using shape optimization which enables to reduce stress levels marginally below yield limit.
4) Reduction of weight was one of our primary aims. We found that weight can be reduced using structural optimization process. After applying load and design constraints, stress analysis and shape optimization was performed a weight reduction of 8.81% is observed.
5) Fatigue analysis predicted the minimum cycles as 5938164.5 before crack starts.

ACKNOWLEDGMENT

I express my sincere thanks to Prof. P.E Ingale (head of the department mechanical engineering) for his kind cooperation for presenting this paper. I additionally extend my genuine on account of every single other individual from the workforce of mechanical building division and my companions for their co-operation and consolation.

REFERENCES

[1] Mr. V.Sivananth Dr. S.Vijayarangan Mr. R.Aswathaman “Fatigue and Impact Analysis of Automotive Steering Knuckle under Operating Load Cases” Altair Technology conference 2015 India.
Conference on Mathematical Methods, Mathematical Models and Simulation in Science and Engineering.


