Fatigue Life Estimation of a Steering Knuckle

#1 Pawar Abhishek, #2 Pathak Charudatta

1 apawar53@gmail.com
2 cspathak.scroe@gmail.com

#1,2 Department of Mechanical Engineering, Savitribai Phule Pune University
Sinhgad College of Engineering, Vadgaon (Bk.), Pune-41

ABSTRACT

Steering knuckle is a prominent component of an automobile steering system. It is subjected to cyclic loads from the suspension system during service life. Fatigue behavior is therefore a key consideration in its design and performance evaluation. Geometric modeling and stress analysis of steering knuckle using finite element analysis software is done and analyzed. This paper analyzes estimation of fatigue life of steering knuckle made from forged steel without induction hardening. Steering knuckle is determined to be induction hardened with varying case depth. Effect of induction hardening on fatigue life is analyzed.

Keywords— Finite Element Analysis, Fatigue life, Induction hardening, Damage factor, Steering knuckle.

I. INTRODUCTION

Steering knuckle is one of the important component of vehicle which is connected to steering, suspension and brake to chassis of vehicle. It undergoes different loading under different conditions. There has been a strong trend towards the adoption of optimum materials and components in automotive industry. Automotive designers have a wide range of materials and processes to select from. Steel forgings are in competition with aluminum forgings and castings, cast iron, and sintered powder forgings. The competition is particularly acute in the chassis, and it is not unusual to find a range of different materials and manufacturing technologies employed within modern chassis components. Steering knuckle is a prominent component in car which actually takes the loads from the wheels and transfers these forces to the suspension system. Steering knuckle is main important part in the vehicle because that requires lots of attention in selection because once it is damaged then it have to replace with 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. Depending on the vehicle and suspension design, the steering hub or spindle will also vary slightly. In the case of all wheel drive or front wheel drive vehicles, this device will also be the location where the power steering comes into play as well.

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. 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 techniques, etc.

© 2015, IERJ All Rights Reserved
II. LITERATURE SURVEY

Mahesh P. Sharma paper deals with 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. In this paper we have done static analysis of steering knuckle. We have design a knuckle which accommodates dual caliper mountings for increasing braking efficiency & reducing a stopping distance of a vehicle. CAD modal of knuckle was prepared in CREO2.0. Static analysis was done in ANSYS WORKBENCH by constraining the knuckle, applying loads of braking torque on caliper mounting, longitudinal reaction due to traction, vertical reaction due to vehicle weight and steering reaction. Also, reducing the weight of vehicle component plays vital role in increasing efficiency of vehicle and reducing fuel consumption. In this paper they have also done shape optimization of same knuckle and saved material resource. Shape optimization of knuckle was done using ANSYS WORKBENCH making objective function as reducing weight. These FEA results are verified by comparing with analytical calculations. Considering these results modal is modified.

Mehrdad Zoroufi and Ali Fatemi performed analysis on steering knuckle undergoes time-varying loadings during its service life. Fatigue behavior is therefore, a key consideration in its design and performance evaluation. This research program aim assess fatigue life and compare fatigue performance of steering knuckles made from three materials of different manufacturing processes. These include forged steel, cast aluminum, and cast iron knuckles. In light of the high volume of forged steel vehicle components, the forging process was considered as base for investigation. Monotonic and strain-controlled fatigue tests of specimens machined from the three knuckles were conducted. Static as well as baseline cyclic deformation and fatigue properties were obtained and compared. In addition, a number of load-controlled fatigue component tests were conducted for the forged steel and cast aluminum 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. It is concluded that the forged steel knuckle exhibits superior fatigue behavior, compared to the cast iron and cast aluminum knuckles.

III. MODELLING

The CAD model of steering knuckle is prepared based on standard nomenclature of given design. Figure shows the CAD model of steering knuckle before optimisation.

A. Load calculation

- road bump case
- raking case
- cornering case

Car wheel designation: (P215/75 R15)

- turning radius: - 6.35 meter
- gross vehicle weight (GVW): - 2200kg
- ass of the vehicle is splitted –
- ass of front wheel = 2200/4= 550kg

Road bump case

Let, Speed of vehicle = 50km/hr (13.9m/s)

\[
U = \frac{x}{t}
\]

\[
t = \frac{1}{13.9} = 0.072 \text{ m/sec}
\]

\[
U_{vertical} = \frac{0.15}{0.072} = 3.47 \text{ m/s}
\]

\[
A_{vertical} = \frac{U_{vertical}}{time} = 48.2 \text{ m/sec}^2
\]
Wheel acceleration force (inertia force) = mass X acceleration  
\[ = 550 \times 48.2 \]
\[ = 26510 \text{ N} \]

**Braking case**

Vehicle deaccelerates (i.e. braking) at a constant 0.5 G

\[ \text{Braking force} = \text{mass X acceleration X0.5 G} \]
\[ = 550 \times 9.81 \times 0.5 \]
\[ = 2697.75 \text{ N} \]

- Braking force acts at a distance from the centre of hub & is therefore a ‘Moment’
- The distance is dynamic radius \( R \) of wheel which is approximately 0.335 m

**Braking moment = Braking force X dynamic radius**
\[ = 2697.75 \times 0.335 \]
\[ = 903.75 \text{ N-m} \]

This moment is transferred to the two brake calliper which are at 101.25 mm distance from the centre of hub & at a 450 from vertical axis. Therefore the force is split into half to be applied equally on both holes & analysed to two components.

\[ F_b = \frac{903.75}{0.10125} = 8926 \text{ N} \]
\[ F_{1/2} = \frac{8926}{2} = 4463 \text{ N} \]

Re-solving the component:

\[ F_x = 4463 \cos(45) = 3155.8 \text{ N} \]
\[ F_y = 4463 \sin(45) = 3155.8 \text{ N} \]

**Cornering case**

Let, vehicle travels around a corner of 10 m radius at a speed 20 km/hr (i.e. 5.56 m/sec)

\[ F_{\text{centripetal}} = \frac{mv^2}{r} \]
\[ = \frac{550 \times 5.56^2}{10} \]
\[ = 1700.25 \text{ N} \]

**IV. FINITE ELEMENT**

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.

![Meshing of Model](image)

The cad model in IGES format is imported in Hyper-Mesh for the preparation of FE model. Then geometry cleanup was done by using options like ‘geom. cleanup’ and ‘defeature’ to modify the geometry data and prepare it for meshing operation. This process involves deletion of curvature of very small radius (less than 2mm) which has less structural significance. Mixed type of elements which contains quadrilateral as well as triangular elements, have been used in analysis. These 2D elements are converted into 3D tetra elements. The sensitive regions have been remeshed by manually considering the shape and size of the parts. Quality check of all the elements has been performed and mesh is accordingly optimized.

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s Modulus, ( E )</td>
<td>210 GPa</td>
</tr>
<tr>
<td>Poisson’s Ratio, ( \nu )</td>
<td>0.29</td>
</tr>
<tr>
<td>Density, ( \rho )</td>
<td>7850 kg/m³</td>
</tr>
</tbody>
</table>
**Stress Life Approach:**

RADIOSS uses the S-N approach for calculating the fatigue life. The S-N approach is suitable for high cycle fatigue, where the material is subject to cyclical stresses that are predominantly within the elastic range. Structures under such stress ranges should typically survive more than 1000 cycles. The S-N approach is based on elastic cyclic loading, inferring that the S-N curve should be confined to numbers greater than 1000 cycles. This ensures that no significant plasticity is occurring. This is commonly referred to as high-cycle fatigue.

Since S-N theory deals with uniaxial stress, the stress components need to be resolved into one combined value for each calculation point, at each time step, and then used as equivalent nominal stress applied on the S-N curve.

In RADIOSS, various stress combination types are available with the default being “Absolute maximum principle stress”. In general “Absolute maximum principle stress” is recommended for brittle materials, while “Signed von Mises stress” is recommended for ductile material. The sign on the signed parameters is taken from the sign of the Maximum Absolute Principal value.

**The fatigue material properties: (S-N curve):**

From above plot the maximum displacement value for lower control arm is 0.3 mm which is very less hence the design for steering knuckle is safe.
From above plot the maximum stress value for steering knuckle is 200 MPa which is less than yield strength hence the design for lower control arm is safe.

Fatigue Damage is a contour plot of the fatigue damage at a given design life. Fatigue damage is defined as the design life divided by the available life. This result may be scoped. The default design life may be set through the Control Panel. For Fatigue Damage, values greater than 1 indicate failure before the design life is reached.

V. CONCLUSION
The steering knuckle has been modelled using catia and analyzed using Hypermesh. The various structural parameters such as Nodal displacements, Stress distribution and fatigue parameters like damage and fatigue life are completely analyzed and studied. This study shows that the areas where the stress concentration is maximum due to the applied load and the portions that has to considered in the design of steering knuckle in order to avoid frequent failures to improve its fatigue life.

ACKNOWLEDGMENT
The author would like to thank the Department of Mechanical Engineering, Savitribai Phule University (SPPU) for their support in this work.

REFERENCES