Mechanics and Mechanical Engineering
Vol. 20, No. 1 (2016) 5–13

Fatigue Analysis of Automative Steering Knucle

S. Madhusudhanan
Department of Mechanical Engineering
Dr.Mahalingam College of Engineering and Technology
Pollachi Tamilnadu, India
madhufea@gmail.com

K. Arun Kumar
V. Praveen Kumar
T. Mahendran
Department of Mechanical Engineering
V.S.B Engineering college
Karur, Tamilnadu, India
arunmailtech@gmail.com
leomahe001@gmail.com

Received (22 December 2015)
Revised (16 January 2016)
Accepted (24 January 2016)

Steering knuckle plays a vital role in automobile which connects the suspension, steering part, wheel hub and brake to the chassis. During its working, it subjected to various loading conditions. so, it requires high accuracy,, quality, and durability. The main theme of this work is to access the fatigue performance of a steering knuckle. This can be performed by a detailed load analysis. Therefore, this study requires two steps. First part of the study involves modeling of the steering knuckle with the design parameters using the latest modeling software CATIA and the second part involves in fatigue analysis using ANSYS WORKBENCH. This provides us to improve the overall knowledge about the component.

Keywords: Steering knuckle, static analysis, fatigue analysis.

1. Introduction

In your vehicle the joint that allows the steering arm to turn the front wheels called steering knuckle. The forces applied on this component are of cyclic nature as the steering arm is turned to maneuver the vehicle to the left or to the right and to the centre again. The linking parts of the steering system and the suspension system have a direct impact on the performance of the vehicle's ride, durability, and steerability. So, the performance of these parts is directly compared to the quality of the vehicle. For this purpose, the strength of the knuckle under the maximum service loads is calculated and the fatigue life is analyzed. Under the same condition, the strain order formation is also calculated and it is verified that this value is within the allowable value.

2. Material testing

The test specimens for the tensile test as per ASTM A746 standard have been taken for knuckle arms as shown in Fig. 1. The stress strain relationship was collected and the graph has been formed. Fig. 2 indicates the stress strain relation of the steering knuckle material. The microstructure of a component was found using the optical microscope is shown in Fig. 3. The graphite's structure is in spherical shape and the ferritic matrix formation is also shown in

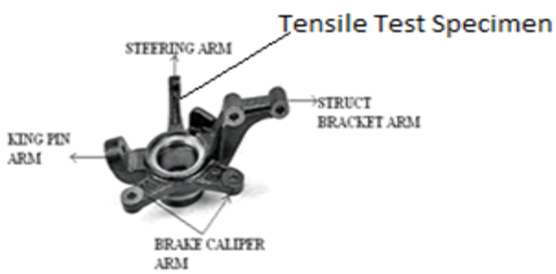

Figure 1 Test specimen for knuckle

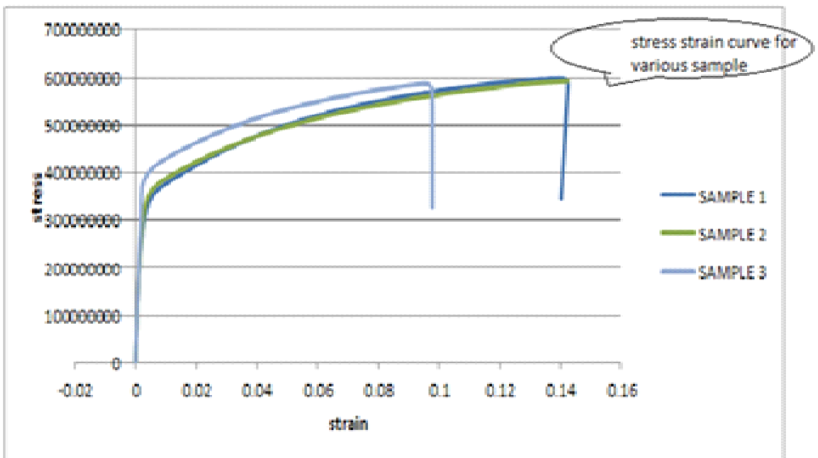

Figure 2 Stress strain relationship graph

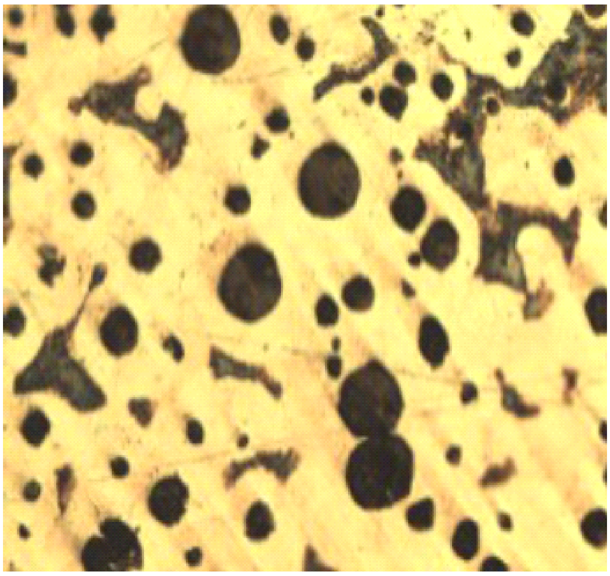

Figure 3 Microstructure of ferrite matrix – SG iron knuckle

3. Design Methodology

Steps Involved in Methodology is given below:
Step 1: Modeling of steering knuckle using 3D modeling software.
Step 2: Finite element modeling of the steering knuckle.
Step 3: Analysis of steering knuckle using ANSYS software:
- element selection,
- discretization,
- mesh generation.

Step 4: Finite element static analysis.
Step 5: Fatigue analysis

4. Finite element modeling

The structures of steering knuckle parts are shown in the Fig. 1; modeling of the component was created using CATIA. Here element type used was tetrahedral element with 70323 nodes and 41067 elements. The boundary condition and meshed model is shown in Fig. 4 and Fig. 5.

5. Analysis

The static analysis was done by using ANSYS WORKBENCH metric converter ProductID15. In15. In the preprocessor, the element type solid 186 was chosen for the knuckle component. Because of elasto–plastic nature of material, the result varies with linear analysis to non–linear analysis.

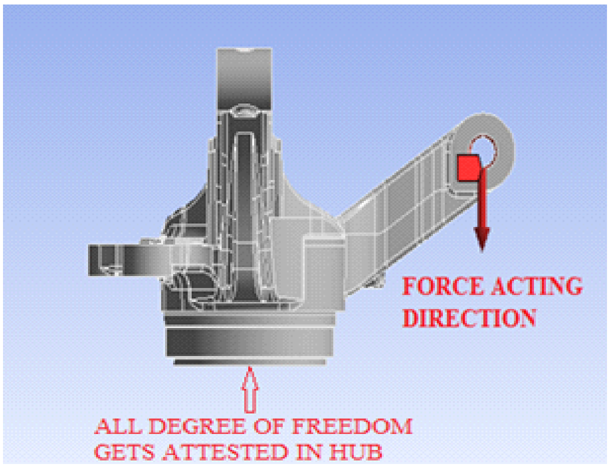

Figure 4 Boundary conditions

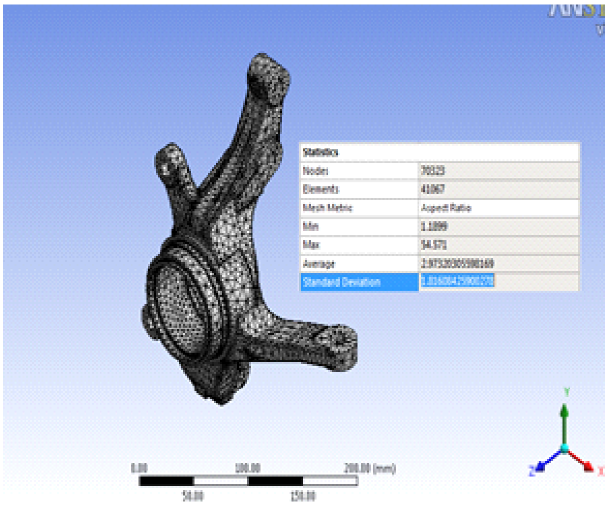

Figure 5 Meshed model of steering knuckle

So in this analysis, material nonlinearity was considered and the average stress strain relation curve data obtained from tensile test is given as input to the software.

The material properties Young's modulus (E) is 1.6E5 N/mm^2 and the Poisson ratio is 0.28. In ANSYS, the Newton–Raphson algorithm was chosen for non–linear analysis.

$$[K]\{u\} = \{F\}$$

Regarding the fatigue analysis, the knuckle arm was analyzed by means of a curve based on the strain–life method. The various stain life parameter called strength exponent, strength co efficient, ductility exponent, ductility coefficient and strain hardening parameter were calculated. The knuckle was analyzed for about 10^6 cycles. In the strain–life approach, the values of stress and strain at the critical location were used to find strain life is given below equation.

$$\frac{\delta \in}{2} = \frac{\sigma'_f}{E}(2N_f)^b + \sigma'_f(2N_f)^c$$

Morrow stain life theory of approach states that increase in effect of mean stress leads to decrease the elastic strain amplitude. Based on the morrow parameter which considers the mean stress effect equation is given below.

$$\frac{\delta \in}{2} = \frac{\sigma'_f - \sigma_m}{E}(2N_f)^b + \sigma'_f(2N_f)^c$$

The fatigue life of component can be obtained using Basquin equation with the help of material properties.

$$\sigma N_f = \sigma'_f(2N_f)^b$$

The factor of safety can be obtained by below equation

$$fos = \frac{1}{\frac{\sigma_a}{\sigma_{-1}} + \frac{\sigma_m}{\sigma_u}}$$

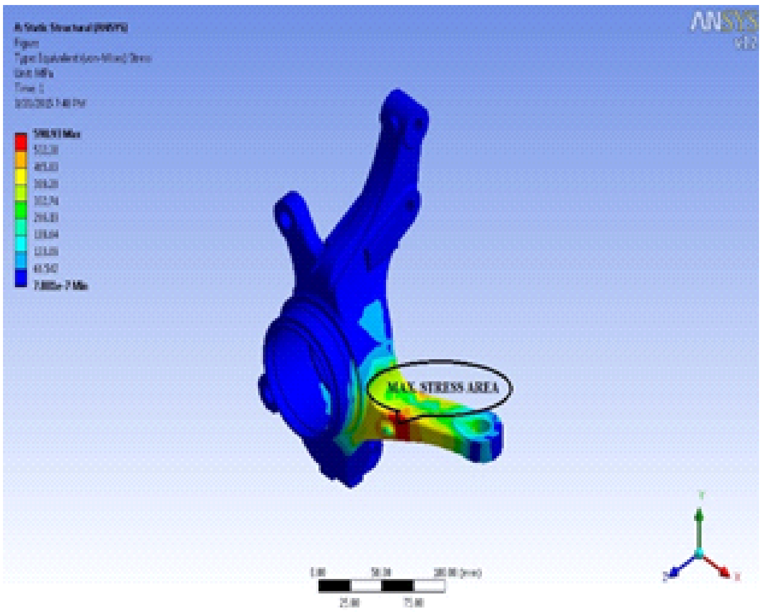

Figure 6 Von Mises stress

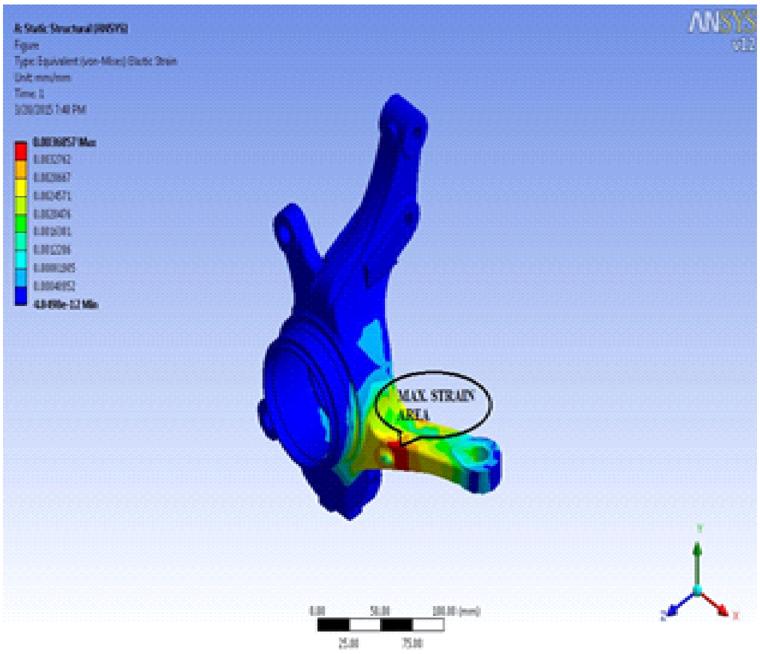

Figure 7 Elastic strain

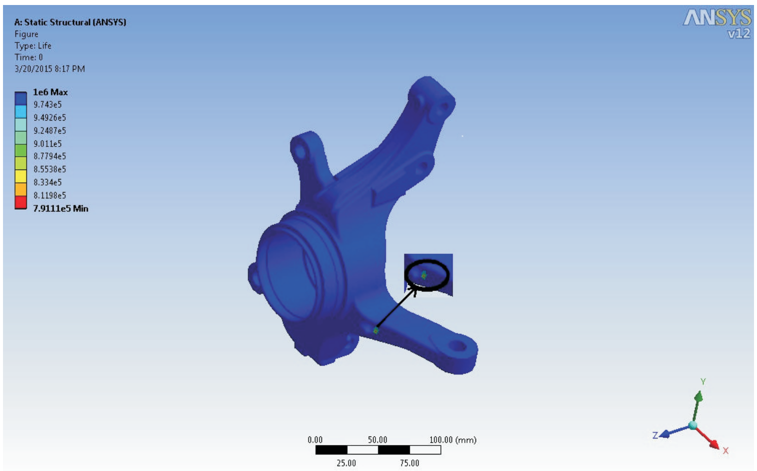

Figure 8 Fatigue life

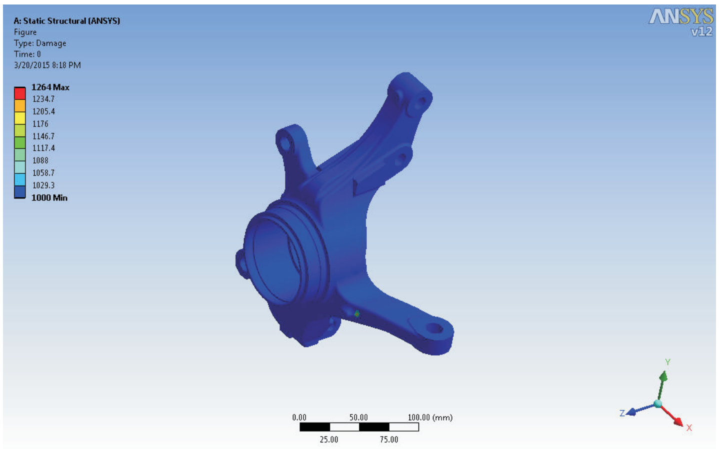

Figure 9 Fatigue damage

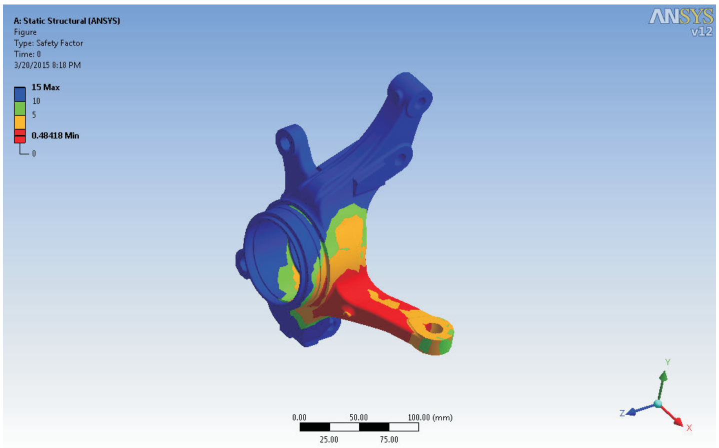

Figure 10 Safety factor

6. Results and discussion

6.1. *Virtual analysis results*

The maximum stress and maximum strain are shown in Fig. 6 and Fig. 7 for the static analysis. The maximum stress concentrating area and maximum strain affecting area are shown in rounded region. The tends to withstand about 598 MPa of stress and strain of about 0.003 which is shown.

The fatigue properties of the analyzed result are shown in below Figs 8–11. These say that the component having load factor below 0.5 exhibit good fatigue strength.

The safety factor also below 1. so the design was safe. The biaxiality indication of the component also shown in below Fig. 11.

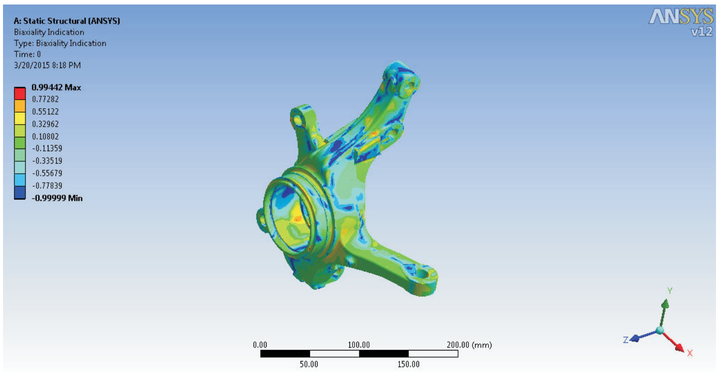

Figure 11 Biaxiality indication

7. Conclusion

Fatigue assessment is performed to find the damage by the applied loads and the number of load cycles versus S–N curves. The failure in the steering arm is indicated and the following results were made:

1. The maximum equivalent stress and the plastic strain in the static analysis were 598 MPa and 0.003, which was below ultimate strength of the material.

2. The stiffness analysis result shows that there was some difference in the loading conditions; most of the steering arm parts involved the largest equivalent plastic strain.

3. The minimum life in the fatigue analysis was 790,000 cycles which was the safe. By controlling the loading factor below the 0.5 we can easily improve the fatigue life of the component and also the safety factor was lies below 0.5 which states design was safe.

4. By means of doing the post heat treatment of the material we can easily enhance the strength thereby there is a chance to get better fatigue life of the component.

References

[1] **Kumar Chandrakar, S., Lal Soni, D. and Gardia, S.**: FEA of a steering knuckle for life prediction, pp. 681–688, **2013**.

[2] **Patel Akash, A.**: Fatigue or durability analysis of steering knuckle, *Small Engine Technology Conference Proceedings*, Society of Automotive Engineers of Japan, Tokyo, pp. 379–389, **2013**.

[3] **Jhala, R. L. and Kothari, K. D.**: Component fatigue behavior and life prediction of a steering knuckle using FEA, pp. 646–657, **2009**.

[4] **Dumber, P.O. and Kweon, S.S.**: Stress Analysis On Steering Knuckle Of The Automobile Steering System, *Materials and Design*, Vol. 28(4), 1198–1210, **2014**.

[5] **Kamal, M., Mustafizur Rahman, M. and Sani, M.S.M.**: Application multibody Simulation For Fatigue Life Estimation, Vol. 7, pp. 912–923, **2013**.

[6] www.sciencedirect.com

[7] www.ijret.com

Nomenclature

K – stiffness matrix of the material
u – displacement matrix
F – applied force in KN
$\Delta\varepsilon$ – strain amplitude
E – young's modulus in MPa
N_f – No of cycles to failure
b – fatigue strength exponent
c – ductility exponent
σ_{N_f} – fatigue life