

FATIGUE ANALYSIS OF AUTOMOTIVE STEERING KNUCKLE

K.Arun kumar¹,V.Praveen kumar²,T.Mahendran³

Department of Mechanical Engineering,

¹Assistant professor,V.S.B Engineering college,Karur,Tamilnadu, India

²Assistant professor,V.S.B Engineering college,Karur,Tamilnadu, India

³Assistant professor,V.S.B Engineering college,Karur,Tamilnadu, India

*¹arunmailtech@gmail.com,²shasha.praveen@gmail.com,³leomahe001@gmail.com

Abstract

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.

Keyword: steering knuckle, static analysis, fatigue analysis.

Nomenclature

K	:	stiffness matrix of the material
u	:	displacement matrix
F	:	applied force in KN
$\Delta\epsilon$:	strain amplitude
E	:	young's modulus in Mpa
N_f	:	no. of cycles to failure
b	:	fatigue strength exponent
c	:	ductility exponent
σ_{Nf}	:	fatigue life

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 Fig 3

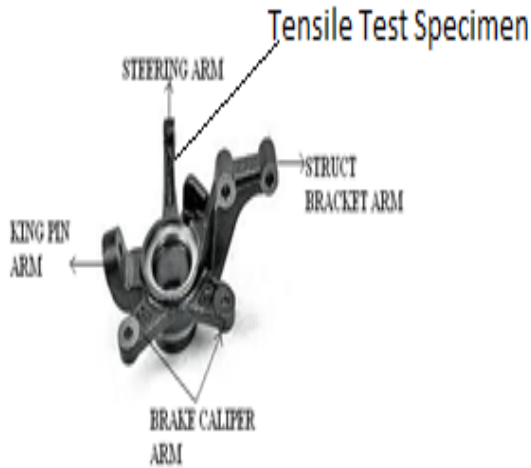


Fig.1. Test Specimen For Knuckle

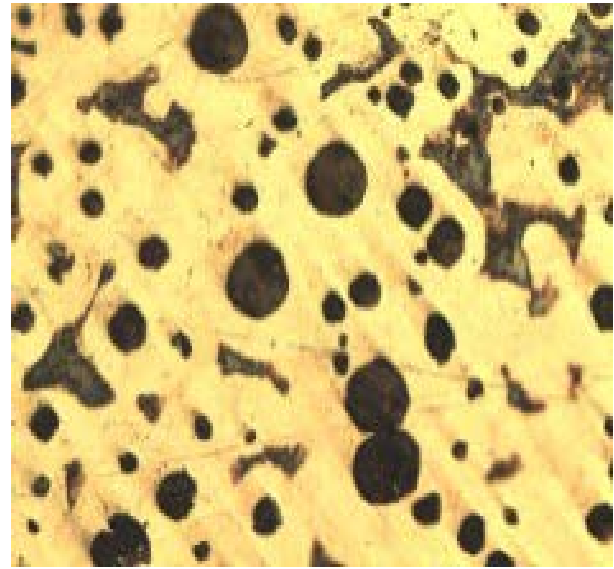


Fig. 3. Microstructure of ferrite matrix-SG iron knuckle

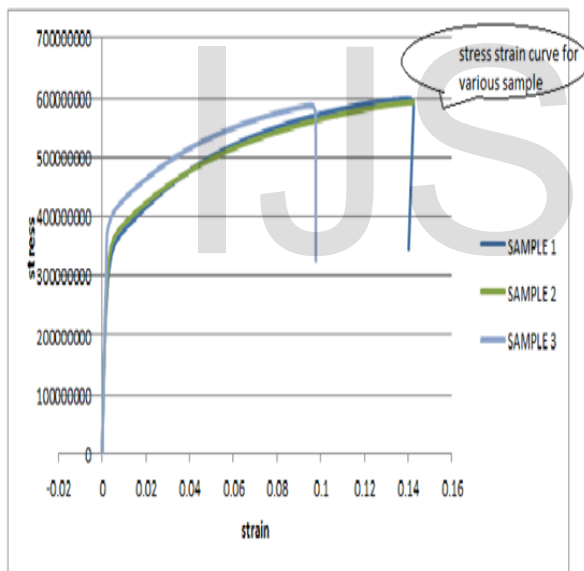


Fig.2. Stress strain relationship graph

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.

- i. Element selection
- ii. Discretization
- iii. 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

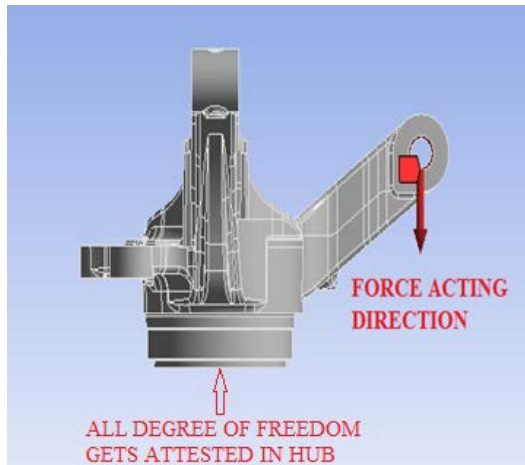


Fig.4. Boundary Conditions

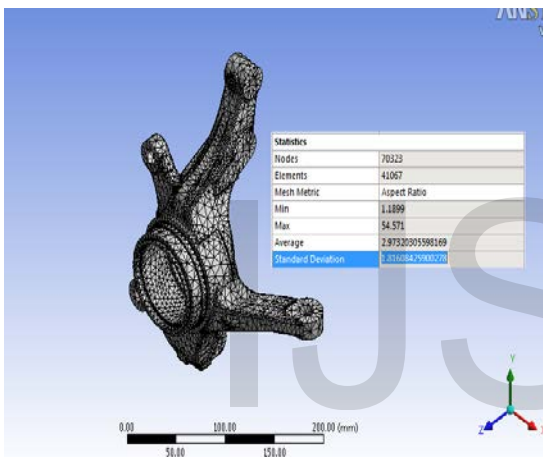


Fig. 5. Meshed model of steering knuckle Analysis

The static analysis was done by using ANSYS WORKBENCH 15. 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. 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 $1.6E5 \text{ N/mm}^2$ 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 coefficient, 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.

$$\frac{\Delta \epsilon}{2} = \frac{\sigma_f'}{E} (2N_f)^b + \sigma_f' (2N_f)^c$$

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.

$$\frac{\Delta \epsilon}{2} = \frac{\sigma_f' - \sigma_m}{E} (2N_f)^b + \sigma_f' (2N_f)^c$$

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

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

5. Results and Discussion

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 598Mpa of stress and strain of about 0.003 which is shown.

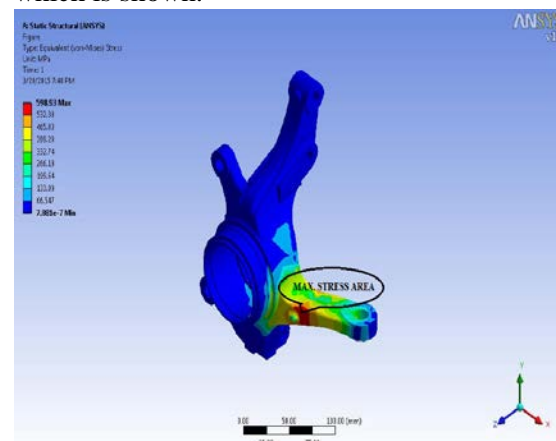


Fig.6. Von-mises stress

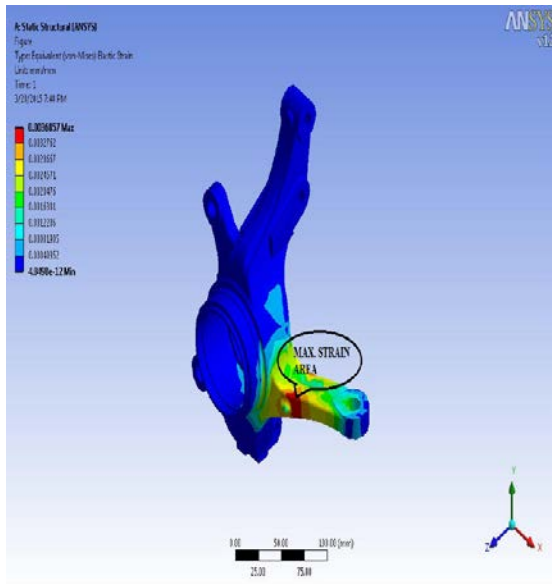


Fig.7.Elastic strain

The fatigue properties of the analyzed result are shown in below Fig8-11. These say that the component having load factor below 0.5 exhibit good fatigue strength.

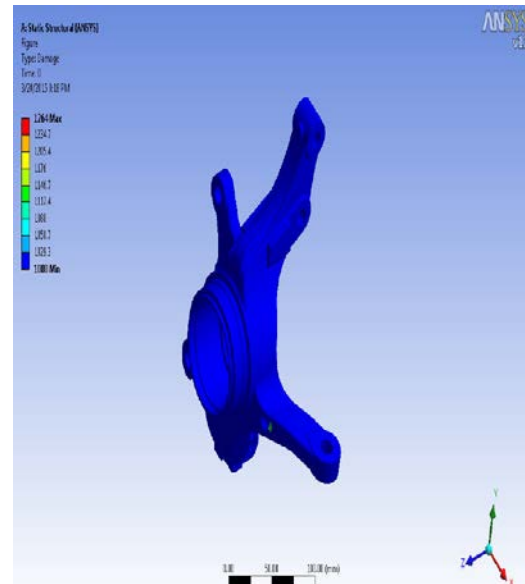


Fig.9.Fatigue damage

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

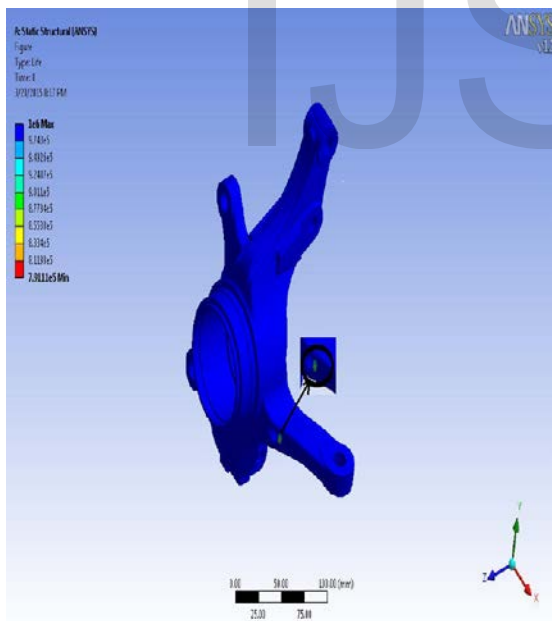


Fig.8.Fatigue life

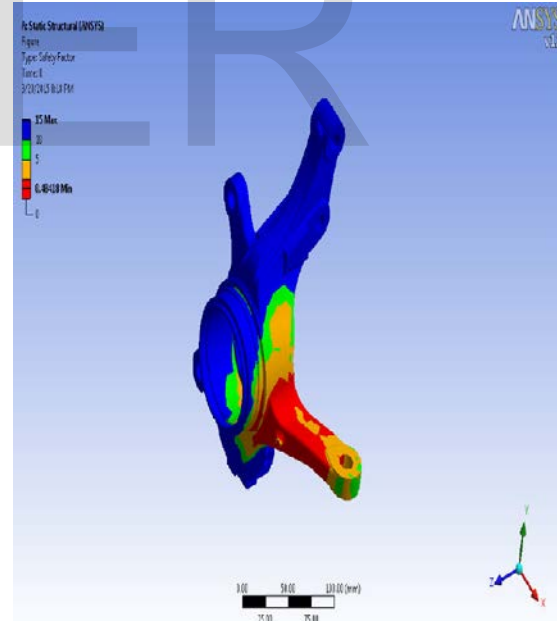


Fig.10.Safety factor

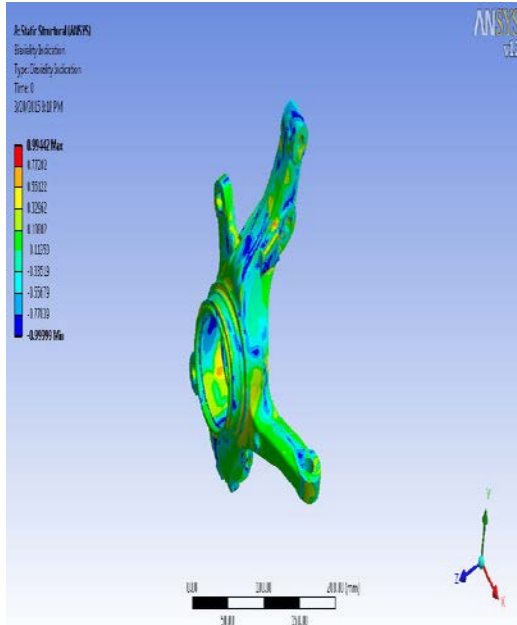


Fig.11. Biaxiality indication

6. 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 598Mpa 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.

Reference

1. Sharadkumarchandrakar,dheerajlalsoni and shahelgardia,(2013),”FEA of a steering knuckle for life prediction”,pp 681-688
2. PatelAkash A,(2013), “Fatigue or durability analysis of steering knuckle”, Small Engine Technology Conference Proceedings, Society of Automotive Engineers of Japan, Tokyo, pp. 379-389
3. R.L. Jhala,K.D. Kothari,(2009)”component fatigue behavior and life prediction of a steering knuckle using FEA”,pp.646–657.
- 4.Purushottamdumber.Kweon S.S,(2014),“Stress Analysis On Steering Knuckle Of The Automobile Steering System”. Materials and Design; volume 28(4):1198–1210.
- 5.M.Kamal, Md. MustafizurRahman, and M.S.M. Sani, (2013),”Application multibody Simulation For Fatigue Life Estimation” Volume 7, pp. 912-923
6. www.sciencedirect.com.
7. www.ijret.com