ARTICLE IN PRESS

Materials Today: Proceedings xxx (xxxx) xxx

Contents lists available at ScienceDirect

Materials Today: Proceedings

journal homepage: www.elsevier.com/locate/matpr

Optimization of fatigue life and fractography analysis of knuckle joint

R. Pugazhenthi [a,*], G. Anbuchezhiyan [c], R.K. Muthuraman [b], M. Vignesh [b], A. Ponshanmugakumar [d]

[a] Department of Mechanical Engineering, Vels Institute of Science, Technology & Advanced Studies, Chennai 600117, India
[b] Department of Aerospace Engineerin,Periyar Maniammal University, Vallam, Thanjavur 613403, India
[c] Department of Mechanical Engineering,Saveetha School of Engineering, Saveetha Institute of Medical and Technical Sciences, Chennai 602105, India
[d] Department of Mechanical Engineering, Sri Sairam Institute of Technology, Chennai-600044, Tamil Nadu, India

ARTICLE INFO

Article history:
Received 22 March 2021
Accepted 1 April 2021
Available online xxxx

Keywords:
Fractography
Fatigue
Knuckle
Optimization
Simulation

ABSTRACT

Steering knuckle is an important component of a vehicle suspension system, which acts as a connection point between the wheel hub and the suspension system. Since, the forces coming from the tires are transferred to knuckle with high varying loads. The design of a knuckle becomes very critical in terms of its life with high factor of safety. At the same time, the design also should be robust with the required fatigue life, right material selection and weight optimization, without affecting the desired performance. There has been a considerable effort placed in this direction to reduce the overall weight of the knuckle system by employing high strength alloys instead of conventional materials or optimizing the geometry without altering the desired functionality and its fatigue life. This project focuses on optimization and fatigue life estimation of an optimized Knuckle using Finite Element Analysis approach and fatigue tester. This is done by studying and comparing the results of a rigid baseline model with that of an optimized model through simulation and experimentation. The optimized model is arrived based on the stresses obtained from the baseline study. The fatigue life of the optimized model is evaluated under fatigue tester and fractography analysis is carried out.

1. Introduction

A steering knuckle is one of the most important part of a vehicle suspension system. This is the major component acts as a connection point between the wheel hub and the suspension system. Hence, the forces coming from the tires are transferred to this knuckle system hence subjecting it to high varying loads. Therefore, the design of a knuckle becomes very critical in terms of its life with high factor of safety. At the same time, the design also should be robust with the required fatigue life, right material selection and optimized weight, without affecting the desired performance.

Zoroufi et al. [1] had fatigue life comparisons of competing manufacturing processes: A study of steering knuckle. The method of manufacturing is the preliminary way in increasing life and reducing cost [2]. Jiang et al. [3] stated that the fatigue life analysis of a steering knuckle based on FEM by ANSYS and ANSYS is the better FEA tool for simulation analysis. Jhala and Kothari [4] worked on component fatigue behaviours and life predictions of a steering knuckle using finite element analysis and found Fatigue life prediction of knuckle is important. Kamal et. al [5], Fatigue life evaluation of suspension knuckle using multi body simulation technique and showed Simulation results are good enough to real time model. It saves the time and capital on real time testing [6,7]. Babu et al. [8], Stress analysis on steering knuckle of the automobile steering system and inferred that knuckle can be analyzed under the actual load conditions, which improve the depth knowledge of its function and performance in terms of durability and quality.

Gnanasekar et al. [9] stated that Finite Element Simulation of Knuckle & Strut Arm Column Assembly for Automotive Steering and concluded that optimization of shape and material can lead to better performance and life [10]. Zhang et al. [11] did research on fatigue life of front axle beam for heavy-duty truck and inferred that analyze the effect of crack parameters such as length and depth on fatigue life [12]. Christianah et. al [13], had design and simulation of fatigue analysis for a Vehicle Suspension System

* Corresponding author.
E-mail address: pugal4@gmail.com (R. Pugazhenthi).

https://doi.org/10.1016/j.matpr.2021.04.018
2214-7853/

(VSS) and its effect on global warming and concluded that the reducing vehicle weight results in less fuel consumption and a decrease in CO2 emission which in turn has an effect on global warming.

The major target of this work is to do weight optimization of a steering knuckle which is known to be study in design and has lot of scope for weight reduction. The current design has a good factor of safety hence has to be reduced in weight to achieve weight reduction in batch production. The 3D knuckle joint model is generated and simulated to determine the optimal fatigue life.

2. Methodology

The methodology of the research work is shown in Fig. 1.

2.1. Solid modelling

SOLID185 Element Description SOLID185 is used for 3-D modeling of solid structures. It is defined by eight nodes having three degrees of freedom at each node: translations in the nodal ×, y, and z directions. The element has plasticity, hyper elasticity, stress stiffening, creep, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperplastic materials. Various element technologies such as B-bar, uniformly reduced integration, and enhanced strains are supported. The meshed image is given in Fig. 2.

The average weight of the vehicle is around 1150 kgs. The design/load calculations are based on this majorly. A worst case load acting on the Knuckle combining all the three load cases was taken for analysis considering conservative approach. These loads when act along with the self-weight makes it more conservative to be considered for the static analysis, ensuring safe design.

Bump load case (3 g) = 3*9.81*Kerb weight
= 33844.5 N (four wheels)
Braking load case (2 g) = 2*9.81*Kerb weight
= 22563 N (four wheels)
Braking load case (1 g) = 1*9.81*Kerb weight
= 11281.5 N (four wheels)

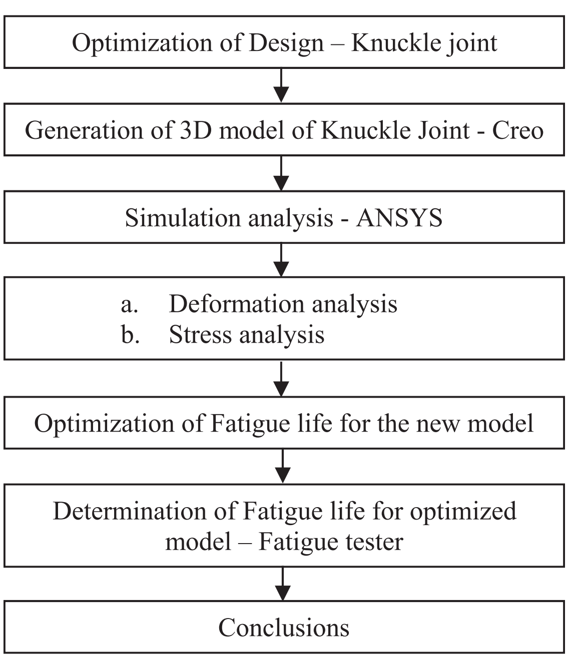

Fig. 1. Research Flow.

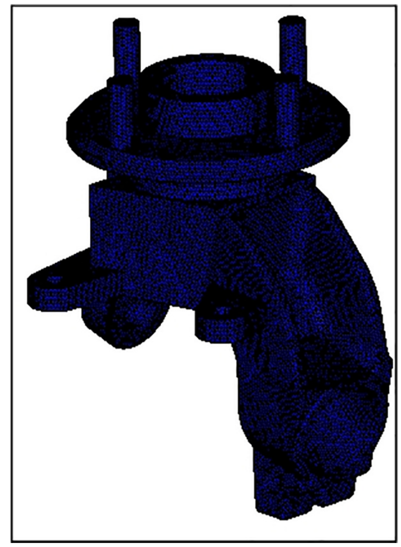

Fig. 2. 3D Model of Knuckle Joint.

2.2. Fatigue test

The deflection analysis of the knuckle joint is done by using the fatigue rig. The testing is done by applying the load at the middle of the knuckle joint and allowed to slide freely over the table of UTM. The cyclic loading frequency is fixed as 10 Hz. By increasing the exposure time, the number of cycles of loading is increased. As initial trial, each category of specimens are given σavg is 80 MPa of loading and is eventually tested for its load bearing capacity against deflection.

3. Results and discussion

3.1. Baseline model analysis

The Baseline results showed a max average stress of 70 MPa on the Steering Knuckle, whereas a concentrated band of stress of 151 MPa due to stress concentration is also observed. This maximum stress due to stress concentration due to stress singularity is ignored, since such sharp corners do not exist in real life scenario. Detailed Deformation and Stress Plots are shown as in Figs. 3 and 4.

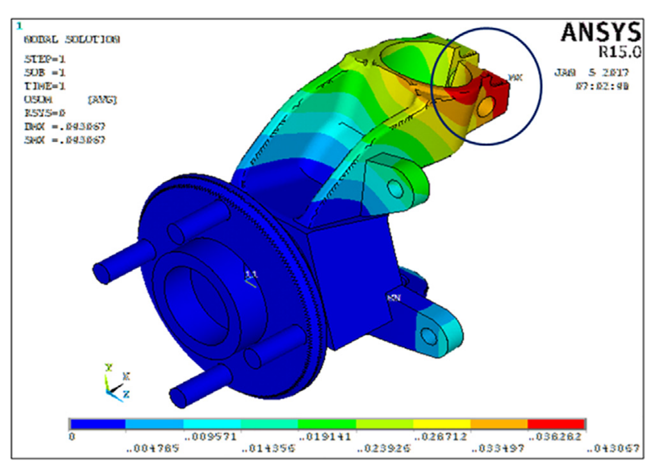

Fig. 3. Deformation Plot – Isometric View.

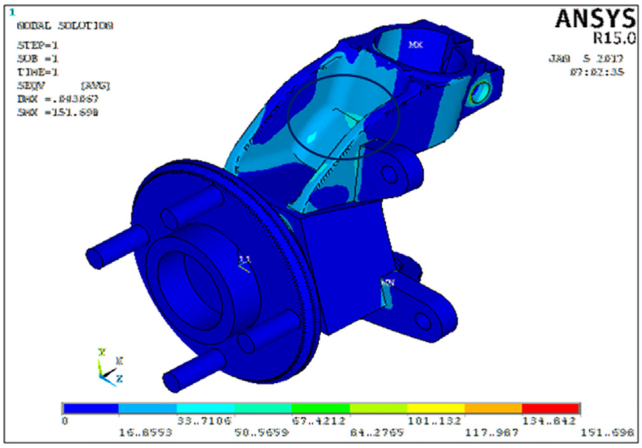

Fig. 4. Stress Plot – Isometric View.

3.2. Optimized model analysis

The Baseline model which was analyzed gave results well below the fatigue limit and stress margin. This showed that the model has extensive room for weight optimization.

The weight of the baseline Steering Knuckle is around 4.8 kg. After reducing the thickness at all zones where the stresses were very well below the margin, the weight has dropped to 4.0 kg, which is closed to around 0.8 kg or 18%. Table 1 and Fig. 5 show the zones where the thickness was decreased and the weight has been reduced Table 2.

The optimized model results showed a max average stress of 80 MPa on the Steering Knuckle, whereas a concentrated band of stress of 189 MPa due to stress concentration is also observed. This maximum stress due to stress concentration due to stress singularity is ignored, since such sharp corners do not exist in real life scenario. Detailed Deformation and Stress Plots are shown as in Figs. 6 and 7, respectively.

Table 1
Variation of Dimension on Knuckle.

Zone	Old Thickness	New Thickness
Zone 1 & 2	11.2 mm	9 mm
Zone 3	10 mm	8 mm
Zone 4	20 mm	16 mm

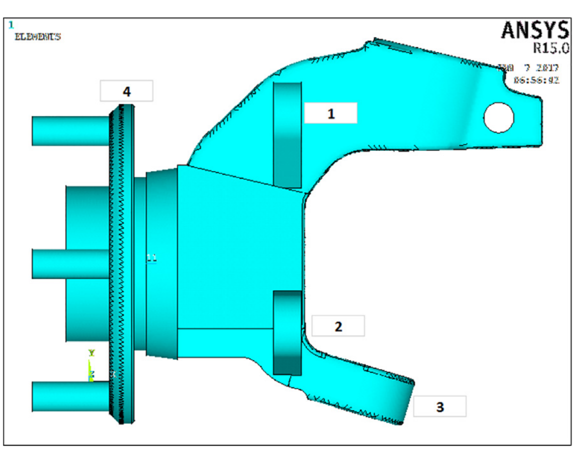

Fig. 5. Optimized Model.

Table 2
Comparison of results.

Factor	Existing Model	Optimized Model
Max. Stress	151 MPa	189 MPa
Avg. Stress	65 MPa	80 MPa
Fatigue life	1.2E6 cycles	1.387992E6 cycles

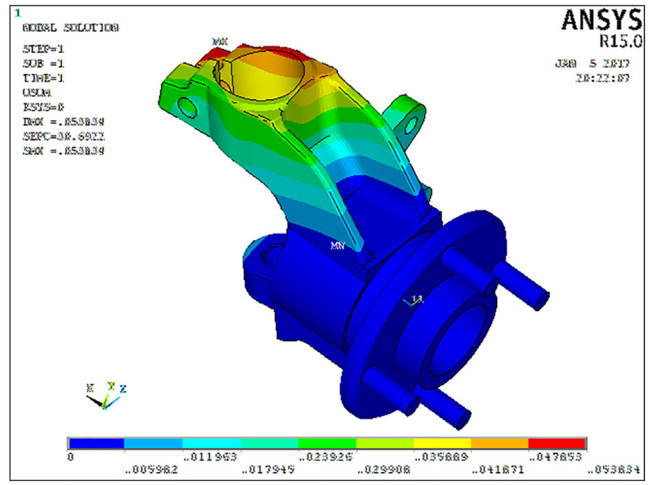

Fig. 6. Deformation Plot for optimized model.

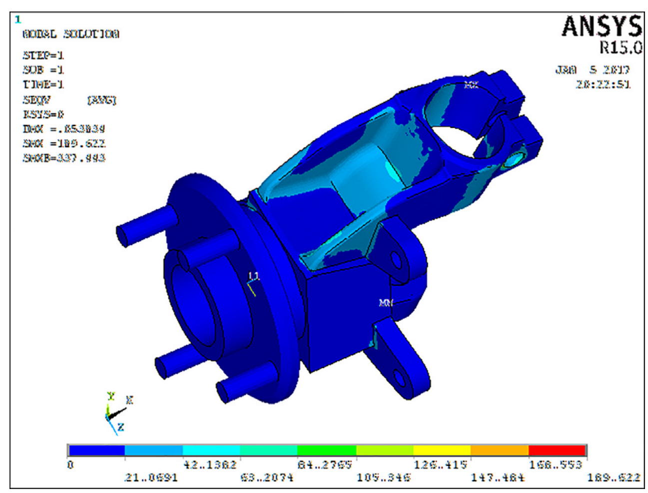

Fig. 7. Stress Plot for optimized model.

3.3. Fatigue analysis

Fatigue of a component is defined as the number of cycles of loading the component can withstand before failing. Fatigue failure occurs when the component is undergoing cyclic loading pattern and can fail even when the operating stress is much lesser the material yield. Hence it is very important to design a component to satisfy its designed life with the operating stress much lesser than the yield strength.

In general, infinite life is referred to any life cycle beyond 1,000,000 cycles. Any mechanical component meeting infinite life requirements can be considered safe. The Endurance limit or Fatigue limit of Cast Iron is calculated using the empirical formula 0.4*Ultimate Strength (Fig. 8).

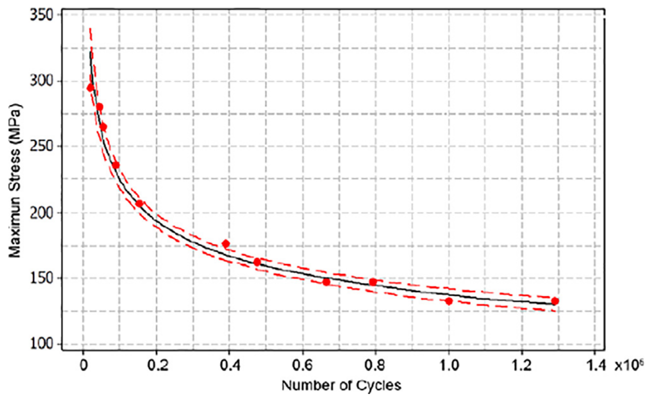

Fig. 8. S-N curve of Cast Iron.

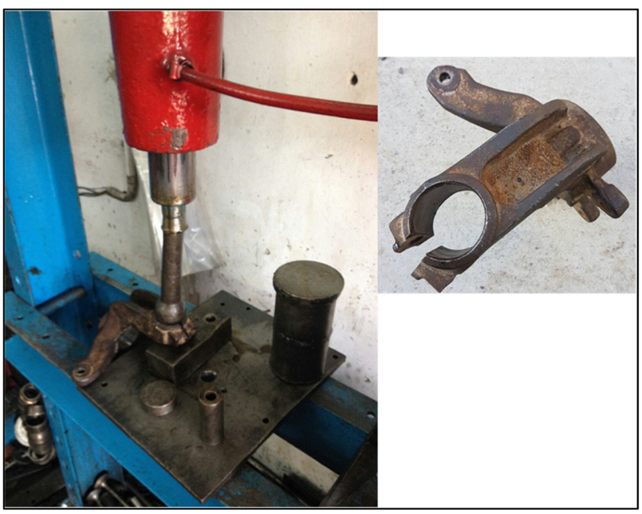

Fig. 9. Photograph of Knuckle Joint Failure.

Optimized model is tested in Fatigue Tester as shown in Fig. 9. For, σ_{avg} = 80 MPa and f = 10 Hz under Tensile – Compression Cycle, the obtained Fatigue life = 1.387992E^6 cycles.

3.4. Fractography analysis

The SEM images are taken at zone to study the failure i.e. crack initiation. Fig. 10 shows the various magnification of the crack developed for the knuckle under fatigue test.

4. Conclusions

The following are the conclusions obtained from the present study,

1. The 3D model is generated for the existing knuckle joint model successfully.
2. The deformation and stress analysis are carried out in ANSYS and the results are recorded successfully.
3. A new optimized model is generated based on design and its deformation and stress analysis are determined.
4. The fatigue strength is increased for the new optimized model with reduced dimension i.e. weight reduction.
5. The avg. stress is increased from 65 MPa to 80 MPa for infinite cycles.
6. The fractography analysis shows the cause and effect of the stress on knuckle joint at zone 1.

The Steering Knuckle is successfully optimized in weight/thickness reduction around 18% compare to existing model, which is calculated using basic principles of engineering.

CRediT authorship contribution statement

R. Pugazhenthi: Methodology, Writing - original draft, Conceptualization. **R. K. Muthuraman:** Data curation. **M. Vignesh:** Formal analysis. **A. Ponshanmugakumar:** Investigation. **G. Anbuchezhiyan:** Writing - review & editing, Validation.

Declaration of Competing Interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

References

[1] Anbuchezhiyan,G,Mohan,B,Sathyanarayanan,D.Muthuramlingam. J. Alloys Compd.719. (2017),125-132.
[2] M. Vignesh, R. Ramanujam, Mater. Res. Express. 6 (8) (2019) 086563.
[3] Anbuchezhiyan,G, Mohan,B&Muthuramlingam D. Arch. Civ. Mech.18,(2018), 1645-1650.
[4] A. Rajasekaran, R. Pugazhenthi, Mater. Today:. Proc. 22 (2020) 766–771.
[5] Pugazhenthi, R., D. Sreeram, SS Vijay Ananth, and A. Devaraju, Materials Today: Proceedings, 22 (2020) 701-704.
[6] M. Vignesh, R. Ramanujam, J. Braz. Soc. Mech. Sci. Eng. 41 (9) (2019) 1–10.
[7] Vignesh, M, Ramanujam, R. P I Mech Eng E-J Pro. https://doi.org/10.1177/0954408920964687.
[8] G.Anbuchezhiyan,B.Mohan,N.SenthilKumar,R.Pugazhenthi(2021), Met. Mater. Int, https://doi.org/10.1007/s12540-020-00906-3
[9] R. Pugazhenthi, S. Sivaganesan, C. Dhansekaran, A. Parthiban, Int. J. of Vehicle Struct. Systems 11 (2019) 2.

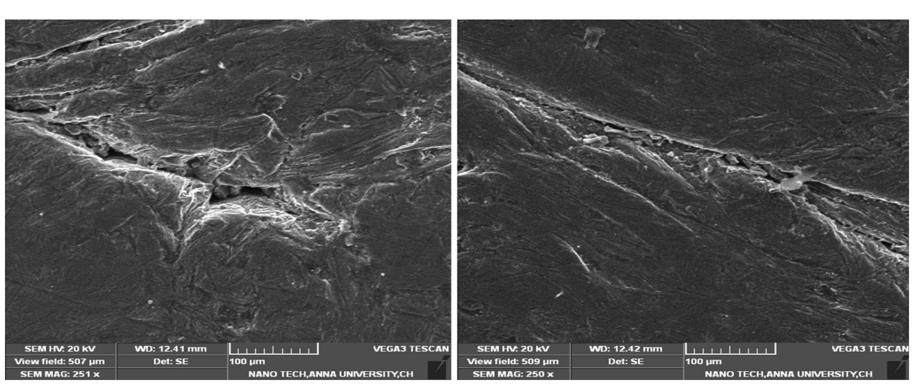

Fig. 10. SEM image showing the crack initiation.

[10] M. Vignesh, K. Venkatesan, R. Ramanujam, S. Vijayan, Appl. Mech Mater. 787 (2015) 643–647.
[11] Reddy, A. Prasad, P. Vamsi Krishna, and R. N. Rao., Transactions of the Indian Institute of Metals, 72.9 (2019): 2533-2546.
[12] M. Vignesh, K. Venkatesan, R. Ramanujam, P. Kuppan, Appl. Mech Mater. 787 (2015) 460–464.
[13] Daniel, S, Ajith Arul, R, Pugazhenthi, R, Kumar, and S, Vijayananth., Defence Technology, 5 (4), (2019): 545-556.

Further Reading

[1] Saravanan, R, M, Santhanam, S, Nadarajan, and R, Pugazhenthi. Journal of Advanced Microscopy Research, 10 (4), (2015): 296-302.
[2] K, Velavan, B.Mohan, G.Anbuchezhiyan,,N.Senthilkumar.Silicon (2021),https://doi.org/10.1007/s12633-020-00928-x
[3] Daniel, S, Ajith Arul, R, Kumar, S, VijayAnanth, and R, Pugazhenthi., Materials Today: Proceedings, 22 (2020): 853-857.
[4] Anbuchezhiyan,G, Mohan, B &Karthikeyan, RV(2015) App.Mechanics and Materials., vol. 766-767, pp. 281- 286
[5] N. Poornachandiran, R. Pugazhenthi, S. Vijay Ananth, T.G. Krishnan, M. Vairavel, AIP Conf. Proc. 2283 (1) (2020) 020113.
[6] P.T. Selvam R. Pugazhenthi C. Dhanasekaran M. Chandrasekaran S. Sivaganesan Materials Today: Proceedings 2020.
[7] G. Anbuchezhiyan, B. Mohan, P.R Kathiresans, Mater. Today Proc. 27 (2020) 271530–271534.