ISSN 2395-1621

Design Analysis, Weight Reduction & Fatigue Life Prediction of Steering Knuckle Arm Using FEA

#1Mr.Kiran S. Bhokare, #2Dr. G. M. Kakandikar

[1]kiranbhokare1980@gmail.com
[2]ganesh.kakandikar@zealeducation.com

[1]Department of Mechanical Engineering, DCOER, Narhe, Pune 41 (MS) India
[2]Associate Professor Mechanical Engineering, DCOER, Narhe, Pune 41 (MS) India

ABSTRACT

Automobile components and structures are regularly subjected to cyclic loading and they are consequently prone to fatigue damage. Steering knuckle, being a part of the vehicle's steering and suspension system, undergoes time-varying loading during its service life. This work shall be consisting of analysing the steering knuckle for the main operating conditions.

The Finite Element methodology is adopted to solve the problem while determining the fatigue life of the steering knuckle. The process of validation for this method is sought through the Physical Experimentation for determining stress levels and/or deflection at the designated locations identified using F.E.Modeling. Since the input used for Fatigue is the stress in the component, validation is sought through comparison of this input data. Fatigue life experimentation is usually not feasible since this is a case of high cycle fatigue. In the physical experimentation we will be finding out the graph for stiffness i.e. Load v/s Deflection for the component. The results of Experimental work shall be compared for results with the F.E. Modeling. This paper focuses on optimization of steering knuckle targeting reducing weight as objective function, while not compromising with required strength and stiffness. Taking into consideration static and dynamic load conditions, structural analysis and fatigue analysis were performed. The concurrence of the results shall offer validation for this thesis work.

Keywords— Fatigue life, FEMFAT, MSC NASTRAN, Steering Knuckle, Weight Reduction.

ARTICLE INFO

Article History

Received :18th November 2015

Received in revised form :

19th November 2015

Accepted : 21st November , 2015

Published online :

22nd November 2015

I. INTRODUCTION

A Steering Knuckle is one of the critical components of vehicle which connects brake, suspension, wheel hub and steering system to the chassis. It undergoes varying loads subjected to different circumstances, while not distressing vehicle steering performance and other desired vehicle characteristics. The knuckle is the major pivot in the steering mechanism of a car or other vehicle, free to revolve on a single axis. The knuckle is vital component that delivers all the forces generated at the Tier to the chassis by means of the suspension system. The design of the knuckle is usually done considering the various forces acting on it which involves all the forces generated by the road reaction

on the wheel when the vehicle is in motion. The design also includes various constraints that are related to the knuckle such as brake system, steering system, drive train and suspension system.

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. Steering Knuckle is subjected to time varying loads during its service life, leading to fatigue failure. Therefore, its design is an important aspect in the product development cycle.

A knuckle component is required to support the load and torque induced by bumping, braking, and acceleration and also helps in steering the tire connecting tie rod and rotating at the kingpin's axis centre. In the design optimization of the knuckle component, a weight should be minimized, while design factors such as strength, stiffness and durability should be satisfied with design targets. The steering knuckle accounts for maximum amount of weight of all suspension components, which requires high necessity of weight reduction. Under operating condition is subjected to dynamic forces transmitted from strut and wheel. The weight reduction of steering knuckle is done such that the strength, stiffness and life cycle performance of the steering knuckle aresatisfied.

Steering knuckle was used as component for study as shown in below fig.1. Suspension system in any vehicle uses different types of links, arms, and joints to let the wheels move freely; front suspensions also have to allow the front wheels to turn. Steering knuckle/spindle assembly, which might be two separate parts attached together or one complete part, is one of these links.

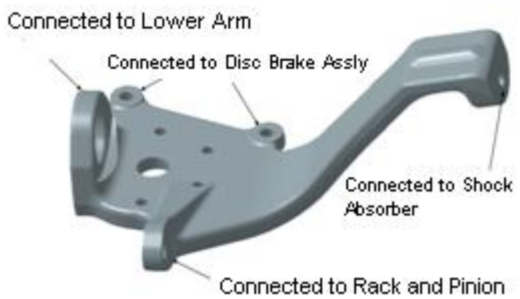

Fig.1 Machined component with all connections

II. LITERATURE SURVEY

Fatigue behavior is, therefore,a key consideration in its design and performance evaluation. Thispaper is aimed to assess fatigue life and compare fatigue performanceof steering knuckles made from three materials of differentmanufacturing processes. These include forged steel, cast aluminum,and cast iron knuckles. In light of the high volume of forged steelvehicle components, the forging process was considered as base forinvestigation. Static as well as baseline cyclic deformation and fatigueproperties were obtained and compared[2]

By reducing mass of the vehicle components, overall mass reduction of a vehicle and lowering of energy consumption demand can be achieved, therefore, improving fuel efficiency. Material resources will also be conserved. The objective of this research is to reduce mass of an existing steering knuckle component of a local car model by applying shape optimization technique. The improved design helps attain 8.4% mass reduction. Although volume reduction and shape changes exist, there is no significant change in maximum stress. This result is satisfactory with optimization in shape only, limited design space and no design change in material properties[3].

The steering knuckle is a part of the vehicle's steering and suspension system which undergoes time-varying loading during its service life. Components such as steering knuckle are subjected to fatigue failures due to cyclic loads arising from various driving conditions.method used in the fatigue life reliability evaluation of the knuckle used in a passenger car steering system. An accurate representation of Belgian pave service loads in terms of response-time history signal was obtained from accredited test track using road load data acquisition. The acquired service load data was replicated on durability test rig and the SN method was used to estimate the fatigue life[5].

The process of designing a light weight knuckle from scratch. The design space is identified for the knuckle and subsequently a design volume satisfying the packaging requirements is created from it. Using OptiStruct, topology optimization is performed on the design volume to derive the optimal load path required for the major load cases. Hypermorph is used to create the required shape variable and Hyper Study is used as optimizer. The process of using Topology optimization for load path generation& Parametric study using shape optimization, reduces the design iteration and intermediate concept models and there by reduces the design cycle time.

There are four disciplines for optimization process:

Topology optimization: It is an optimization process which gives the optimum material layout according to the design space and loading case.

Shape optimization: This optimization gives the optimum fillets and the optimum outer dimensions.

Size optimization: The aim of applying this optimization process is to obtain the optimum thickness of the component.

Topography: It is an advanced form of shape optimization, in which a design region is defined and a pattern of shapevariable will generate the reinforcements.[10]

III. SCOPE AND OBJECTIVE

A. Scope

Existing Design of the Steering Knuckle shall be evaluated using Computational Methodology using suitable CAE tools. The geometry of the Arms of the Steering Knuckle shall be the input for this work which shall undergo Finite Element Analysis to determine its Fatigue life. The F.E Modeling for design evaluation shall be pursued for 'Structural' analysis. The aspect of 'fatigue' would be investigated to improve design & maintain performance over 'durability'. Conclusion phase is to include physical experiment for validating 'stiffness' of the spring. Recommendation isbeing offered upon comparison using Test Report.

B. Objectives

1) Identify through literature survey the possible causes of failure of the component.
2) Modeling and FEA analysis of the component :
   a) Secure input data in the form of CAD model (geometry).
   b) Engage pre-processor with loading & boundary conditions.
   c) Apply solver and view results over post processor for structural assessment.
3) Conducting experimentation on the component: Record the load v/s defection OR load v/s stress through experimentation with the Static Analysis results.
4) Validate the hypothesis: Validation of results obtained by Finite Element Method with Experimental results.
5) Modifying the design with removal of material from low stress areas and ensuring the Fatigue life under safe zone.

IV. PROBLEM DEFINITION

Steering knuckle is one of automotive part that frequently carries load from several directions. It is connected to the steering parts and strut assembly from one side and the wheel hub assembly from the other; it has complex restraint and constraint conditions and tolerates a combination of loads. Each circumstances of the road give the different impact to the steering knuckle. In addition, parameters such as internal defects, stress concentrations and gradients, surface finish, and residual stresses can have considerable influence while designing for fatigue. For this work, the need is to identify a typical load case for analysis while offering variation to the critical parameter/s of Design to determine the influence of the change in parameter over the fatigue results. Further process carries modifying the design with removal of material from low stress areas while maintaining fatigue life of the component under safe zone.

V. METHODOLOGY ADOPTED

A. Methodology

The normal mode analysis and stiffness of the steering knuckle is obtained using finite element analysis. Finite element modeling is done using HYPERMESH and the necessary boundary conditions and material are imposed on it. The setup is solved using MSC NASTRAN solver. Weight reduction is done using removing material from low stress areas in the steering knuckle. The weight reduction is done using Topology optimization by meeting the strength and stiffness. And the corresponding weight reduction is analysed. The design modification is done keeping manufacturing feasibility intact with few iterations, the optimized design satisfying all the strength, stiffness & life-cycle performance criteria is evolved.

B. Typical Methods Used for study

1) Analytical Method: For analysis purpose, Meshing and boundary conditions are applied using Hyper mesh as a pre-processor. For Static solving purpose typically MSC-NASTRAN is used. Results would be predicted using Hyper view Software. For Fatigue life analysis Static analysis input is given to FEMFAT.
2) Mathematical Calculations: In this phase steering knuckle being a complex geometry with multiple connections we calculate the stress i.e. loads v/s deflection simplifying the geometry. This is a approximation process.
3) Physical Experimentation: This methodology shall be deployed as an alternative methodology for validating the hypothesis. The Static analysis results determined by Analytical methodology are validated with the results achieved from Physical experimentation.

C. Steps followed in Analytical Methodology

1) Preparing the CAD Model: Below fig.2 shows the CAD model of steering knuckle. CAD model of steering knuckle was developed in 3D modeling software CATIA. It consists of stub hole, brake caliper mounting points, steering tie-rod mounting points, suspension upper and lower A-arm mounting points. Knuckle design mainly depends on suspension geometry and steering geometry.

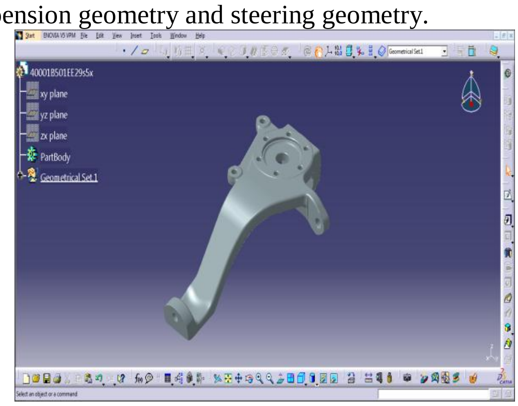

Fig.2 CAD Model created using CATIA V5.

2) Meshing the CAD Model: CAD model of knuckle converted into STEP file. This model is imported into Hyper mesh. Geometry cleanup was performed prior to meshing of model. Further meshing is carried out on the model.For better quality of mesh fine element size is selected as shown in fig.3 and fig.4

No Of Elements = 269320
No of Nodes = 62961
Type of element = tetrahedral
Type of Meshing = Solid mesh

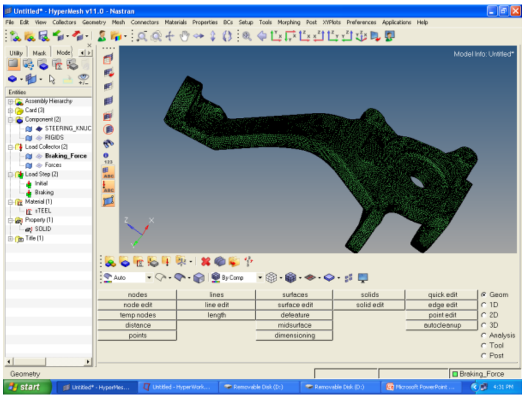

Fig.3 Meshed Benchmarked model using HYPERMESH

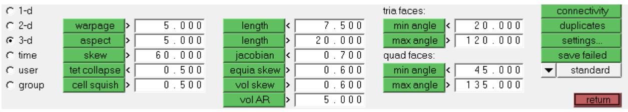

Fig.4 Meshed Elements details

3) Applying the Boundary Conditions: The base geometry is simulated for the actual loading conditions. The load conditions are obtained from data acquisition system.

There are two types of load acting on knuckle i.e. force and moment.This knuckle is designed for vehicle of 2300 kg so braking force acting on it produces moment:

Moment = force* perpendicular distance.

The fig.5 shows the boundary conditions applied

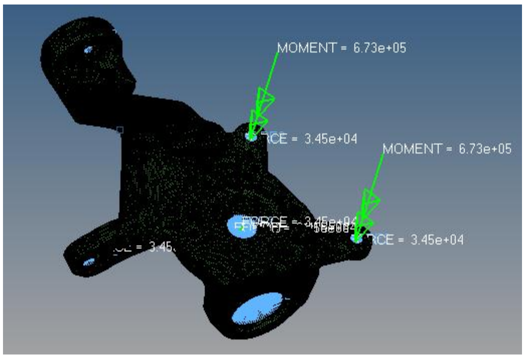

Fig.5 Boundary Conditions and Constraints

The loading conditions applied are given in table 1 below:

Loading Condition		**Loads**
Braking Force	1.5*g	17250
Lateral Force	1.5*g	17250
Steering Force	Steering Effort of 50N	50N
Load on knuckle hub along X	3*g	17250
Load on knuckle hub along Y	3*g	17250
Load on knuckle hub along Z	1*g	5750

Table.1 Loading conditions of knuckle

= 1.5*g *78Nmm
= 1.5*(2300/4)*10*78
= 672750Nmm

4) Applying the Material Properties: Material details and properties such as Density, Poisson's ratio and Young's Modulus are assigned to further to the analysis.

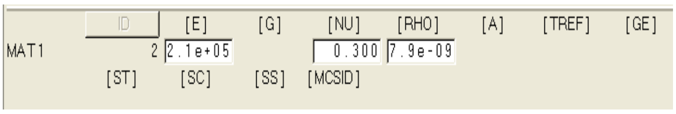

Fig.6 Material properties

5) Solving the Model for Static Analysis: MSC NASTRAN is used to carry out the Static analysis.Fig.7 and 8 shows Displacement and Stress plots received from MSC NASTRAN

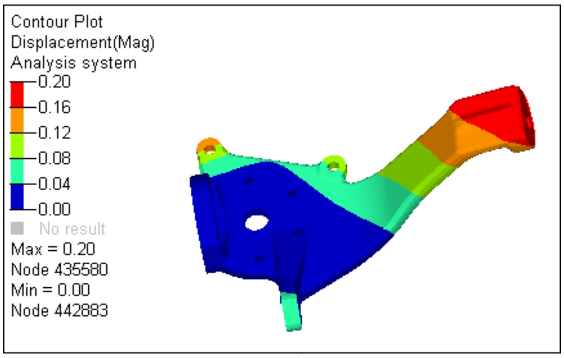

Fig.7 Displacement Plot for Benchmarked Model

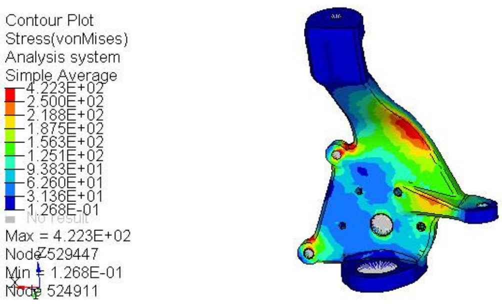

Fig.8 Stress Plot for Benchmarked Model

6) Fatigue Analysis for Life prediction: Output from Static Analysis is given as input to FEMFAT for carrying out the Fatigue analysis. Output from the FEMFAT is the Stiffness Plot and the Fatigue life Plot at different locations in the Steering Knuckle as shown in fig.9

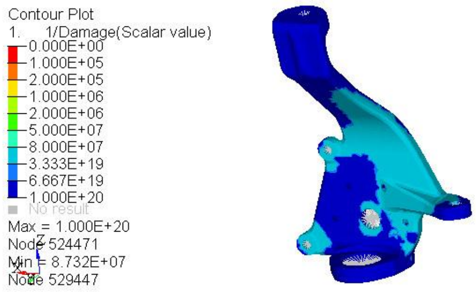

Fig.9 Fatigue Life plot for Benchmarked Model

D. Mathematical Calculations

In this phase steering knuckle being a complex geometry with multiple connections we can calculate the stress i.e. loads v/s deflection simplifying the geometry by considering each arm as a cantilever beam. This is an approximation process shown below.

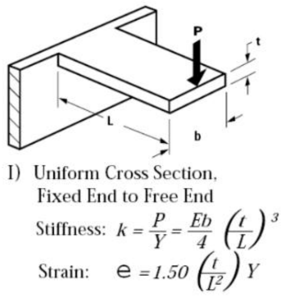

Fig.10Cantilever beam example for strain calculation

E. Physical Experimental Setup

Special Fixture is designed to hold the component for doing the Static Analysis. Ultimate Tensile Strength (UTM) machine shown in fig.11 below is used to do the Static Analysis. Load using the Load cells and Plunger is applied on one of the Arm of the Steering Knuckle. Output received from the experimental setup is Load v/s Deflection Plot. Theresults from the experimentation are used for validation with the Analytical results.


Fig.11 Physical Experimentation of Benchmarked Model

VI. VALIDATION, RESULTS AND DISCUSSION

A. Validation of the Results

Validation process is done on bases of comparing Static Analysis and Experimentation results. For this work, "Stiffness" of the component is considered as a significantresponse for validation. Load v/s Deflection graph is plotted for the values of interest during experimentation. The same is compared with the analytical results for positive concurrence.

B. Results and Discussion

Results obtained from above projects are as below.

Results from Static Analysis and Experimentation

Results obtained from Static analysis using CAE tool is validated using with the experimentation carried on UTM machine for static analysis. In both the cases the load v/s deflection plot is compared. This validation is taken as an input for carrying out the fatigue life prediction process using analytical method in FEMFAT.

Table.2 Comparison of Analytical and Experimentation results

Load Applied (N)	Deflection determined by FEA (mm)	Deflection recorded during Experimentation (mm)	% Variation
28000N	7.024	7.26	3.13%

VII. CONCLUSION

Few stresses are found in the benchmarking model for Steering Knuckle Arm.Material reduction will be carried out at the locations where the stress values are comparatively less. This process of material removal will be carried out with respect to manufacturing feasibility of the component with respect to forging process without disturbing the existing parting line.

Fatigue life is predicted using FEMFAT for the benchmarked model. Fatigue life experimentation is not feasible due to time consuming and costlier approach.

VIII. FUTURE SCOPE

Future scope of this research is modifying design of the steering knuckle arm. This process of material removal willbe carried out with respect to manufacturing feasibility of the component with respect to forging process without disturbing the existing parting line. Fatigue life of the component will ensured to be in safe zone even after modifying the design further with respect to weight reduction. Target weight reduction achieved for this work is 5-8 % of the original weight of the component.

ACKNOWLEDGEMENT

Thanks to Dr G. M. Kakandikar for his valuable contribution in dissertation on Design Analysis, Weight Reduction, Fatigue life prediction of Steering Knuckle Arm.

REFRENCES

[1] MehrdadZoroufi and Ali Fatemi, "Fatigue Life Comparisons of Competing ManufacturingProcesses: A Study of Steering Knuckle" - SAE International 2003.

[2] Sharad Kumar Chandrakar, DheerajLalSoni and ShohelGardia, "FEA of A Steering Knuckle for Life Prediction" - International Journal of Engineering Research and Technology, ISSN 0974-3154 Volume 6, Number 5 (2013), pp. 681-688.

[3] Nassir S., Al-Arifi., Abu S., Optimization of Steering Knuckle Using Taguchi Methodology, International Journal of Comuter Theory and Engineering, Vol. 3, No. 4, August 2011, pp 552-556.
[4] Mahendra L. Shelar and Prof. H. P. Khairnar, "Design Analysis and Optimization of Steering Knuckle Using Numerical Methods and Design of Experiments" 2014 IJEDR | Volume 2, Issue 3 | ISSN: 2321-9939.
[5] E. A. Azrulhisham, Y. M. Asri, A.W. Dzuraidah, N.M. Nik Abdullah, A. Shahrum and C. H. CheHassan, "Evaluation of Fatigue Life Reliability of Steering Knuckle Using Pearson Parametric
Distribution Model" International Journal of Quality, Statistics, and Reliability Volume 2010, Article ID 816407, 8 pages.
[6] PurushottamDumbre, A. K. Mishra, V. S. Aher, "Structural Analysis of Steering Knuckle for Weight Reduction" ISSN 2250-2459, ISO 9001:2008 Certified Journal, Volume 4, Issue 6, June 2014.
[7] Wan Mansor Wan Muhamad, EndraSujatmika, Hisham Hamid, &FarisTarlochan, "Design Improvement of Steering Knuckle Component Using Shape Optimization" International Journal of Advanced Computer Science, Vol. 2, No. 2, pp. 65-69, Feb. 2012.
[8] Rajesh M.Metkar, VivekK.Sunnapwar, SubhashDeoHiwase, VidyaSagarAnki, MahendraDumpa, "Evaluation of FEM based fracture mechanics technique to estimatelife of an automotive forged steel crankshaft of a single cylinder diesel engine" Procedia Engineering 51 (2013) 567 – 572.
[9] B.Babu, M. Prabhu, P.Dharmaraj, R.Sampath , "Stress Analysis on Steering Knuckle of the Automobile Steering System. eISSN 2319-1163, pISSN 2321-7308
[10] VirajKulkarni, AmeyTambe, Optimization and Finite Element Analysis of Steering Knuckle – Altair Technology Conference-2013 India.
[11] Razak I.H.A, Yusop M.Y.M, Yusop M.S.M, Hashim M.F, Modeling, Simulation and Optimization Analysis Of Steering Knuckle Component For Race Car eISSN 2319-1163, pISSN 2321-7308