Gnanasekar et al. 2015. *Int. J. Vehicle Structures & Systems*, 7(4), 128-131
ISSN: 0975-3060 (Print), 0975-3540 (Online)
doi: 10.4273/ijvss.7.4.01


**International Journal of Vehicle Structures & Systems**
Available online at www.maftree.org/eja

# Finite Element Simulation of Knuckle and Strut Arm Column Assembly for Automotive Steering System

**N. Gnanasekar[a], Rama Thirumurugan[b] and S. Madhusuthanan[b]**

[a]*Dept. of Mech. Engg., P.A. College of Engineering and Technology, Pollachi, Tamilnadu, India*
*Corresponding Author, Email: sekarfea@gmail.com*
[b]*C-DAT, Dept. of Mech. Engg., Dr. Mahalingam College of Engineering and Technology, Pollachi, Tamilnadu, India*

**ABSTRACT:**

*Steering knuckle is one of the important parts in the vehicle steering system. Under certain operating conditions, the knuckles reliability threatens the safety of people and vehicles directly. At emergency braking condition, mostly strut arm and steering arm have maximum deflection in steering knuckle when it is subjected to various load cases. In this work, finite element analysis of the spheroid graphite (SG) iron strut arm of steering knuckle with strut mount assembly was carried out to predict its deflection under static load. The analysis result was compared with that of the experimental results to put forward directions to optimize the shape and material selection.*

**KEYWORDS:**

*Steering knuckle; Finite element analysis; Spheroid Graphite iron; Static analysis*

**CITATION:**

N. Gnanasekar, R. Thirumurugan and S. Madhusuthanan. 2015. Finite Element Simulation of Knuckle and Strut Arm Column Assembly for Automotive Steering System, *Int. J. Vehicle Structures & Systems*, 7(4), 128-131. doi:10.4273/ijvss.7.4.01

## 1. Introduction

In motor vehicles, the mechanical parts are subjected to different load cases during their service life. Steering knuckles are very crucial component of the vehicle's steering and suspension system. This has to deflect as much as possible before it fails due to unexpected loads. Since knuckle is made up of cast iron, the deflection study for the given load is very important. Geometric optimization of the knuckle has been investigated through finite element (FE) analysis to reduce the weight. Failure of steering knuckle component leads to the loss of orientation of the vehicle [1]. The steering knuckle component acts as a junction between the steering, wheel hub assembly and suspension system. Steering knuckle system consists of a strut arm at the top, lower control arm at the bottom and a steering arm on the side as shown in Fig. 1.

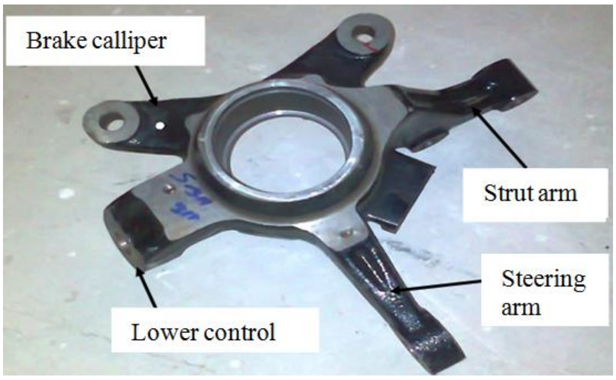


**Fig. 1: Steering knuckle component**

The wheel rod is connected to the steering parts and the strut arm assembly with suspension system from one side and the wheel hub assembly from the other side. It has complex connections and tolerates a combination of static and dynamic loads during its service period. These forces are transferred to chassis through the strut arm, steering and lower arm assemblies. Therefore, displacement and stress analysis of these arms are essential for knuckle to satisfy it's functional as well as safety requirements [2]. The steering knuckles are usually manufactured using forged steel, cast aluminium, and cast iron. The durability of these materials and manufacturing processes were studied and reported that the durability of forged steel was superior to the other two materials. Nowadays, spheroid graphite (SG) iron is mostly used in the commercial automobile sector to manufacture the steering knuckle [3].

The experimental methods are costly and time consuming process to predict the performance of steering knuckle component when compared to other methods. Hence, researchers are looking for new methods like FEM, statistical tools and optimization codes to study the performance. A statistical method was used to predict the failure probability of steering knuckle parts to replace the testing time [4]. A probabilistic design method was approached to bring the improvement in the product fatigue life prediction and structural reliability [5]. The optimum casting design of knuckle was proposed through detailed stress analysis by using urethane models [6]. The steering-arm portion undergoes repeated bending loads during the vehicle's movement and it leads to failure because of bending fatigue condition. Through simulations and experiments, it was found that aluminium metal matrix composite knuckles could safely replace SG iron and optimized the strut arm region using genetic algorithm for minimum bending stress and deflection [7]. It was reported that the fatigue and impact behaviour were improved by titanium carbide particulates in aluminium matrix when compared with conventional materials [8].

Linear analysis can provide most of the information about the behaviour of a structure and can be a good approximation for many analyses. It is also the basis of nonlinear analysis in most of the cases. In this work the strut arm region was analyzed for maximum deflection with strut mount assembly under static loading using experiment and FE methods. The results of FE analysis were compared with experimental results to put forward directions for improvements in design.

## 2. Numerical analysis methodologies

### *2.1. FE Modelling*

The geometric model of the steering knuckle designed in CATIA was imported into the Hypermesh software to clean up the geometry and generate the mesh with size control, curvature and proximity options in order to generate a reasonable quality mesh. In this study, three dimensional 10-Node Tetrahedral SOLID 187 Structural elements are used for meshing steering knuckle because of its quadratic displacement behaviour. The steering knuckle is discretized into 65,257 elements with 1,08,083 nodes. Three dimensional 8-node structural SOLID 185 element is used for meshing strut mount because of its linear displacement behaviour. The strut mount is discretized into 17,915 elements with 28,101 nodes. Fig. 2 shows the discretized FE model of steering knuckle and strut mount assembly. In this study, the steering knuckle strut arm is modelled with SG iron material property. The strut mount is modelled with steel property. The material properties are given in Table 1.

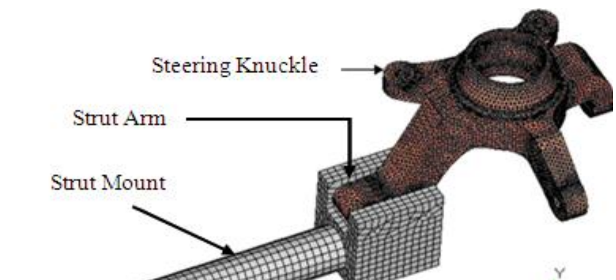


**Fig. 2: FE model of steering knuckle with strut mount assembly**

**Table 1: Material properties**

| Material Property | SG Iron | Steel |
|---|---|---|
| Young's Modulus | 172 GPa | 210 GPa |
| Poisson's ratio | 0.29 | 0.3 |
| Density | 7.10 mg/m$^3$ | 7.80 mg/m$^3$ |

### *2.2. Contact between strut arm and knuckle*

The contact has been established between the strut mount and strut arm of steering knuckle component in order to transfer the force. Contact problem is highly nonlinear and it is important to investigate the contact between two bodies. The FE model developed using Hypermesh package is imported into ANSYS software. In contact analysis, various components are replaced by pilot nodes that are connected to the steering knuckle. Hub bolt hole centreline is fixed in all degrees of freedom with pilot node while the loading is applied to the strut joints through struts. It provides accurate results in the strut arm region [9]. A node to surface contact is established between the strut mount and strut arm of steering knuckle. The lower control arm is fixed in all directions. The established contact elements are displayed in Fig. 3.

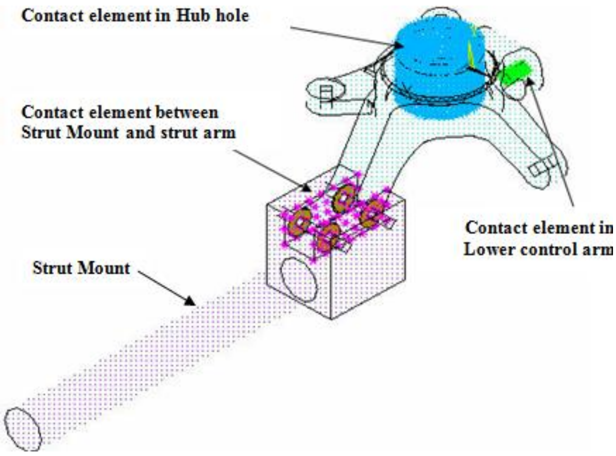


**Fig. 3: Contact and target surfaces model with strut mount assembly**

In this analysis TARGET 170 element is used at the nodes of strut mount surface to transfer the force to strut arm of the steering knuckle. To create a contact between surfaces of the strut mount and strut arm, CONTACT 175 element is used. The internal multipoint constraint (MPC) approach is combined with certain bonded and no separation contact definitions to define various contact assemblies and kinematic constraints. The internal MPC approach can overcome the drawbacks of the traditional contact algorithms and other MPC tools which are available in ANSYS software.

### *2.3. Structural analysis with actual loading conditions*

Structural analyses of a steering knuckle with strut mount assembly for two static load cases as given in Table 2 are carried out by assuming linearly elastic behaviour of the components and material. In this analysis, the static load is applied at the end of the strut mount assembly at a distance of 300mm from the centre axis of hub hole. These two loading cases are referred from the experimental procedure.

**Table 2: Load cases**

| Load Type | Direction of load | Load (kN) |
|---|---|---|
| Case 1 | X-Axis towards the brake caliper arm | 17.5 |
| Case 2 | X-axis towards the steering arm | 12 |

## 3. Experimental procedure

In this study, the steering knuckle with strut mount assembly is tested for static behaviour of SG material. The hydraulic actuator (Instron) is used in this testing with a capacity of 25 kN. The steering knuckle mounted on a dedicated fixture with its hub hole is shown in Fig. 4. The static load cases are applied to the strut arm of steering knuckle through the strut mount column.

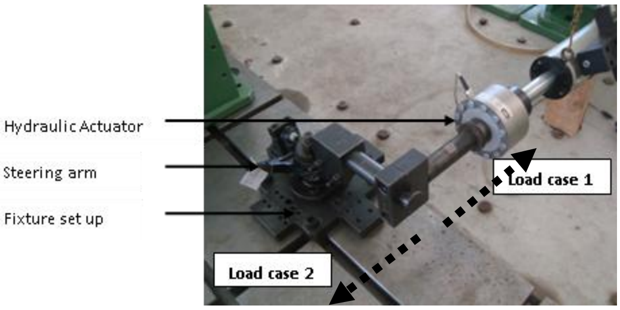


**Fig. 4: Static test setup**

## 4. Results and discussion

The results of static analysis for the two load cases are reported in this section using numerical method and experimental method. For Case 1 loading condition, the maximum deformation is predicted at the end of the strut mount column in FE analysis as presented in Fig. 5. The maximum deformation is obtained at same strut mount column end in the experimental result which is compared with FE analysis result as presented in Fig. 6. The deformation of strut arm column assembly is linearly increased for the given load cases which is matched well with the experimental results. For Case 2 loading condition, the maximum deformation is predicted at the end of the strut mount column of steering knuckle from FE analysis as shown in Fig. 7.

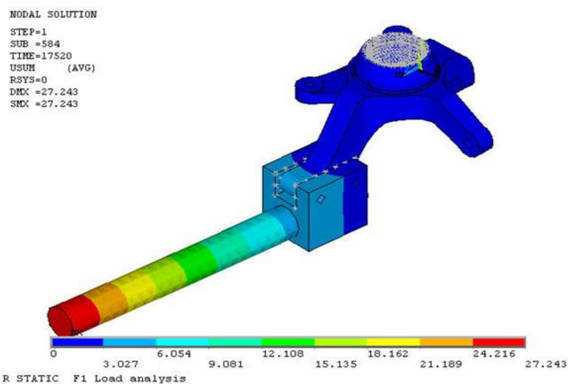


**Fig. 5: Deformation of strut arm column assembly for the load case 1 from FEA result**

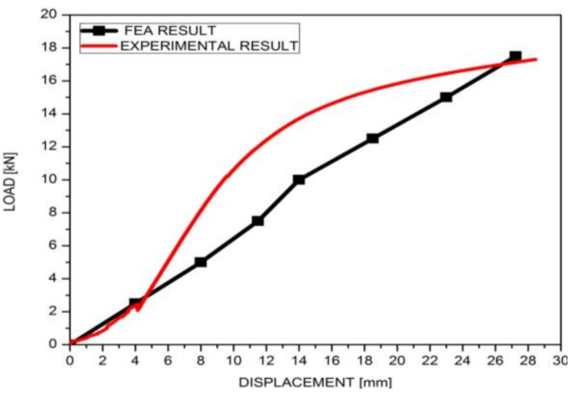


**Fig. 6: Comparison of load vs. Deformation value for load case 1**

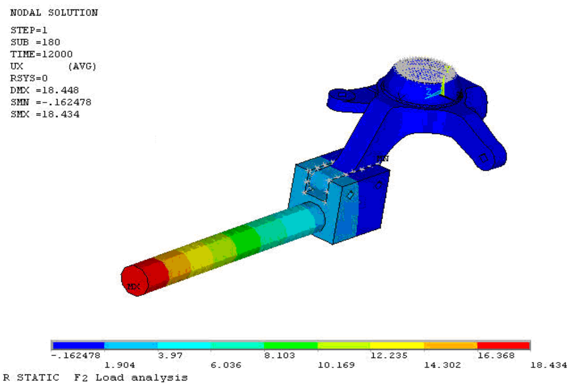


**Fig. 7: Deformation of strut arm column assembly for load case 2 from FEA result**

The maximum deformation is obtained at same strut mount column end in the experimental result which is compared with FEA result as presented in Fig. 8. The deformation of strut arm column assembly is linearly increased for given load cases which are matched well with experimental results.

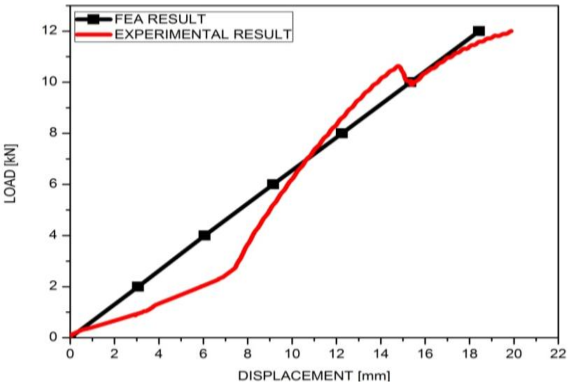


**Fig. 8: Comparison of load vs. deformation value for load case 2**

## 5. Conclusions

The strut arm column of steering knuckle is analyzed with strut mount assembly using experimental method. The experimental method is very expensive to analyze the performance of the steering knuckle component. In this study, the FE model was developed to evaluate the behaviour of a strut arm column of the steering knuckle component for SG iron material with the same boundary conditions as used in the experimental setup. The FE analysis result was compared with experimental results for two load cases. From this comparison, FE analysis results were closely matched with experimental result for both load cases. The FE analysis models and procedure are good enough for further analysis of steering knuckle to find out its structural performance under different loading conditions and it can reduce the number of prototypes and testing of steering knuckle components.

### CITATION:

The authors thank the management of Dr Mahalingam College of Engineering and Technology, Pollachi, Tamil Nadu, India and M/s Sakthi Auto Components Limited, Perundurai, Erode, Tamil Nadu, India for their support for carrying out this work.

## REFERENCES:


[1] G.K. Triantafyllidis, A. Antonopoulos, A. Spiliotis, S. Fedonos and D. Repanis. 2009. Fracture characteristics of fatigue failure of a vehicle's ductile iron steering knuckle, *J. Failure Analysis. and Prevention*, 9, 323-328. http://dx.doi.org/10.1007/s11668-009-9245-y.

[2] F. Pingqing, Z. Bo and Q. Long. 2011. The analysis on destruction forms of steering knuckle, *Proc. Third Int. Conf. Measuring Tech. & Mechatronics Automation*, Shanghai, China. http://dx.doi.org/10.1109/icmtma.2011.740.

[3] M. Zoroufi and A. Fatemi. 2006. Experimental durability assessment and life prediction of vehicle suspension components: A case study of steering knuckles, *Proc. IMechE Part D, J. Automobile Engg.*, 22c0, 1565-1579. http://dx.doi.org/10.1243/09544070JAUTO310.

[4] E.A. Azrulhisham, Y.M. Asri, A.W. Dzuraidah, N.M. Nik Abdullah, A. Shahrum and C.H. Che Hassan. 2010. Evaluation of fatigue life reliability of steering knuckle using Pearson parametric distribution model, *Int. J. Quality, Statistics and Reliability, Article ID. 816407*.

[5] R. D'Ippolito, M. Hack, S. Donders, L. Hermans, N. Tzannetakis and D. Vandepitte. 2009. Improving the fatigue life of a vehicle knuckle with a reliability-based design optimization approach, *J. Statistical Planning and Inference*, 139, 1619-1632. http://dx.doi.org/10.1016/j.jspi.2008.05.032.

[6] S. Houshito, Y. Watanabe, M. Goka and Y. Ishihara. 1989. Feasibility study on the application of high strength ductile iron to automotive chassis parts, *Int. J. Materials and Product Technology*, 4, 285-299.

[7] S. Vijayarangan, N. Rajamanickam and V. Sivananth. 2013. Evaluation of metal matrix composites to replace spheroidal graphite iron for a critical component steering knuckle, *Materials and Design*, 43, 532-541. http://dx.doi.org/10.1016/j.matdes.2012.07.007.

[8] V. Sivananth, S. Vijayarangan and N. Rajamanickam. 2014. Evaluation of fatigue and impact behavior of titanium carbide reinforced metal matrix composites, *Materials Science & Engineering Part A*, 597, 304-313. http://dx.doi.org/10.1016/j.msea.2014.01.004.

[9] R.L. Jhala, K.D. Kothari and S.S. Khandare. 2009. Component fatigue behaviors and life predictions of a steering knuckle using finite element analysis, *Proc. Int. Multi Conf. Engineers and Computer Scientists*, Hong Kong.


## EDITORIAL NOTES:


*Edited paper from National Conference on Technological advances in Mechanical Engineering TAME 2015, 20 August 2015, Chennai, India.*

*GUEST EDITOR: Dr. K. Umanath, Dept. of Mech. Engg., VelTech HighTech Dr Rangarajan Dr Sakunthala Engg. College, Chennai, India.*