© 2015 IJEDR | Volume 3, Issue 2 | ISSN: 2321-9939

Design and Shape Optimisation of a Steering Knuckle of an off-road vehicle using Solidworks and ANSYS Workbench

[1]Jatin Agrawal

[1]BTech Scholar,
[1]School of Mechanical and Building Science,
[1]Vellore Institute of Technology, Chennai, India

***Abstract* - Light weight, low fuel consumption and high durability are the primary demands for any vehicle. The steering knuckle is one of the most critical component of an off-road vehicle. The study involves designing of steering knuckle of an off-road vehicle, using Solidworks, and performing its shape optimization, using ANSYS Workbench, in order to meet the required strength parameters at the cost of minimum weight. The Study has been successful in designing the steering knuckle satisfying the required parameters and achieving a weight reduction of 42.8% in comparison with its initial model.**

***Index Terms* – Steering Knuckle, FEA, Shape Optimisation**

I. Introduction

Steering knuckle is a major component of an off-road vehicle. It is a part of vehicle suspension system which allows the steering arm to turn the front wheels. It supports the vertical weight of the vehicle and undergoes various static and dynamic loads. It is attached to the suspension components, Lower and Upper Arms from the roll cage, and tie rod. The Steering Knuckle accommodates spindle which supports the hub along with bearings. The Steering Knuckle is also known as upright.

Steering knuckle is not a standard part of vehicle component. So, the design may differ to fit the required suspension geometry. In an off-road vehicle, the design of the steering knuckle has to be very precise in order to minimize any kind of unwanted problems in suspension geometry. Since, the Steering Knuckle is produced in very high numbers, so a small reduction in its weight may lead to large scale savings. Therefore, the need of light weight components is vital. Also, the minimal weights will give substantial impact to fuel efficiency. It is also believed that the lighter steering knuckle may produce greater power and less vibration, resulting from the less inertia and reduction in centrifugal force. Although, it needs to be as light weight as possible, the component also must be very strong and rigid, due to harsh and high time varying loads for the race car driving conditions.

II. Methodology

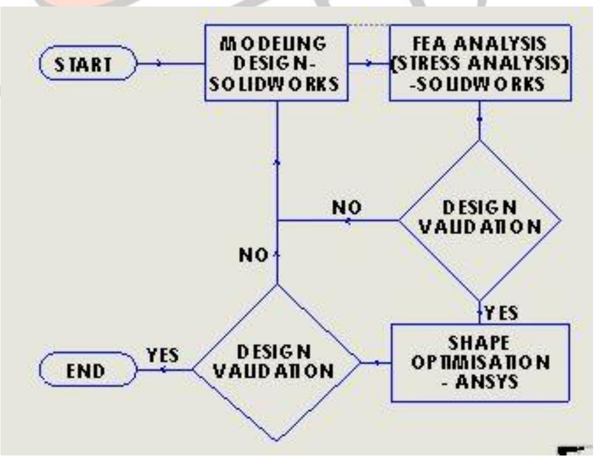

Figure 1: Flowchart of the design methodology adopted.

The study was carried out with suitable material selection as well as valid finite element analysis (FEA) and shape optimization studies. First part of this study involves modelling of steering knuckles and analysis of the stresses and displacement under actual load conditions. A CAD and FEA software; SolidWorks, is applied for modelling as well as for static analysis studies. Shape optimization is the second part, for which, ANSYS Workbench was utilised.

III. DESIGN OF STEERING KNUCKLE

Calculation of loads

While doing the analysis of Knuckle, forces to be considered are – Self-weight, Braking, Steering and Cornering/Lateral forces. In order to maintain the durability of the design, the model is subjected to extreme conditions as suggested by Sharma [3], considering G forces when all the forces are considered to be acting simultaneously. The weight of the vehicle in this case is supposed to be 2000N. Considering the weight, the magnitude of each of the forces were calculated and depicted in the table below.

Table 1: Loading Conditions

Self-weight	1g	250.00 N
Braking force	1.5g	375.00 N
Cornering/Lateral force at upper fixture	1.5g	965.29 N
Cornering/Lateral force at lower fixture	1.5g	1715.29 N
Steering force		50 N

Material Selection

The objective of the study is to design a steering Knuckle of minimum weight with maximum strength possible. For that, aluminum alloys are the best option for nowadays automobile industry due to them being light weight as well as has low density and compatible yield strength. In the present study, one type of aluminum alloy was selected which is Aluminum 6061 alloy. The other type of alloy available was Aluminium 6063 but Aluminium 6061 was given preference as it has better yield strength and durability. Table 2 depicts the physical and mechanical properties of the Aluminum 6061 alloy.

Table 2: Material Properties

Name	6061 Alloy
Yield strength:	5.51485e+007 N/m^2
Tensile strength:	1.24084e+008 N/m^2
Elastic modulus:	6.9e+010 N/m^2
Poisson's ratio:	0.33
Mass density:	2700 kg/m^3

Considering above facts, a CAD model of the steering Knuckle was prepared using Solidwork Software as shown in Fig 2. The model was designed, considering general suspension geometry parameters of an off-road vehicle. The initial model was designed comprising of basic design necessities. The center spindle hole was extruded further in order to provide adequate support, and tackle lateral moments generated by lateral forces. A grooving at the caliper mounting was provided keeping in mind the design of the caliper.

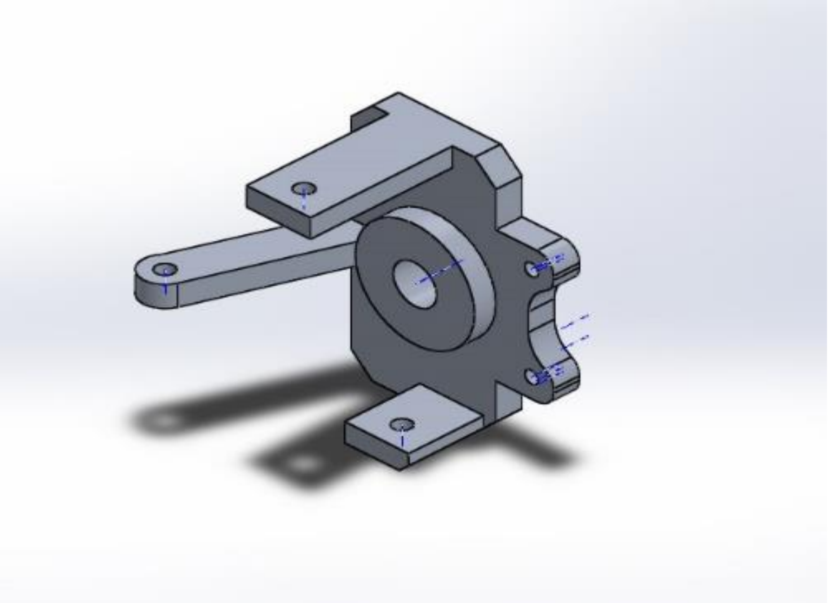

Figure 2: CAD Model of the Initial design

Material Selection

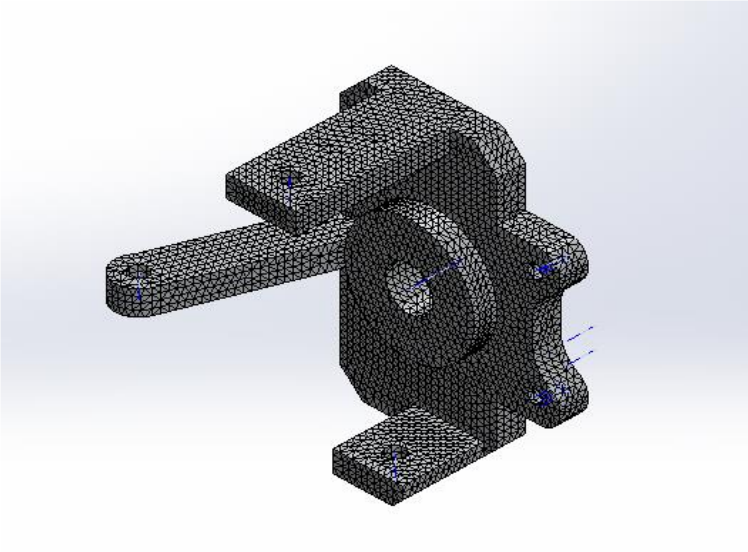

Figure 3: Meshing of the model

Table 2: Mesh Information

Mesh type	Solid Mesh
Element Size	0.142675 in
Tolerance	0.00713373 in
Mesh Quality	High
Total Nodes	84552
Total Elements	55460

Meshing of the initial model was performed using Solidworks mesh generator. The mesh parameters as given in Table 2. The meshed model is shown in the Fig. 3.

IV. STATIC ANALYSIS

Stress Analysis with respective maximum stress values obtained

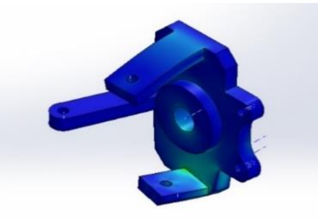

Figure 4:

9.14337e+007 N/m^2

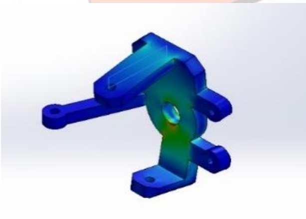

Figure 5:

8.08783e+007 N/m^2

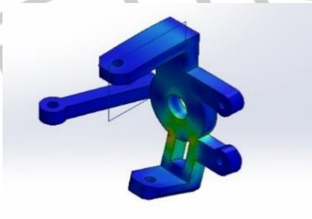

Figure 6:

9.92371e+007 N/m^2

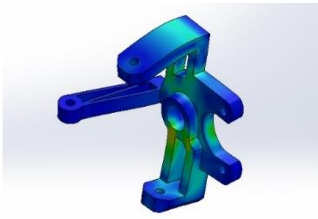

Figure 7:

4.52042e+007 N/m^2

Deformation Analysis with respective maximum deformation value obtained

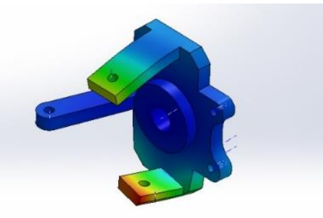

Figure 8:

0.116847 mm

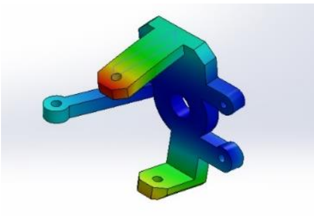

Figure 9:

0.30486 mm

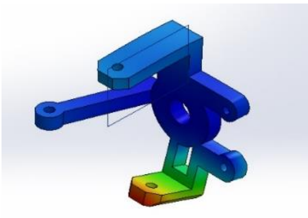

Figure 10:

0.360835 mm

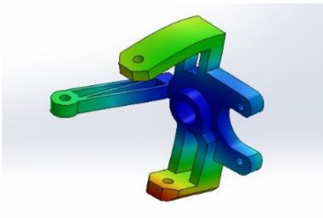

Figure 11:

0.0890976 mm

The static analysis of the model was performed using Solidworks simulation. All the loads as mentioned in Table 1 were applied and the respective maximum stress and displacement data were obtained. According to the data obtained, further modifications were made in order to get the adequate model satisfying the required parameters. The subsequent models obtained are shown in the Fig. 5, Fig. 6 and Fig. 7. From Fig. 7 and Fig. 11, it can be seen that the last model fulfill the required stress and deformation criteria, and the corresponding values are well within the safety limit.

V. Shape Optimisation

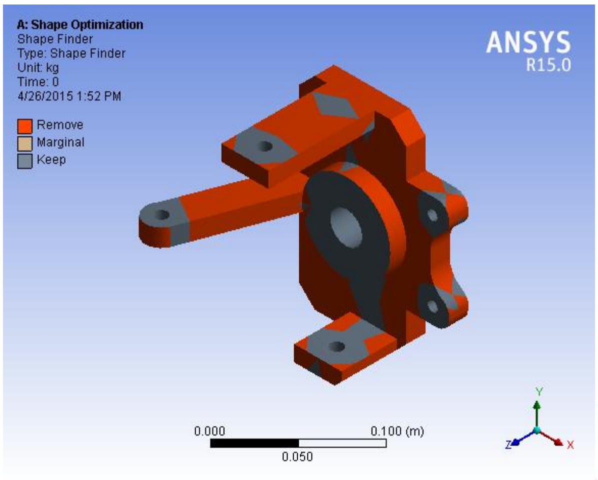

Figure 12: Initial Model

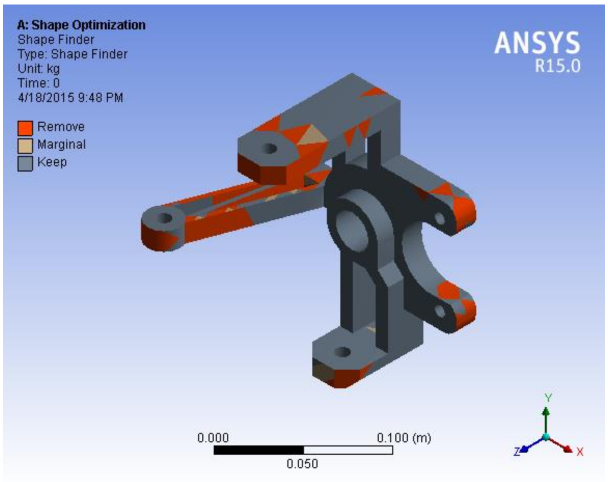

Figure 13: Final Model

The shape optimization of the initial model was performed using ANSYS Workbench. The shape optimization of the first model was performed after the static analysis. Shape optimization was carried out under the same loading conditions. The shape optimisation results obtained are shown in Fig. 12. It highlights the areas from where the excess material can be removed in order to reduce the weight. After the final model satisfying the static analysis was obtained, Shape optimisation of the final model was performed in surge of any further material reduction. It can be seen from Fig. 13 that no more considerable material removal can be done.

VI. Optimised Model

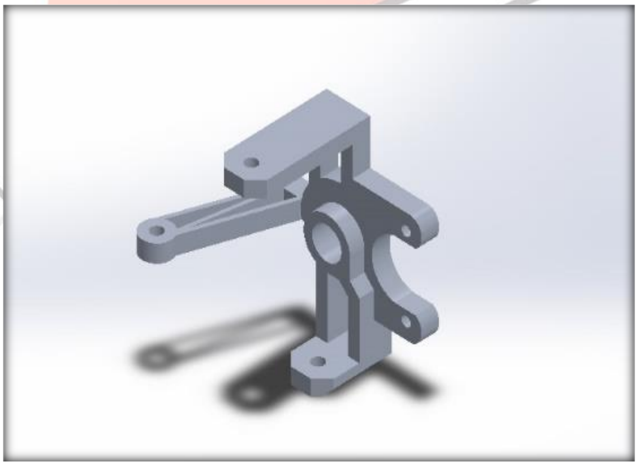

Figure 14: Final Optimised Model

The final optimized model is shown in Fig. 14. It has s maximum Von Mises stress value of 45.20 MPa and maximum displacement value of 0.09 mm at extreme conditions. The values are well within the safety limit.

VII. Results

Table 3: Summary of Results

Property	Initial Model	Final Model	Percentage
Weight	10.0665 N	5.75319 N	42.848%
Maximum Stress	9.14337e+007 N/m^2	4.52042e+007 N/m^2	50.54%
Max Displacement	0.116847 mm	0.0890976 mm	23.74%

Table 3 summarizes the results obtained after the stress, deformation analysis, and shape optimization process. It can be observed that the final model has been successful in getting a weight reduction of 42.848% from 10.066 N to 5.753 N. Also, a stress reduction of 50.45% was obtained reducing from 91.4 MPa to 45.204 MPa. The maximum deformation has also been reduced to 23.74%. All the values obtained in final model comply with the standard values accepted all over the world for automotives.

VIII. Conclusions

Initial model of knuckle is shown in Fig. 2. It has max. Stress of 91.43 MPa. After applying load and design constraints, stress analysis and shape optimization was performed. Fig. 14 shows the optimised model. The mass reduction for the front knuckle was found to be 42.8% compared to the initial model. The results are satisfactory considering shape optimization, with limited design space given and no change in material properties.

Stress analysis and shape optimisation can be used to reduce the weight of existing knuckle component by 42.80% while meeting the strength requirement, with limited design space given with or without change in material properties. Therefore, the overall weight of the vehicle can be reduced to achieve savings in raw material costs and consequently processing cost, as well as, improve fuel efficiency and reduce carbon emissions to help sustain the environment.

IX. References

[1] Rice, R. C., ed., 1997, "SAE Fatigue Design Handbook", Society of Automotive Engineers, Warrendale, 3rd Edition.

[2] B.Babu, M. Prabhu, P.Dharmaraj, R.Sampath, "Stress analysis on steering knuckle of the automobile steering system", International Journal of Research in Engineering and Technology, 2014, pp. 363-366.

[3] Mahesh P. Sharma, Dinesh S. Mevawala, Harsh Joshi, "Static Analysis of steering knuckle and Its shape Optimization", IOSR Journal of Mechanical and civil Engineering, 2014, pp. 34-38.

[4] Badil A. Jawad, Brian D. Polega, "Design of Formula SAE Suspension Components", SAE Technical Paper Series, 2002,

[5] Viraj Rajendra Kulkarni, Amey Gangaram Tambe, "Optimization and Finite Element Analysis of Steering Knuckle", Altair Technology Conference, 2013, India, pp. 2-7.