Optimization of Knuckle Joint by using FEA

Ms. Nilesha U. Patil[1*], Mrs. Rupali S. Sewane[2], Mr. Kashinath H. Munde[3]

[1]ME Student, Department of Mechanical Engineering, APCOER, Pune 09, India, Maharashtra, ***nupatil2@mail.com***
[2]Asst. Professor Department of Mechanical Engineering, SKNCOE, Pune 41, India, Maharashtra,
rupalisewane@gmail.com@gmail.com
[3]Asst. Professor Department of Mechanical Engineering, APCOER, Pune 09, India, Maharashtra,
kashinathmunde@mail.com

ABSTRACT

Knuckle joint is one of the most important components of an automobile. A knuckle joint is used to connect two rods whose axes either coincide or intersect and lie in one plane. These joints are used for different types of connections e.g. tie rods, tension links in bridge structure. The aim of the present paper is to study and calculate the stresses in Knuckle joint and optimize the model of Knuckle joint of TATA-709 vehicle . Currently, knuckle joint contains excess material, leads to increase in weight of the vehicle. Directly affects the mileage and cost. In this work, modelling of the knuckle joint is done using 3D software. Here CATIA V5 has been used for modelling. The simulation part is carried out using the Analysis software, ANSYS. With the Boundary constrains and the twisting moment applied, the knuckle joint is analyzed. Then using Topology optimization material is removed. Again, analysis is done on an optimized model for stresses and deformation and optimized values are tabulated.

Keywords: ***Knuckle joint, Stress analysis, Optimization FEA, ANSYS etc.***

---

1. INTRODUCTION

A knuckle joint is used to connect two rods which are under the action of tensile forces, when a small amount of flexibility or angular moment is necessary. However, if the joint is guided, it may support a compressible load. This joint can be readily disconnected for adjustments or repairs. The common examples of the knuckle joints are: link of a roller chain, tension link in a bridge structure, tie rod joint of roof truss, tie rod joint of jib crank, etc. A typical knuckle joint consists of three parts: an eye, a fork, and a knuckle pin. The end of one rod is formed into an eye and the end of other rod is formed into fork with an eye in each of the fork leg. The eye is inserted into the fork and after aligning the holes in the eye and fork, the knuckle pin is inserted through them. The knuckle pin has a head at one end and at the other end it is secured by a collar and a taper pin or split pin. The knuckle joint has wide applications such as bicycle chain, Tractors, Automobile wipers, cranes, robotic joints structural members.etc.

In this investigation, Knuckle joint of TATA-709 is used as component for study. CAD model of steering knuckle is developed in 3D modeling software CATIA V5. In the design optimization of the knuckle component, a weight should be minimized. While designing factors such as strength, stiffness and durability should be satisfied with design targets. Mass or weight reduction is becoming important issue in automobile industry. Here topology optimization method is used to reduce material. Topology optimization is a mathematical approach that optimizes material layout within a given design space, for a given set of loads and boundary conditions such that the resulting layout meets a prescribed set of performance targets.

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. Knuckle joint 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. The weight reduction of steering knuckle is done such that the strength, stiffness and life cycle performance of the steering knuckle are satisfied.

2. FE ANALYSIS

2.1 Geometric Modelling

CAD model of steering knuckle was developed in 3D modelling software CATIA V5. It consists of Fork end, Single eye, Collar, Split pin, knuckle pin.

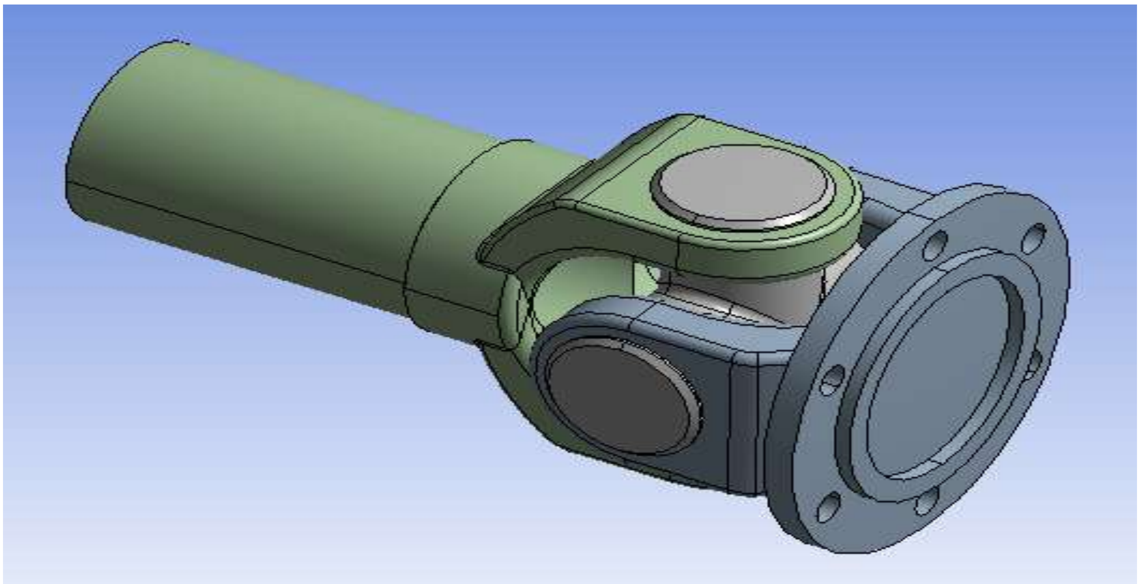


Fig. 2.1 Solid Model Knuckle Joint - Tata 709 Vehicle

2.2 Material Properties

There are several materials used for manufacturing of knuckle joint such as S.G. iron (ductile iron), white cast iron and grey cast iron. But grey cast iron mostly used. Forged steel are most demanding material for this application. For this Structural steel is used.

Properties of Outline Row 3: Structural Steel

	A	B	C
1	Property	Value	Unit
2	Material Field Variables	Table	
3	Density	7850	kg m^-3
4	Isotropic Secant Coefficient of Thermal Expansion		
6	Isotropic Elasticity		
7	Derive from	Young'...	
8	Young's Modulus	2E+11	Pa
9	Poisson's Ratio	0.3	
10	Bulk Modulus	1.6667E+11	Pa
11	Shear Modulus	7.6923E+10	Pa
12	Alternating Stress Mean Stress	Tabular	
16	Strain-Life Parameters		
24	Tensile Yield Strength	2.5E+08	Pa
25	Compressive Yield Strength	2.5E+08	Pa
26	Tensile Ultimate Strength	4.6E+08	Pa
27	Compressive Ultimate Strength	0	Pa

Table 1. Properties of Material

 ISSN: 2321-8169
478 – 484

2.3 Meshing

The process for generating a mesh of nodes and elements consists of three general steps:

- CATIA offers a large number of mesh controls from which you can choose as needs dictate.
- Set the element attributes.
- Set mesh controls (optional).

It is not always necessary to set mesh controls because the default mesh controls are appropriate for many models. If no controls are specified, the program will use the default settings to produce a free mesh. Alternatively, you can use the Smart Size feature to produce a better quality free mesh.

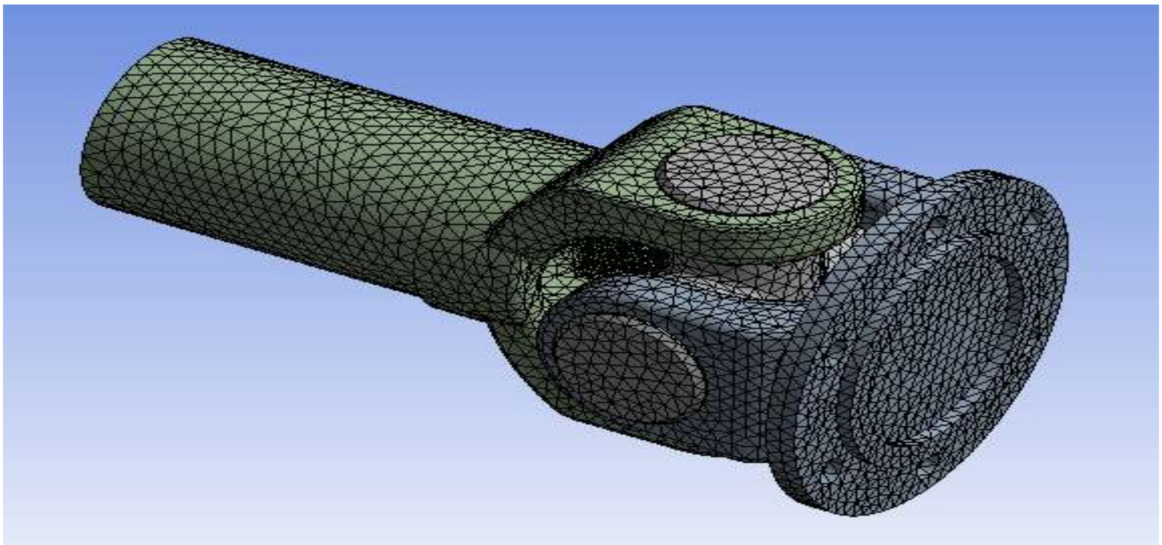


Fig. 2.2 Meshing of Model of Knuckle Joint

- Element Type: Second order Tetrahedron
- Elements count: 132722
- Nodes count: 195359

2.4 Boundary Condition

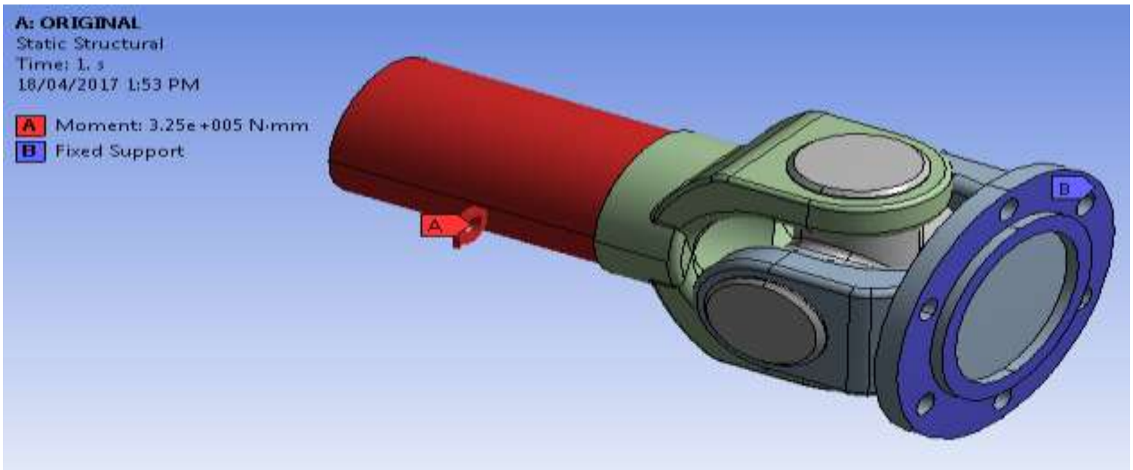


Fig. 2.3 Boundary Condition in ANSYS

The knuckle joint is fixed at one end at point B on the fork side. The moment of 3.25e+005 N-mm is applied on the other end at A.

2.5 Von Mises

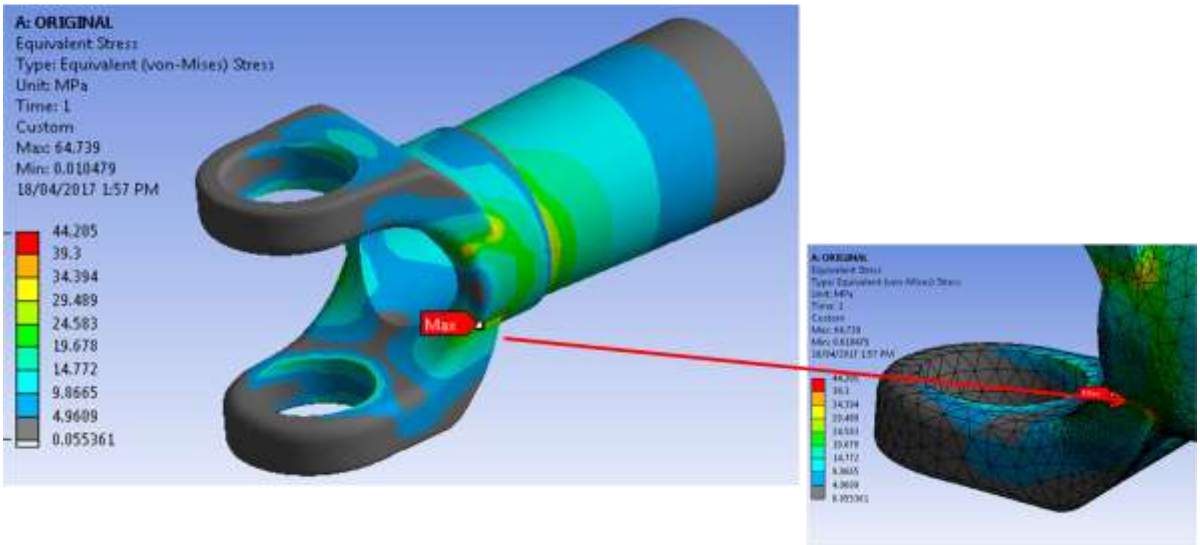


Fig. 2.4 Von Mises stresses by using ANSYS

Application of moment with given boundary conditions produces different stresses, the maximum stress is developed at the fork end which is 44.205 MPa.

2.6 Deformation

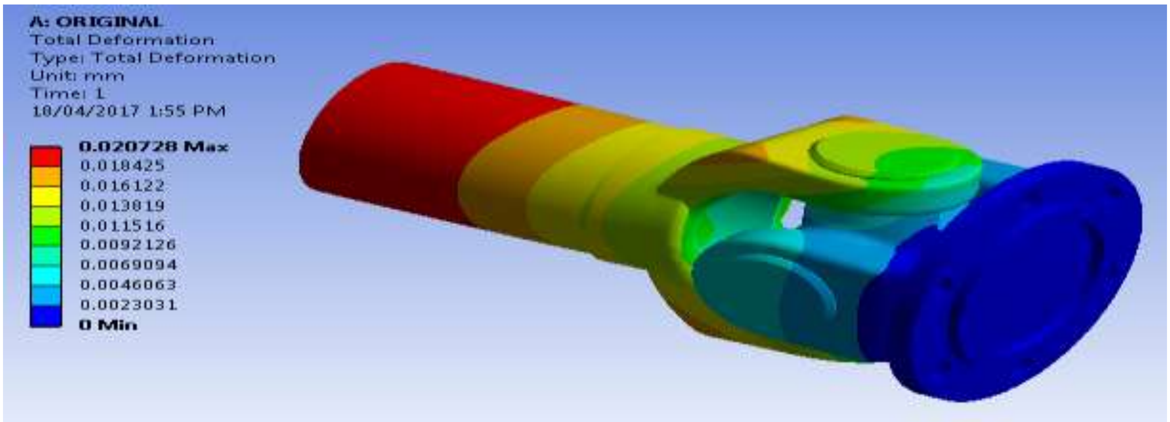


Fig. 2.5 Deformation Results in ANSYS

Maximum deformation found in the given model is 0.020728 mm and that of the minimum is 0.00230 mm.

3. OPTIMIZATION

Topology optimization is a mathematical approach that optimizes material layout within a given design space, for a given set of loads and boundary conditions such that the resulting layout meets a prescribed set of performance targets.

All manufacturing enterprises strive to develop the optimized product commonly by reducing the weight while ensuring they produce cost effective products that meet their design functionality and reliability. Structural optimization tools like topology and shape optimization along with manufacturing simulation are becoming attractive tools in product design process. These tools also help to reduce product development time. Shape optimization gives the optimum fillets and the optimum outer dimensions. Objective of this investigation is to reduce weight of steering knuckle of rear driven vehicle having double wishbone type suspension system. This paper focuses of static analysis and shape optimization. Finite element analysis has been used to implement optimization and maintaining stress and deformation levels and achieving high stiffness. Reduction of weight has been one the critical aspects of any design. It has substantial impact on vehicle performance, fuel efficiency and in turn reduces the emissions.

3.1 Optimized Solid Model

In optimization 5 mm material removed from top and bottom surfaces of fork end to reduce weight.

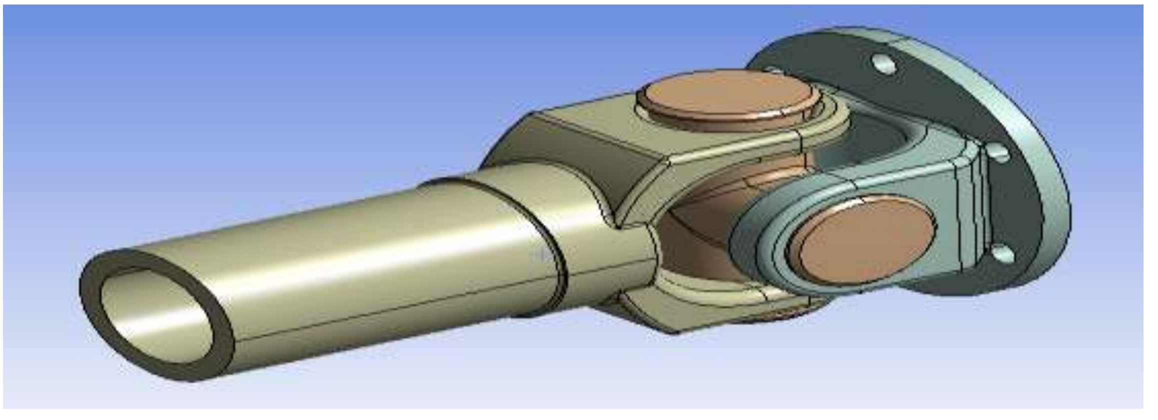
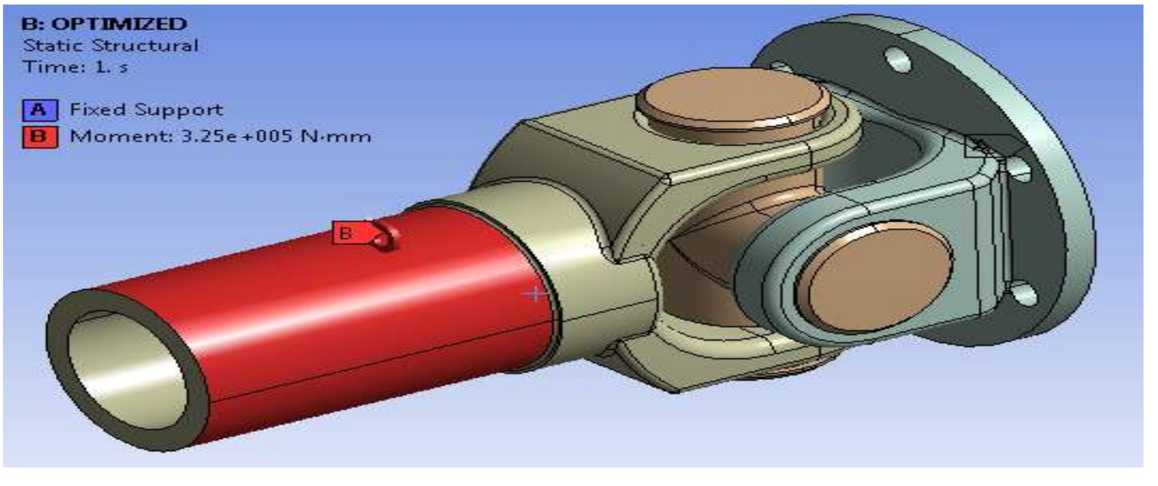


Fig.3.1 Optimized Solid Model

3.2 Boundary Condition for Optimized Model

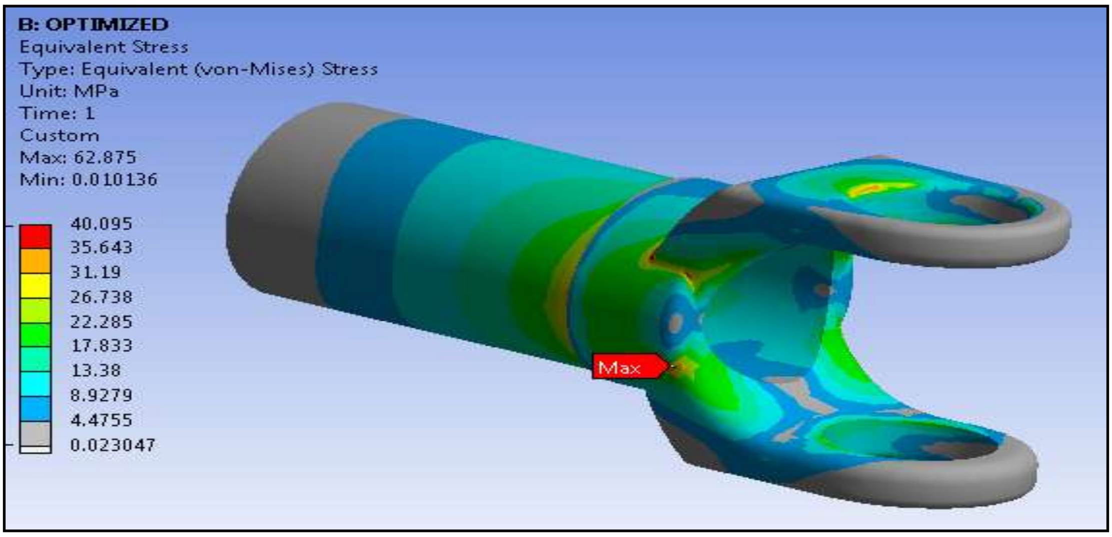


3.2 Boundary Condition

Optimized model of knuckle joint is fixed at one end at point A. The moment of 3.25e+005 N-mm is applied on the other end B.

3.3 Von mises stress for optimized model

Maximum stress is 40.095MPa at fork end and it is 62.875 Mpa for the whole optimized model knuckle joint.



3.3 Von mises stress for optimized model

3.4 Deformation results for optimized model

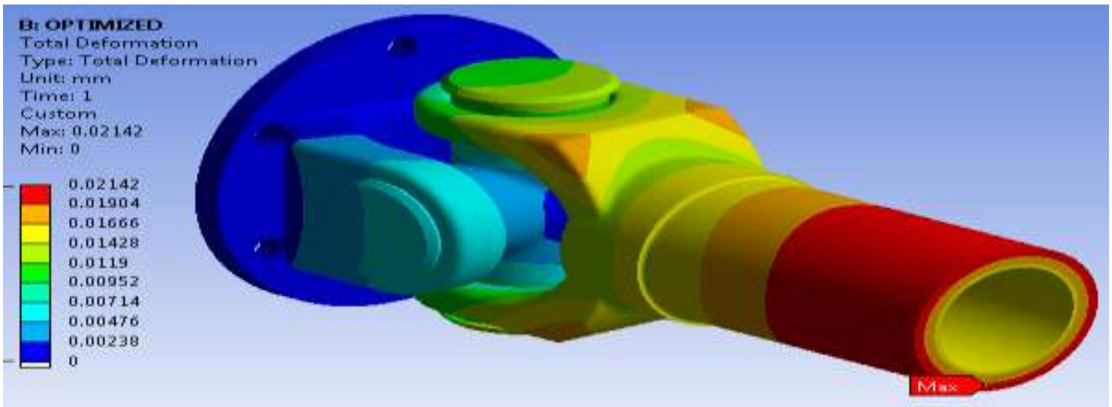
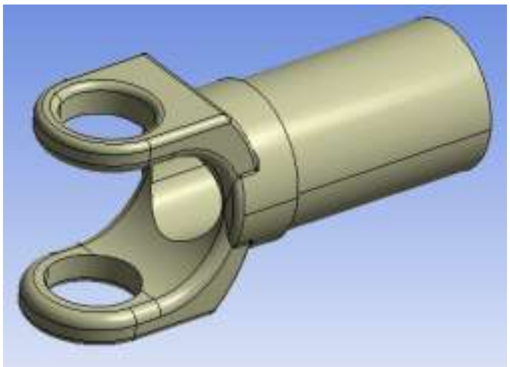


Fig. 3.4 Deformation in optimized Model

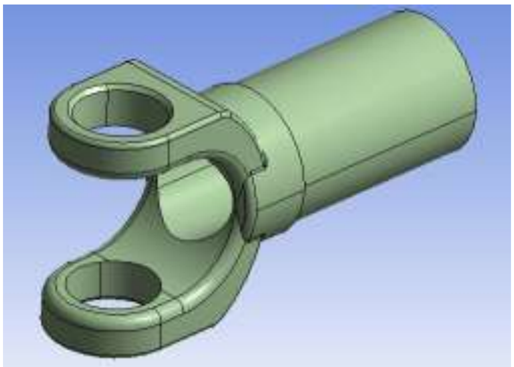
Maximum deformation for optimized model is 0.02142 mm and that of the minimum is 0.00238mm.

3.5 Comparison Of Original Model And Optimized Model

Original fork end mass (2.2039 kg) was reduced to Optimized Model (2.0399 kg) i.e around 8% reduction is achieved through optimization.



Original Model



Optimized Model

4. RESULTS

		Initial Design	Optimized Design	% Reduction
Stress (Mpa)	Max	44.205	40.095	
	Min	0.0553	0.0230	
Deformation (mm)	Max	0.0207	0.0214	
	Min	0.00230	0.00238	
Mass (Kg)		2.2039	2.0399	8%

Table-2 Summary of results

5. CONCLUSION

From the results of finite element analysis it is observed that the maximum stress value is within the safety limit. The maximum displacement value is also very less. Shape optimization method used in this study in reducing the mass of fork end of knuckle by 8%. This optimization process also gives small change on the displacement. It means that change of volume and shapes doesn't influence significantly to stiffness of the structure. Therefore, the overall weight of the vehicle can be reduced to achieve savings in costs and materials, as well as, improve fuel efficiency and reduce carbon emissions to sustain the environment.

ACKNOWLEDGMENT

Though efforts have been taken to complete this paper, it would not have been possible without the kind support and help of some individuals and organization. Authors would like to extend their sincere thanks to all of them.

Authors are highly indebted to Prof. Thakare S.B. (Principal, APCOER, Parvati) and Dr. A. V. Deshpande (Principal, SKNCoE, Pune) for providing with all the necessary facilities for the research. Authors wish to express their thanks to Asst. Prof. Kashinath Munde (P.G.Co-ordinator APCOER, Parvati) for his guidance for providing necessary information regarding this work. Authors also wish to extend their gratitude towards Dr. A.P.Pandhare (HOD, Mechanical Engg.,SKNCoE,Pune).

REFERENCES

[1] Mahesh P. Sharma, Denish S. Mevawala, et.al, *Static Analysis of Steering Knuckle and Its Shape Optimization, IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684, p-ISSN: 2320-334X*

[2] R. d'Ippolitoa, M.Hackb, et.al, *Improving the fatigue life of a vehicle knuckle with a reliability-based design optimization approach, Journal of Statistical Planning and Inference 139 (2009) 1619 -- 1632*

[3]Sangamesh B. Herakal, Ranganath Avadhani, Dr.S.Chakradhar Goud, *Structural Static Analysis Of Knuckle Joint, International Journal of Engineering Research and General Science Volume 4, Issue 2, March-April, 2016, ISSN 2091-2730*

[4]Pankaj Dulani, S. A. K. Jilani, *Diameter and Spiral Thickness Optimization of Knuckle Joint Using Neural Network, International Journal of Science and Research (IJSR) ISSN (Online): 2319-7064*

[5] Purushottam Dumbre, A. K. Mishra,et.al , *Structural Analysis of Steering Knuckle for Weight Reduction, International Journal of Emerging Technology and Advanced Engineering ISSN 2250-2459, ISO 9001:2008 Certified Journal, Volume 4, Issue 6, June 2014*

[6] Ameya Bhusari, Aditya Chavan, et.al , *Fea & Optimisation Of Steering Knuckle Of Atv, Proceedings of 54th IRF International Conference, 8th May, 2016, Pune, India, ISBN: 978-93-86083-10-4*

[7] Sanjay Yadav, Ravi Kumar Mishra, Varish Ansari, et.al, *Design and Analysis of Steering Knuckle Component, International Journal of Engineering Research & Technology (IJERT), ISSN: 2278-0181, Vol. 5 Issue 04, April-2016*