IJSRD - International Journal for Scientific Research & Development| Vol. 4, Issue 08, 2016 | ISSN (online): 2321-0613

Discerning the Results of Topology Optimization to Realize Mass Reduction for Steering Knuckle Arm

Pranjal Ramesh Kelkar[1] **Prof. V. R. Bajaj**[2] **Swapnil S.Kulkarni**[3]
[1,2,3]Department of Mechanical Engineering
[1,2,3]Alard College of engineering & management, Pune, India

Abstract— The work is done to change geometry for the Steering Knuckle Arm in an attempt to reduce the weight of the component. While doing so, the mass is removed from the specific regions of the part which do not contribute to the stresses acting on the component. This is observed in the contour plot for stress over the Finite Element Model. This is further analyzed using Topology Optimization which expedited the areas which are the non-contributing regions through 3D mass densities pertaining to the stress. The modified geometry created which is feasible for manufacturing. The experimentation for the component is done to validate the alternative geometry in terms of its performance relating to its stiffness. The response for stiffness is aimed to be maintained during the transition from the benchmark geometry to the new geometry. The precise geometry addressing the functional need of the component shall be expedited to stay compatible for the manufacturing process. The work is concluded with the recommendation for the new design with a reduced mass of the material.

Key words: Steering knuckle, Weight reduction, Topology Optimization

I. INTRODUCTION

This paper attempts to present the further work done as published in the Review paper for my publication earlier. The results for analysis and experimentation has been compiled in this paper.

II. SCOPE / OBJECTIVES

1) Study the literature related to working conditions and functional requirement of steering knuckle.
2) Study the 3D geometry for FE modeling.
3) Identify the material properties and load, boundary condition for given geometry.
4) Study the analysis results of a given geometry and identify the scope for optimization using FEA tool.
5) Experimentation of a benchmark model for validation purpose.
6) Propose the best suitable variant as per requirement.

III. METHODOLOGY

The following methodologies could be deployed for analyzing the problem and finding appropriate solution. By mathematical calculation we can find out forces acting on component. Finite Element Analysis could offer insights into the behavior of the component while the same is subjected to loading. Experimentation shall provide as methodology for validating the FEA results.

A. Finite Element Analysis:

Finite element analysis is carried out in 3 steps-

1) Pre-processing: For pre-processing, HyperMesh 13 is used. In HyperMesh CAD model is imported in the form of neutral format. Meshing criteria is decided as per geometry. Material property, load and boundary conditions are applied to meshed model.
2) Processing: For processing, Optistruct solver would be used to solve the model.
3) Post-processing: HyperView interface would be used to study the results in the form of graphs or contour.

B. FEA analysis of Existing Component:

For CAD model CATIA V5 interface is used. Meshing is carried out in HyperMesh 13.0 interface. Tetrahedral element type is used for meshing. No of elements and nodes in meshed model are 47,214 and 77,627 respectively. MAT 1 material card used for static analysis. Material properties assigned to components are as follows:

Material- Forged Steel SAE Grade 11v37
$E=2.1\times10^5 N/mm^2$
Poisson Ratio=0.3
$Density=7.89\times10^{-9} Ton/mm^3$

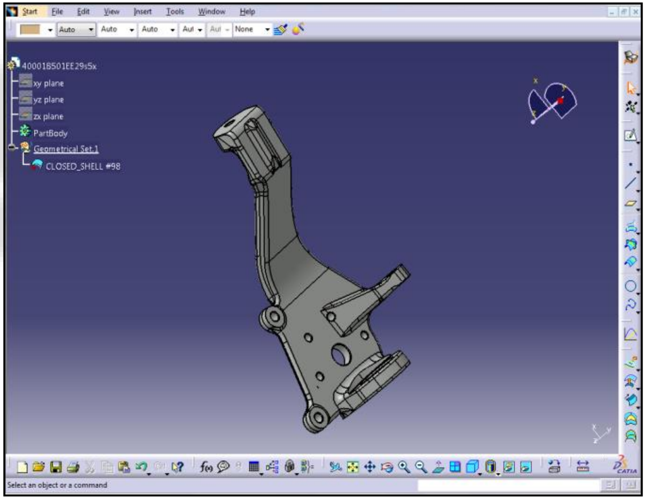


Fig. 1: Steering knuckle CAD Model

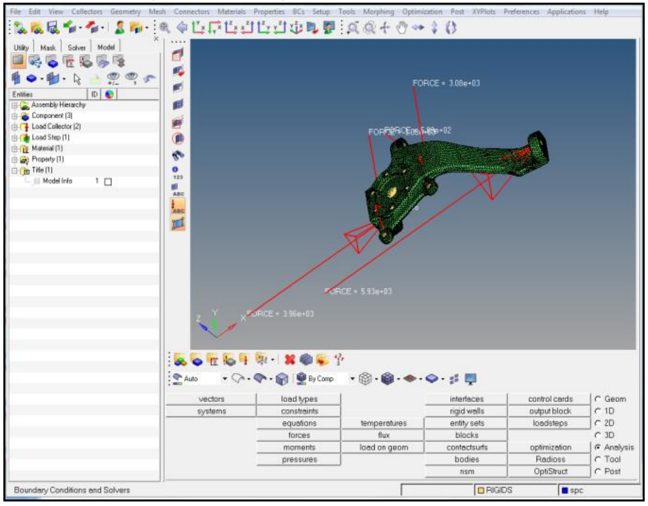


Fig. 2: Steering knuckle arm in meshing interface

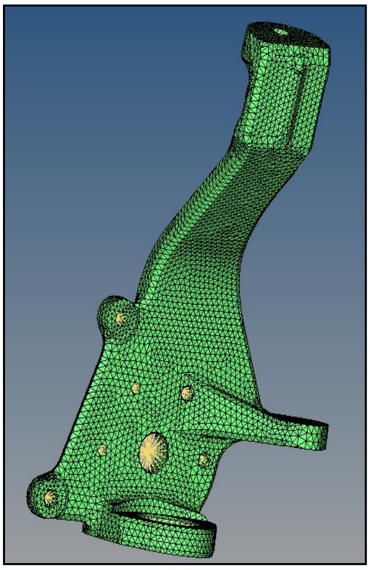
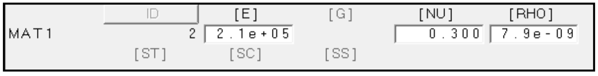


Fig. 3: Meshed model of steering knuckle arm



	ID	[E]	[G]	[NU]	[RHO]
MAT1	2	2.1e+05		0.300	7.9e-09
	[ST]	[SC]	[SS]		

Fig. 4: Material Properties

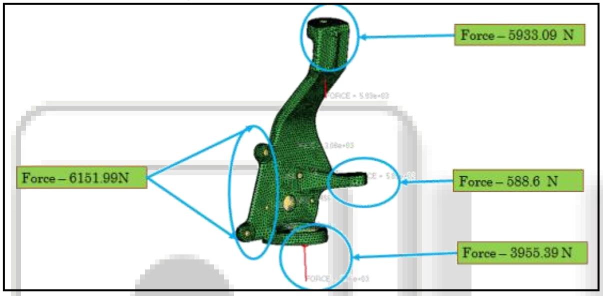


Fig. 5: Load and boundary condition applied to geometry

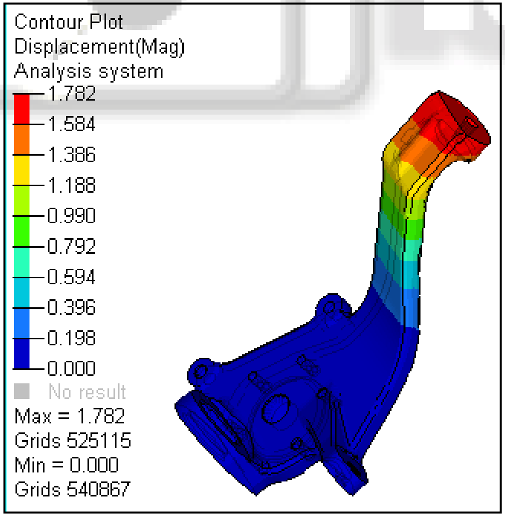


Fig. 6: Displacement plot

From the fig. the nature and extent of displacement can be studied over the contour plot. The max displacement of 1.782 mm is found along the longest arm.

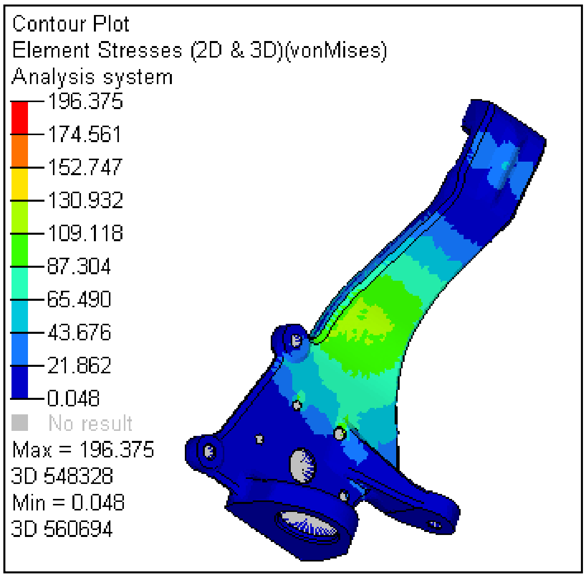


Fig. 7: Stress Plot

Figure No. 7 depicts the distribution of stresses along the geometry of the component. The maximum stress recorded as 196.375 MPa is limited to a very insignificant span of the geographical region within the component. The stresses peaking from 130.932 to 153.747 MPa are observed at the central region of the component.

C. Topology optimization of Existing Component:

In Topology optimization need to defined designed & non designed area so that to remove material from design area where no stress or displacement is observed as shown in below figure no 8 & 9

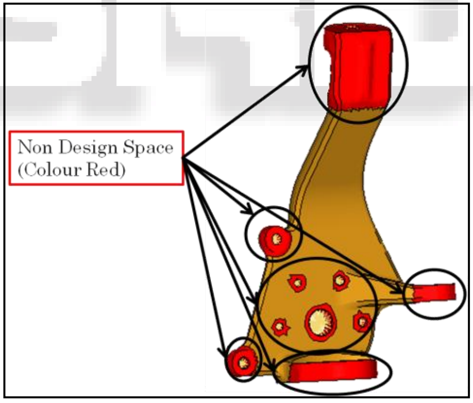


Fig. 8: Non design space

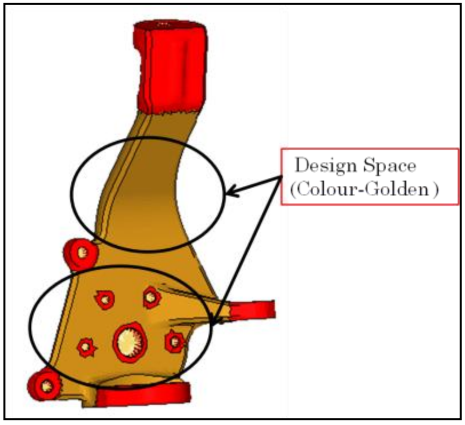


Fig. 9: design space

After which by using Optistruct as solver we find out the solution for Topology optimization as shown in below fig. 10

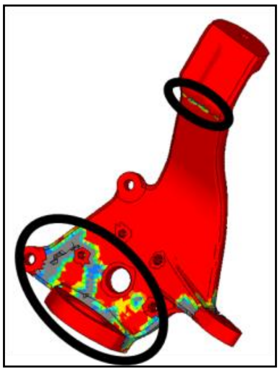


Fig. 10: Optimization result

Area shown in result, Material removal at these regions could affect the structural performance of the part since the load carrying features are located at the extremities around the same region.

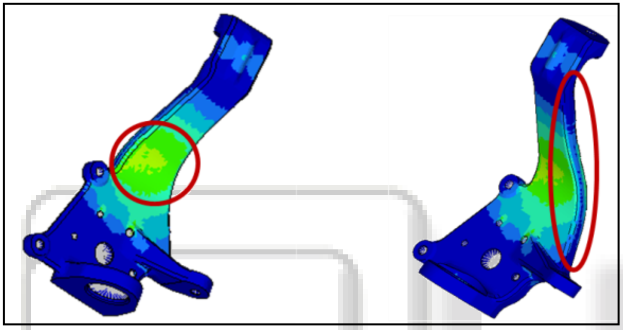


Fig. 11: Feasible solution

Material removal could be explored at the region on one side of the part. Closed cavity around the central region shall offer discontinuity to the lines of stress passing through this area. For region marked the material could be chopped off from the side without causing loss of structural strength at this region. As shown in below fig. 12

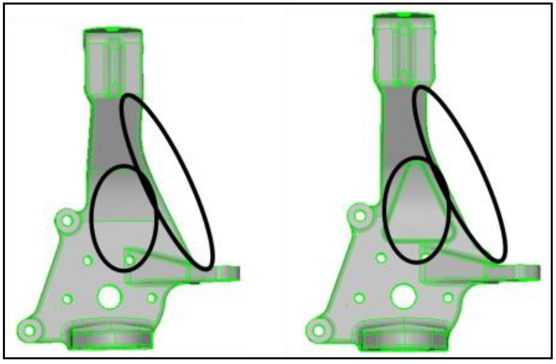


Fig. 12: Material removal

As shown in above fig material is removed is also feasible for manufacturing process.

Below fig 13 & 14 shows the FEA results for modified / optimized component.

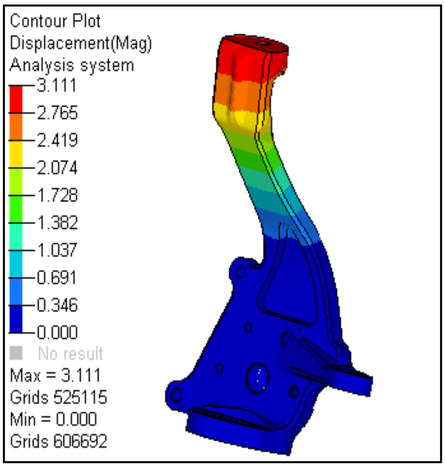


Fig. 13: Displacement Plot (optimized component)

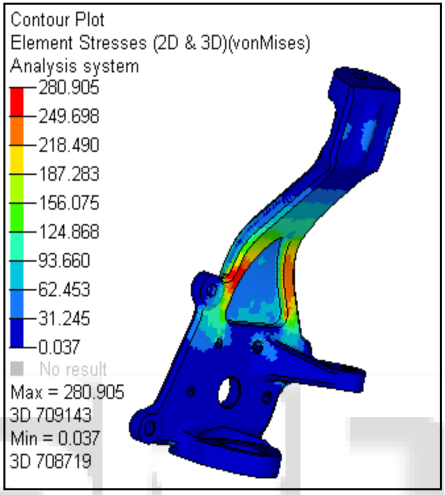


Fig. 14: Stress Plot (optimized component)

IV. RESULTS & DISCUSSION

From results outcome through FEA both for Existing component & optimized component is shown which helps to understand comparison of behavior of component before & after optimization. The comparison is as shown in below fig.15

	Displacement (mm)	Von-Mises Stress (MPa)	Mass (Kg)	% Mass reduction
Benchmark	1.78	196.37	7.898	0%
Optimized	3.11	280.905	7.166	9.27%

Fig. 15: Comparison Table

From stress and displacement plot, it is observed that max displacement observed in the steering knuckle is 1.782 mm and stress is 196.375 MPa where as in optimized component displacement found 3.11 mm & stress is 280.9 MPa. Result also shows 9.27% (0.732 Kg) mass reduction. The results obtained in optimized component is acceptable as the results are within limits of material properties

A. *Experimental Set Up:*

Figure 16 shows th test setup for determining the stiffness of the steering knuckle. The gradually increasing load will be applied and corresponding deformation is determined. The load from the load cells present on the UTM machine will be applied gradually. Display attached to the machine will give a corresponding plot for load Vs displacement i.e. stiffness of the component.

Fig. 16: Test setup for physical experimentation to calculate stiffness

V. CONCLUSION

Optimized steering knuckle is analyzed under various loads from connected parts. Max Displacement Observed 3.11mm. Von-Misses stress observed 280.905 MPa. Stresses are below yield limit (Yield strength for Forged Steel SAE Grade 11v37 > 556 MPa).

By above results shows that Topology optimization method is helpful to reduce the weight of component. Reduction of weight by 9.27% keeping maximum stresses & displacement within limit. The stresses observed in optimized component is within limit so it shows that change in component weight can increase the fuel efficiency of vehicle, reduce weight, reduce cost of Component.

REFERENCES

[1] Mahesh P. Sharma, Denish S. Mevawala, Harsh Joshi, Devendra A. Patel. “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, PP 34-38.

[2] Purushottam Dumbre, A. K. Mishra, V. S. Aher “Structural Analysis of Steering Knuckle for Weight Reduction”. International Journal of Emerging Technology and Advanced Engineering. Website: www.ijetae.com (ISSN 2250-2459, ISO 9001:2008 Certified Journal, Volume 4, Issue 6, June 2014)

[3] J. K. Kim, Y. J. Kim, W. H. Yang, Y. C. Park, K.-H. Lee. “Structural design of an Outer tie rod for a passenger car”. International Journal of Automotive Technology, Vol. 12, No. 3, pp. 375−381 (2011), DOI 10.1007/s12239−011−0044−6.

[4] Jie Zhou & Lei Lin & Yan Luo, “The multi-objective optimization design of a new closed extrusion forging technology for a steering knuckle with long rod and fork.” Springer-Verlag London 2014, Int J Adv Manuf Technol (2014) 72:1219–1225, DOI 10.1007/s00170-014-5742-6.

[5] Viraj Kulkarni, Amey Tambe, “Optimization and Finite Element Analysis of Steering Knuckle”. Altair Technology Conference 2013 India.

[6] Lothar Harzheim, Gerhard Graf, “A review of optimization of cast parts using topology optimization”. Springer-Verlag 2005 Struct Multidisc Optim (2006) 31: 388–399, DOI 10.1007/s00158-005-0554-9

[7] Kamlesh Chavan, Sanjay Deodas, Swapnil Kulkarni.“Mass reduction for steering knuckle arm in suspension system through topology optimization in CAE”. International Journal of Advanced Engineering Research and Studies E-ISSN2249–8974

[8] W. M. Wan Muhamad, E. Sujatmika, Hisham Hamid and Faris Tarlochan, “Modeling, Simulation and Optimization Analysis on Steering Knuckle Component for Purpose of Weight Reduction”

[9] Bo Myung Kim, Jae Won Kim, Il Dong Moon and Chae Youn Oh “Optimal combination of design parameters for improving the kinematics characteristics of a midsize truck through design of experiment”. Journal of Mechanical Science and Technology 28 (3) (2014) 963~969, www.springerlink.com/content/1738-494x, DOI 10.1007/s12206-013-1167-7