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]
[1,3]Student [2]Assistant Professor
[1,2,3]Department of Mechanical Engineering
[1,2,3]Alard College of Engineering & Management, Pune, India

Abstract— This work aims to investigate the probable alternative geometry for the Steering Knuckle Arm in an attempt to reduce the weight of the component. While doing so, the mass needs to be removed from the geographical regions of the part that do not contribute to the stresses supported by the component. This could be visualized over the contour plot for stress over the Finite Element Model. This is further analyzed using Topology Optimization for facilitating the review of the non-contributing regions through 3D mass densities pertaining to the stress. The alternative geometry generated during the research work should be feasible for manufacturing. The experimentation for the component shall 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 shall be concluded with the recommendation for the new design with a reduced mass of the material.

Key words: Steering Knuckle, Altair Hyperworks, FEA, Weight Reduction, Topology Optimization

I. INTRODUCTION

Manufacturing processes face major competitions in automotive industry to produce lighter, cheaper and more efficient components that exhibit more precise dimensions, needless machining and require less part processing. Technology leaders follow two main routes to improve these processes; development and integration of currently available technologies, and invention of new technologies optimized with respect to various design or commercial aspects. The value of the know-how depends on a process's ability to usefully differentiate its capabilities from those of its competitors. Material mechanical properties and manufacturing parameters play decisive roles and the weaknesses and strengths of each manufacturing process need to be available to designers in these respects, to enable them to choose the optimum choice for the specific component and application. In automotive industry, designers have a wide range of materials and processes to select from. Steel and aluminum forgings and castings, cast irons, and powder forgings have found broad applications in automotive safety-critical systems. The competition is particularly acute in the chassis, and it is not unusual to find a range of different materials and manufacturing technologies employed within modern chassis components.

In automotive suspension, a steering knuckle is that part which contains the wheel hub or spindle, and attaches to the suspension components. It is variously called a steering knuckle, spindle, upright or hub, as well. The wheel and tire assembly attach to the hub or spindle of the knuckle where the tire/wheel rotates while being held in a stable plane of motion by the knuckle/suspension assembly. The arm of the knuckle joint sticks out, to which the steering mechanism attaches to turn the knuckle and wheel assembly. The forces exerted on this assembly are of cyclic nature as the steering arm is turned to maneuver the vehicle to the left or to the right and to the centre again. The weight of the vehicle is going on increasing due to additional luxurious and safety features. The increasing weight of the vehicle affects the fuel efficiency and overall performance of the vehicle. Therefore the weight reduction of the vehicle is the real need of today's automotive industry.

II. LITERATURE REVIEW

In the interest of Optimization techniques lot of research in completed & in process. Out of which many articles are focused on Topology Optimization and Shape Optimization of component. The main interest of this research is to reduce or optimize the weight of component to increase the efficiency of machine & reduce the cost of component. In optimization techniques research authors using finite element analysis to find out stresses which occur on component before optimization. From this reference by finding out the non-design area researcher remove the material to optimize the component [1][2][5][7][8].

It also proves that by changing the material properties or material the mechanical properties can be changed by which it helps to reduce the weight of component [3].

Another reference paper [4] Authors targeted on optimization of manufacturing cost of the component which reduced the manufacturing cost of component. In which they change existing forming process change to closed extrusion die process to achieve complex shape without flash forging.

Researcher also co-relate optimization technique with manufacturing feasibility .In optimization technique we use Optistruct as solver in FEA which can give us optimized solution for component but it's possible that manufacturing feasibility is not checked in this option. So researcher develop a program TOPSHAPE [6] which also check whether solution given by Optistruct is feasible or not.

In reference paper [9] researcher combine the parts into the assemblies and consider it as single component & try to optimize the components which also successfully done.

From above literature review it's found that the optimization can be done with shape optimization or Topology optimization. Both techniques are used in industries for optimization purpose following are definitions for some optimization techniques.

III. 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.

IV. METHODOLOGY

The following methodologies could be deployed for analyzing the problem and finding appropriate solution. Preliminary investigation could be pursued using mathematical treatment. Finite Element Analysis could offer insights into the behavior of the component while the same is subjected to loading. Experimentation shall provide as an alternative methodology for validating the hypothesis.

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.

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
$E=2.1x10^5 N/mm^2$
Poisson Ratio=0.3
Density=$7.89x10^{-9}$ Ton/mm^3

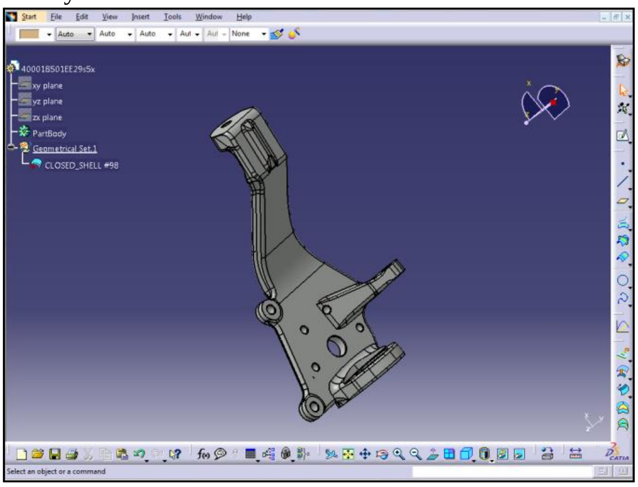

Fig. 1: Steering knuckle CAD Model

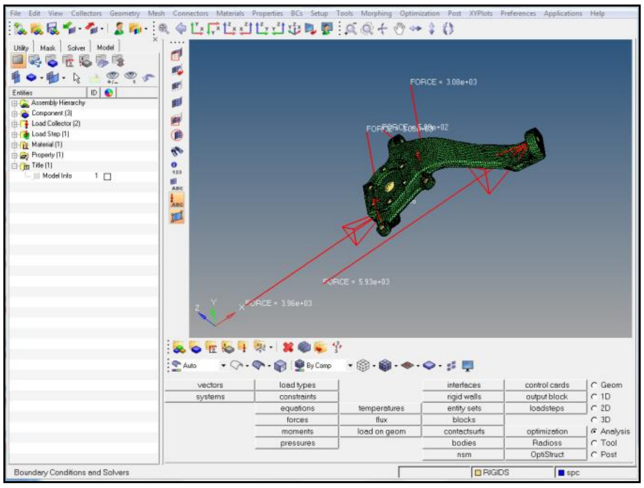

Fig. 2: Steering knuckle arm in meshing interface

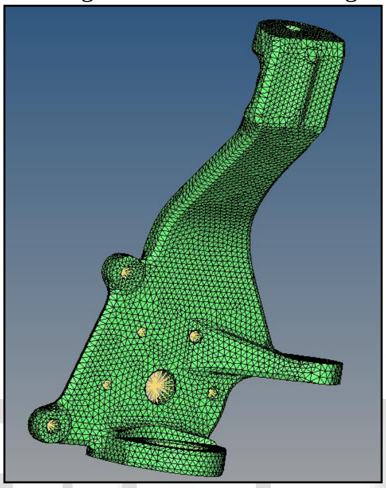

Fig. 3: Meshed model of steering knuckle arm

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

Fig. 4: Material Properties

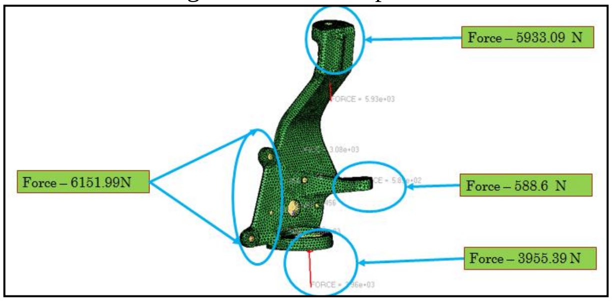

Fig. 5: Load and boundary condition applied to geometry

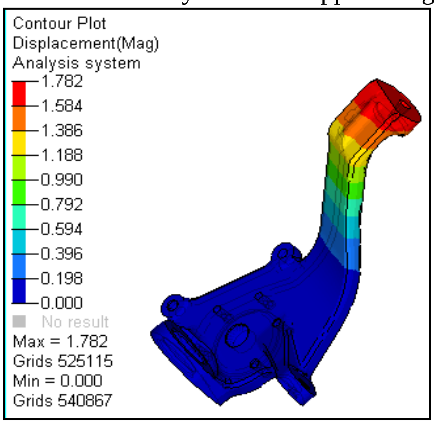

Fig. 6: Displacement Plot

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.

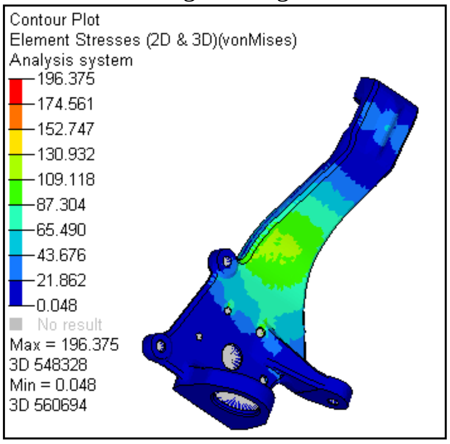

Fig. 7: Stress Plot

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.

V. Results & Discussion

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. In color contour, lower range of displacement and stress is shown in blue color while higher range is shown in red color.

A. *Experimental Setup*

Figure 8 shows the typical 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. 8: Typical test setup for physical experimentation to calculate stiffness

VI. Conclusion

Steering knuckle is analyzed under various loads from connected parts. Max Displacement Observed 1.782 mm. Von-Misses stress observed 196.38 MPa. Stresses are below yield limit (Yield strength for Forged Steel SAE Grade 11v37 > 556 MPa).

Keeping above results as a reference we have to optimize the steering knuckle arm so that stress & deflection should not be exceed more than specified deflection & von misses stresses observed in current steering knuckle arm. Also we need to check feasibility for manufacturing of optimized steering knuckle arm.

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

