Design And Experimental Evaluation of Steering Knuckle Arm For Stiffness

Deepika Pujari[1], R. N. Yerrawar[2]
[1](M.E Mech. Design Department, MESCOE, SPPU, India)
[2](Professor in M.E Mech. Design Department, MESCOE, SPPU, India)
Corresponding Author: Deepika Pujari1

Abstract: *The scope of this dissertation shall undertake the structural loads of the `linear type' that could be critical for failure of the component. The evaluation of such loads and the influence of the same over the intended structural strength of the component in avoiding premature failure shall be undertaken for study and analysis. CAE Software tools like HyperMesh for pre-processing and RadioSS or Optistruct for Structural Analysis of the component shall be deployed during the `Analysis Phase' of the work. Linear loads shall be applied during the experimentation and the corresponding displacement shall be recorded for evaluation. Comparison of the same vis-a-vis the analytical or numerical results should address the scope of validation of the work.*
Index Terms*: Sports Utility Vehicle, Steering Knuckle Arm*

I. Introduction

In automotive steering system knuckle is that part which contains a wheel spindle or hub. It attaches to suspension components and is called as a steering knuckle or spindle or upright/ hub. The tire and wheel assembly are attached to the hub or spindle of the steering knuckle. The tire/ wheel rotate while they are being held in a stable plane of motion by the suspension or knuckle assembly. The arm of the steering knuckle comes out to which the steering mechanism attaches so that it can turn the wheel and knuckle assembly. The cyclic nature forces are exerted on the assembly when the steering arm is turned to man-oeuvre the vehicle to the right, left and centre. The vehicle weight goes on increasing as safety and luxurious features get added. This increasing vehicle weight affects fuel efficiency and also overall vehicle performance. Therefore, reduction of the vehicle weight is real need in today's automotive industry.

II. Problem Statement/ Definition

The structural integrity of the component needs to be evaluated in the light of mass reduction while also maintaining the performance as close to the existing as possible. On one hand the weight of the vehicle is attempted for reduction while on the other, the components being mass optimized should be able to perform as desired without premature failure. The component - Steering Knuckle Arm Fig. 1 is exposed to varying loads during its life time before failure.

III. Summary Of Literature Review

From the review we can summarize that, several vehicle platforms and power train configurations were considered to assess the sensitivity of fuel economy to mass variation. Both kinematic and dynamic investigation of the designed suspension system is performed. Failure analysis of the automobile has been studied [1]. Some aimed to assess fatigue life and compare fatigue performance of steering knuckles made from different materials of different manufacturing processes [2]. Authors have studied and suggested an alternate material in place of existing. Blend of optimizations have also been suggested. The process of using Topology optimization for load path generation & Parametric study using shape optimization, reduces the design iteration and intermediate concept models and there by reduces the design cycle time [3]. Structural Analysis is displayed to make an inventive design and aimed to achieve substantial reduction in weight. Some have discussed an integrated design and manufacturing approach that supports the shape optimization [4]. Durability assessment is performed to establish the damage applied by loads. Its becomes important that; On one hand the weight of the vehicle is attempted for reduction while on the other, the components being mass optimized should be able to perform as desired without premature failure.

IV. Methodology

1) Design
We shall arrive at a suitable Design for the existing component that would effectively address the nature of loads being considered for this work.

2) Evaluation
Linear loads shall be applied during the experimentation and the corresponding displacement shall be recorded for evaluation.

3) Result
The results will be obtained from FEA and Experimentation. By comparing the results obtained from both we will be validating component stiffness.

Fig 1. Shows the Methodology - Flowchart for FEA
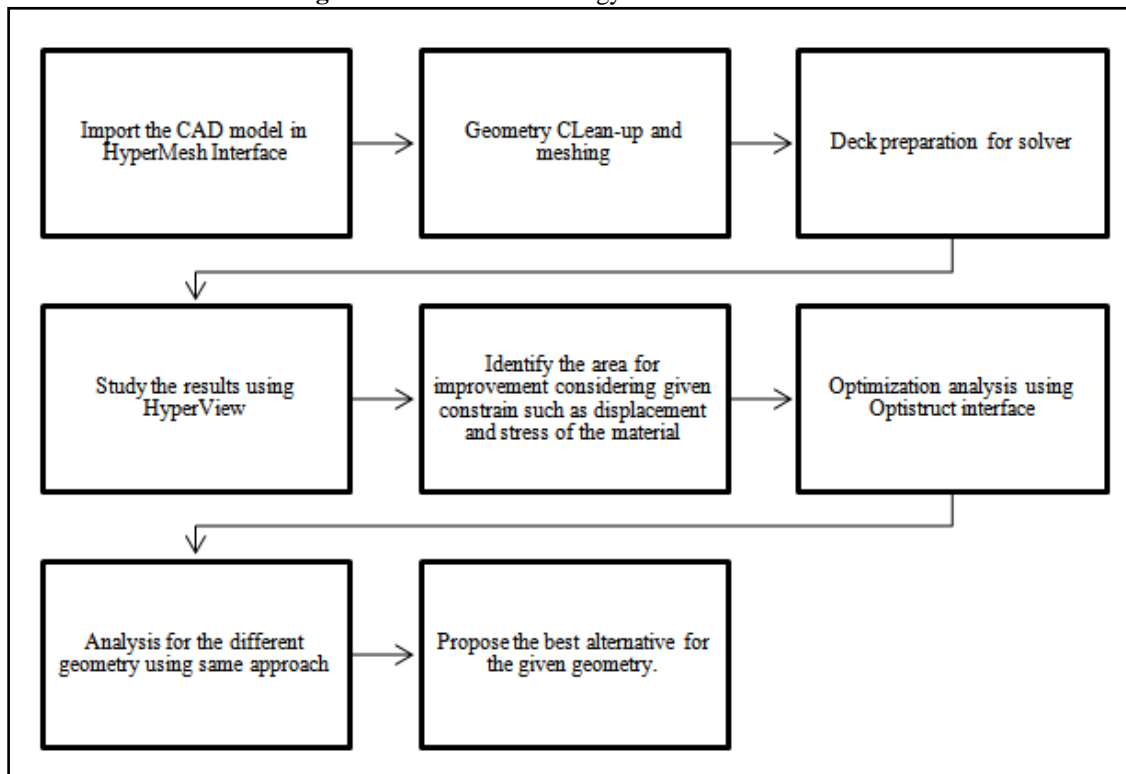
Fig 1. Methodology - Flowchart for FEA

V. Analysis Of Existing Steering Knuckle

The steering knuckle needs to be rigid and strong as it is subjected to power thrust. Vehicle weight decides the size and shape of knuckle component as the vertical load directly acts on it.

1. Modeling of the Steering Knuckle
The modelling is finished utilizing SolidWorks with the assistance of standard measurements that we collected from survey. SolidWorks is one of the modelling software accessible in the market which helps us not only to do the modelling of the components additionally analysis of the same.

2. Interface for Pre-Processing
Altair HyperMesh is a market-driving, multi-disciplinary finite element pre-processor which deals with the age of the biggest, most complex models, beginning with the import of CAD geometry to trading prepared to-run solver file. Hyper Mesh can be utilized to create a tetra mesh or a brick mesh. Import the CAD Model in Hypermesh Interface. Fig 2. Shows the HyperMesh – Interface with imported CAD Model for pre-processing
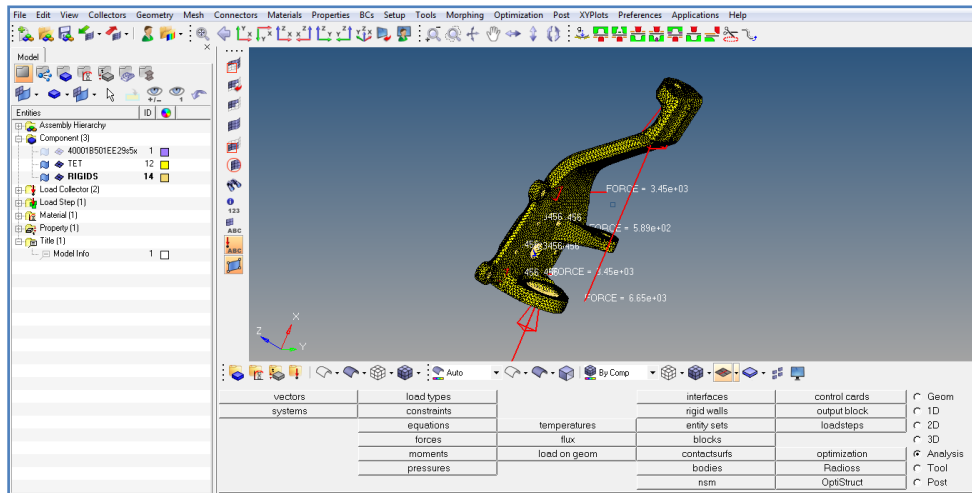
Fig 2. HyperMesh – Interface with imported CAD Model for pre-processing

3.   FE Entities
Mesh Model –
Element type   = Tetra second order
Element size   =4mm
No of elements   = 47214
No of Nodes   = 77627
Pre Processing   = HyperMesh 11.0
Solver   = Optistruct
Post Processing   = HyperView
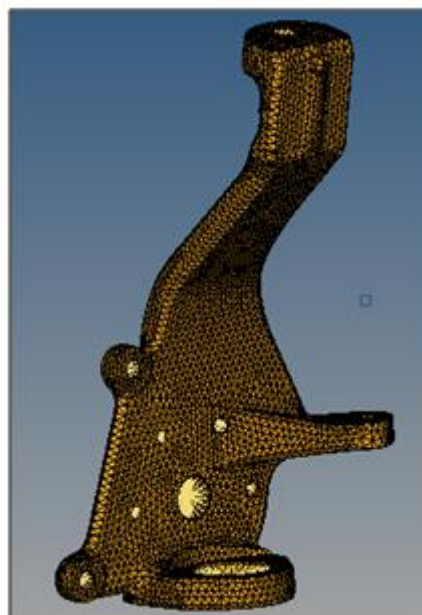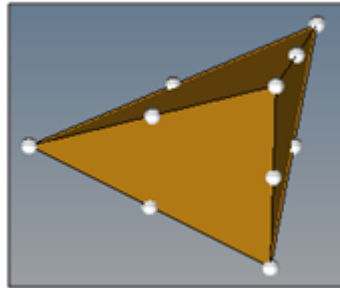Analysis   = Linear Static

Fig 3. Meshing of Steering Knuckle

Fig 4. Tetra element

Fig 5. FE Entities and Settings

FE Entities and Settings for existing model are shown in Fig 3, 4, 5.

4. Material Properties
Material- Forged Steel AISI Grade 4140; E=2.1x10^5N/mm2
Poisson Ratio=0.3, Density=7.89x10^-9 ton/mm3
Material properties settings have to be done as shown in Fig 6.
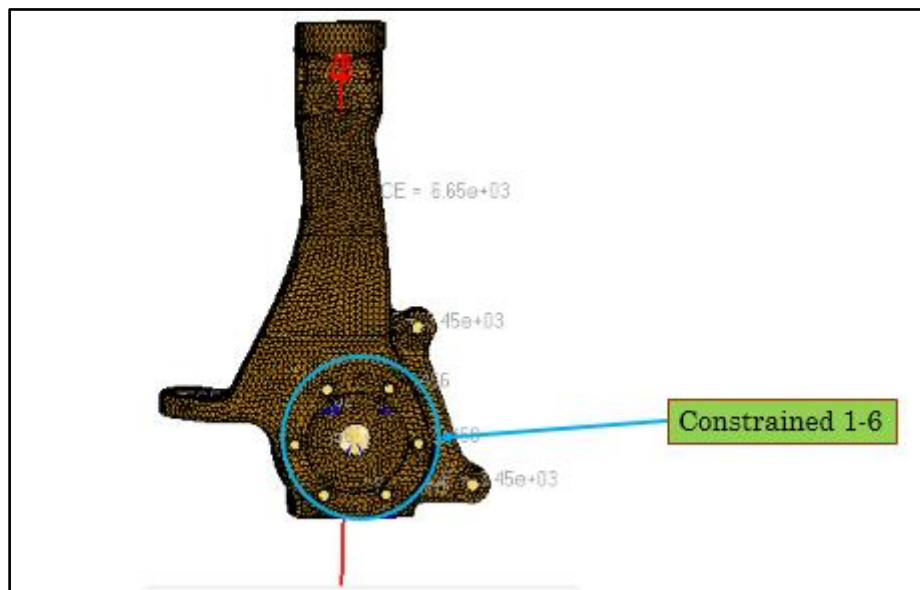

Fig 6. Material properties settings

5. Boundary Conditions
The boundary condition is the application of a force and/or constraint. In HyperMesh, boundary conditions are within what are called load collectors.
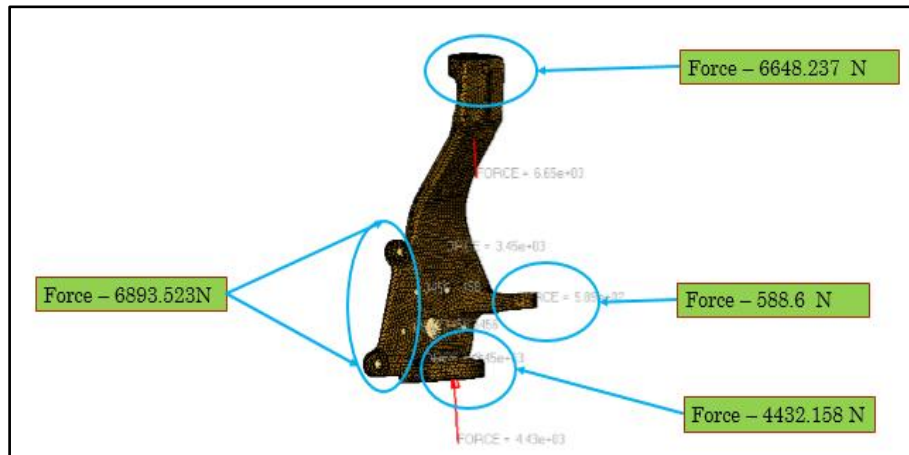
(a)   Applied forces

(b)   Applied constraint (all 6 dof locked)

Fig 7. Boundary conditions (Variant 1)

The calculated values of load need to be applied for boundary conditions as shown in Fig 7. Load collectors might be made utilizing the right click setting menu in the Model Browser (Create > Load Collector). Regularly (particularly at the beginning) a load collector is needed for the constraints (Single Point Constraints) and a second one is required for the forces or pressures. Remember, you can place any constraints (e.g. nodes constraint) with respect to dof 1, or nodes with constraints dof 123, etc. in a single load collector. The same rule applies for forces/pressures. They are stored within a single load collector regardless of their orientation and magnitude.

6. Hyper View Result Viewer

HyperView enables to visualize data interactively as well as capture and standardize the post-processing activities using process automation features as shown in Fig 8.
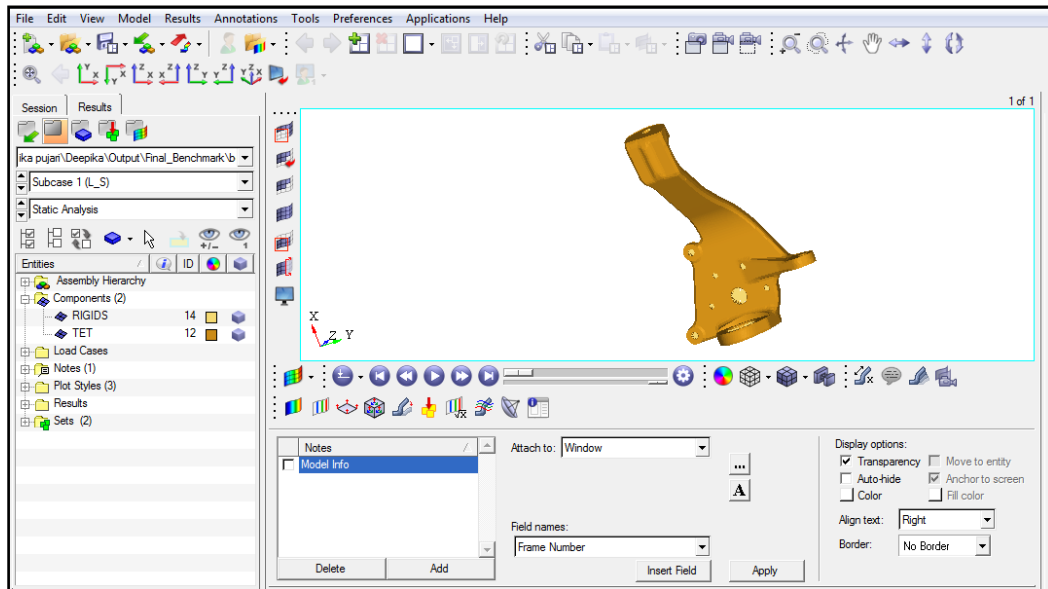

Fig 8. HyperView result viewer

VI. Concept Generation Of A Variant Knuckle

A design space is the initial part from which material is removed until a final shape is reached during optimization. Any part that is a design space will be reshaped during optimization, while any part that is not a design space will remain as it is. There can be any number of design spaces in the model.
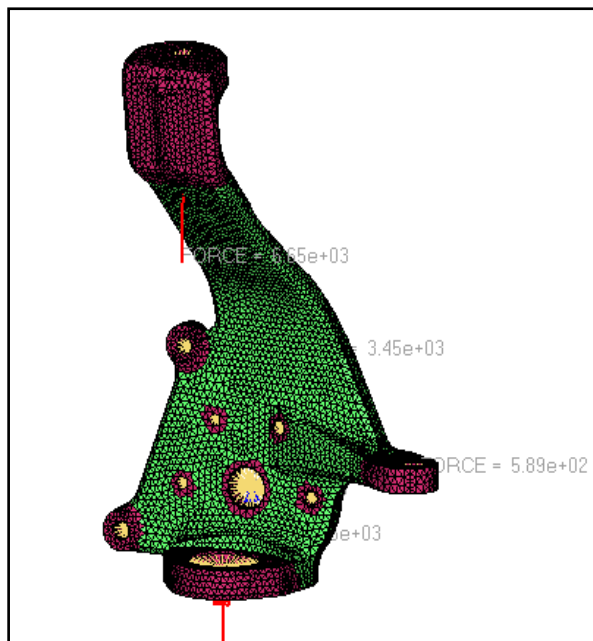

Fig 9. Design and non-design space of knuckle

A design space can have any shape or topology as long as it is a solitary strong volume. A section that is utilized as a design space ought not be extremely detailed. To guarantee the most flexibility to create a shape, utilization of the least complex design space ought to be made. The more fine elements in the design space, the more it will take to run the optimization. The green colour portion of the component as shown in Fig 9. is a design space from which the material is supposed to be removed. The purple colour portion is non-design space from which the material should not be removed as they are critical region. The design variable, objective, constraints and responses must be defined as shown in Fig 10, 11, 12 and 13 respectively.
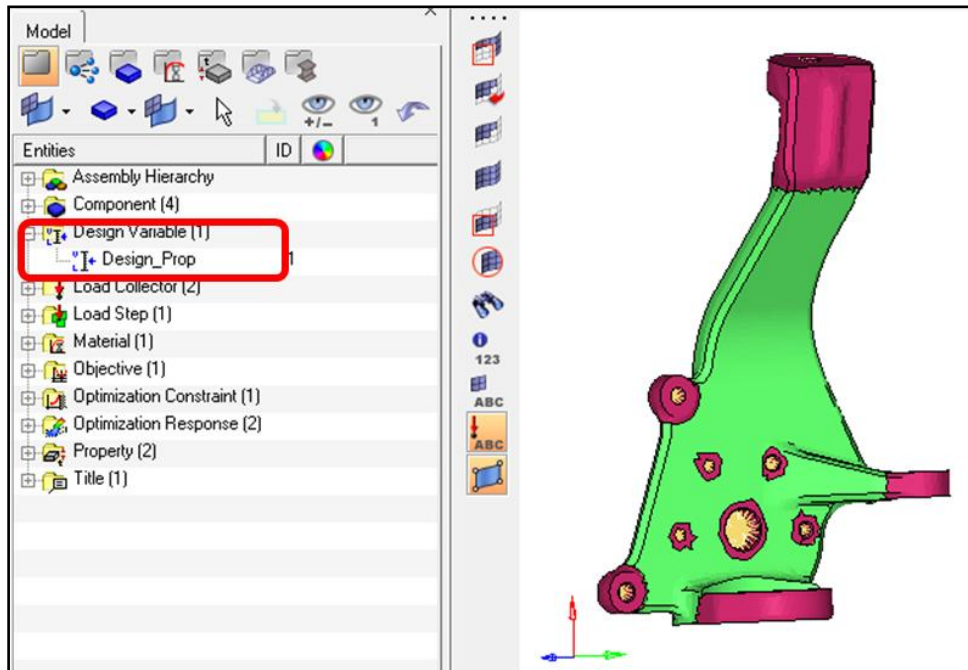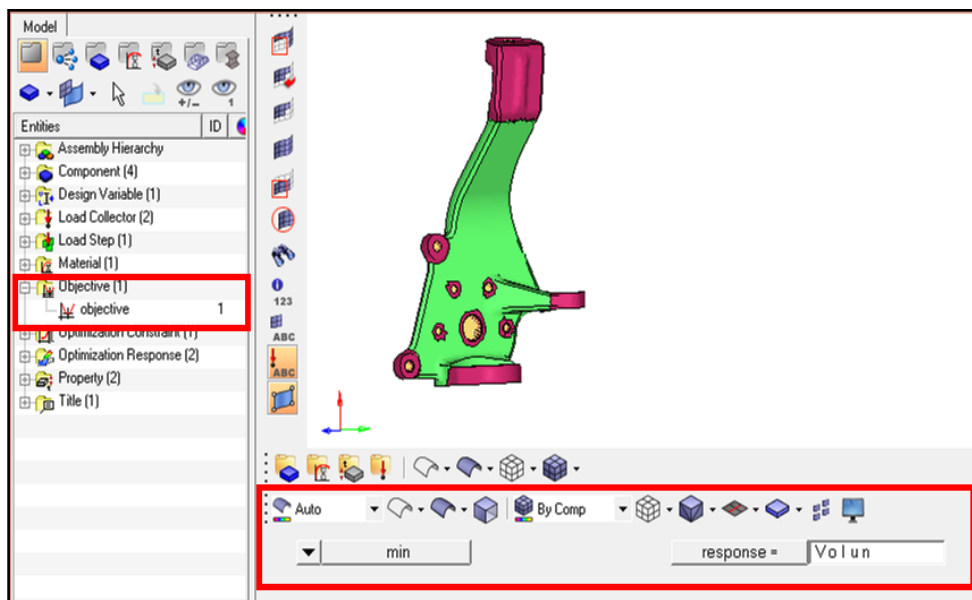
Fig 10. Defining design variable
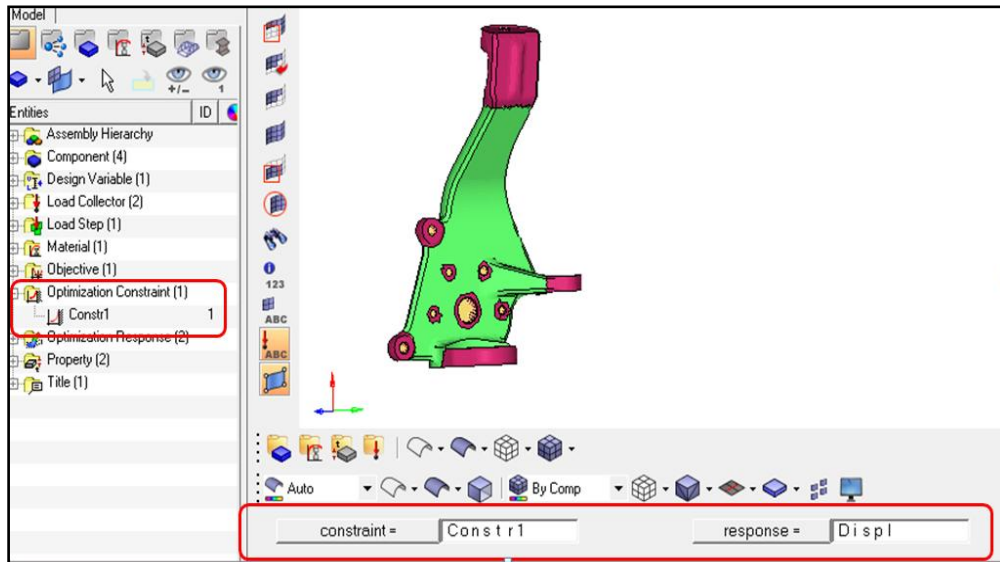

Fig 11. Defining objective

Fig 12. Defining optimization constraint on displacement response
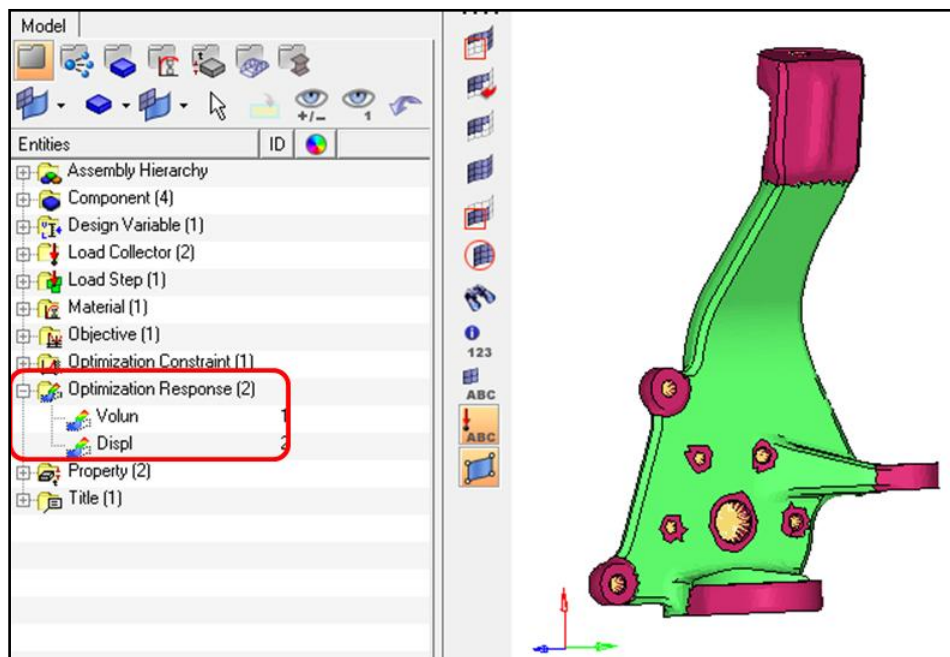

Fig 13. Defining optimization responses

VII. Analysis And Results For Existing Knuckle

The analysis is done for the existing knuckle and stress, displacement plots are taken.

A. Stress Plot:

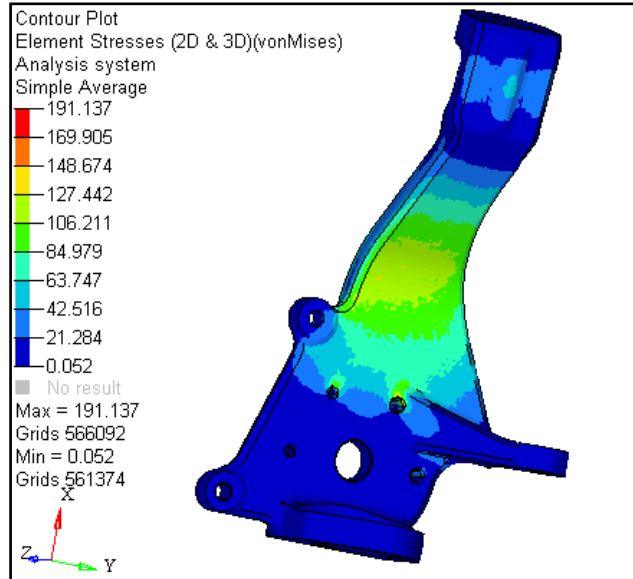From Fig. 14 the value of maximum stress is observed as, 191.137Mpa

Fig 14. Stress induced =191.137 N/mm2

B.   Displacement value:
From Fig 15. it is observed that the value of maximum displacement is 1.996mm.
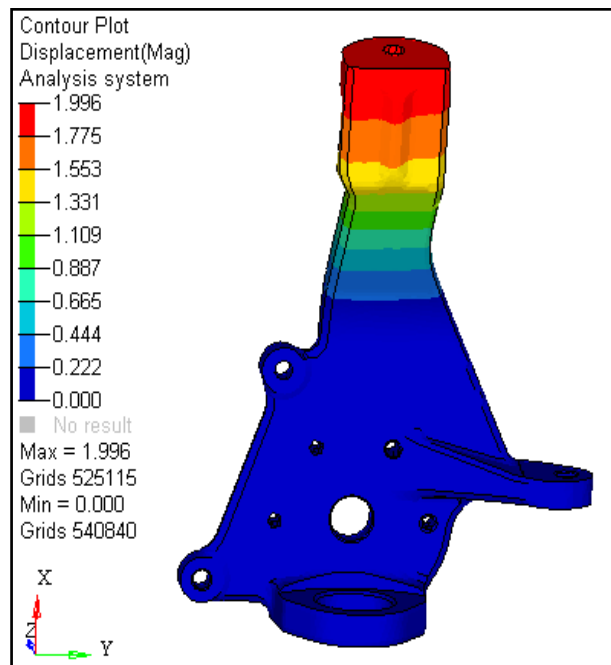

Fig 15. Displacement value =1.996 mm

VIII.   Conclusion For Existing Knuckle
It is necessary that the stresses should be below yield limit (Yield strength for Forged Steel AISI Grade 4140 > (415) Mpa). The Parameters of the knuckle in Table 1 can be put up as,

Table 1 Parameters for Existing Knuckle

Variant No.	Displacement (mm)	Von-Mises Stress (MPa)	Mass (Kg)	% Mass reduction
1.	1.99	191.14	7.898	0%

The modified design for the part is to be safe upon performing structural analysis for the new variant. The part has to become lighter and cheaper while contributing to the environmental concern for fuel efficiency of the

vehicle and lower processing cost for the part itself. Topology Optimization should be thought of as it offer crucial inputs along with the Structural Analysis for the part.

Fig 16. Show the original photograph of the Existing model.

Fig 16. Existing or Variant 1

References

[1]. Mohd Viqaruddin, D.Ramana Reddy, Structural optimization of control arm for weight reduction and improved performance, Materials for Manufacturing (ICAAMM-2016), Materials Today: Proceedings 4 (2017), *pp. 9230–9236*

[2]. Prof R. L. Jhala, K. D. Kothari and Dr. S.S. Khandare "Component Fatigue Behaviors And Life Predictions Of A Steering Knuckle Using Finite Element Analysis" International Multi Conference of Engineers and Computer Scientists 2009 Vol II

[3]. Rajeev Sakunthala Rajendran, Subash Sudalaimuthu and Mohamed Sixth "Knuckle Development Process with the Help of Optimization Techniques" Altair Technology Conference, India (2013)

[4]. S.Vijayarangan, N. Rajamanickam and V. Sivananth "Evaluation of metal matrix composite to replace spheroidal graphite iron for a critical component, steering knuckle" Materials and Design 43 (2013), pp. 532–541

[5]. K. Vikranth Reddy, Madhu Kodati, Kishen Chatra, Sandipan Bandyopadhyay, A comprehensive kinematic analysis of the double wishbone and MacPherson strut suspension systems, Mechanism and Machine Theory 105 (2016), *pp. 441–470*

[6]. Y. Samant Saurabh et al., Design of Suspension System for Formula Student Race Car, Procedia Engineering 144 (2016), *pp. 1138 – 1149*

[7]. Balázs Németh et al., Independent wheel steering control design based on variable-geometry suspension, IFAC-PapersOnLine 49-11 (2016), *pp. 426–431*

[8]. H. Masching, K.-U. Bletzinger, Parameter free structural optimization applied to the shape optimization of smart structures, Finite Elements in Analysis and Design 111 (2016), *pp. 33-45*

[9]. K. Baynal, M. Makaraci And K. Gulbudak, "Solution for Failure Analysis of Automotive Axle Knuckle Pull-Out", International Journal of Automotive Technology, Vol. 11, No. 5, *pp. 701−710 (2010)*

[10]. L. Harzheim, G. Graf, A review of optimization of cast parts using topology optimization, II-Topology optimization with manufacturing constraints Struct Multidisc Optim (2006) 31, *pp. 388–399*

[11]. Mahesh P. Sharma, Denish S. Mevawala, Harsh Joshi, Devendra A. Patel, Static Analysis of Steering Knuckle and Its Shape Optimization

[12]. Purushottam Dumbre, A.K.Mishra, 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, *pp. 552-557*

[13]. M. Jimenez, J. Martinez, U. Figueroa And L. Altamirano, "Estimated S-N Curve for Nodular Cast Iron: A Steering Knuckle Case Study", International Journal of Automotive Technology, Vol. 15, No. 7, *pp. 1197−1204 (2014)*

[14]. S. Pagerit, P. Sharer, A. Rousseau "Fuel Economy Sensitivity to Vehicle Mass for Advanced Vehicle Powertrains", *2006-01-0665*

[15]. L.J.H. Schulze, D. Goldstein, A. Patel, E. Stanton and J. Woods, "Torque Production Using Handwheels of Different Size During a Simulated Valve Operation Task" International Journal Of Occupational Safety And Ergonomics 1997, Vol. 3, No. 3^1, 109-118