Impact Factor (SJIF): 5.301

International Journal of Advance Research in Engineering, Science & Technology

e-ISSN: 2393-9877, p-ISSN: 2394-2444

Volume 6, Issue 11, November-2019

Design, Analysis and Optimization of ATV's Front Steering Knuckle

Murtaza Betma Wala[1], Mohammed Kanchwala[2], Vivek Jha[3], Prof. Shashank Singh Pawar[4]

[1]Mechanical Engineering, CDGI Indore
[2]Mechanical Engineering, CDGI Indore
[3]Mechnanical Engineering, CDGI Indore
[4]Mechnanical Engineering, CDGI Indore

Abstract — ***The steering knuckle is one of the most important components used in automobiles to turn the front wheel and supporting vertical weight of vehicle. In the current research steering knuckle used in BAJA ATV vehicle is analyzed using ANSYS FEA software under static loading conditions of maximum impact. The stresses, deformation and factor of safety are obtained from analysis. Subsequent analysis is conducted to determine the effect of reduced thickness of knuckle on stress, deformation and factor of safety. The findings have shown that 20% weight reduction is possible by reducing thickness of knuckle without much compromise in strength and safety factor. The CAD model of knuckle is developed using CATIA V5 design software.***

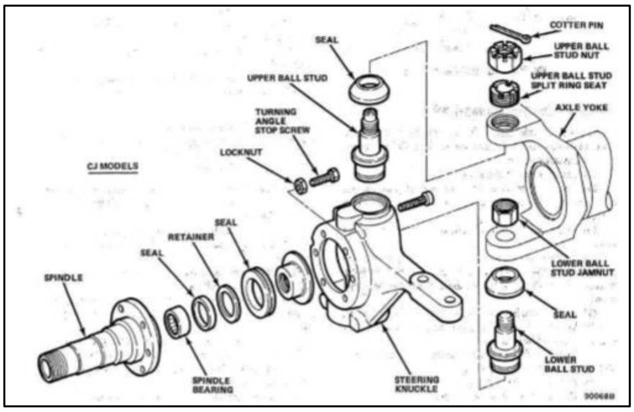
Keywords-FEA, ANSYS, Steering knuckle, weight reduction

I. INTRODUCTION

Steering knuckle is a prominent component in car which takes the loads from the wheels and transfers these forces to the suspension system. Structural Components such as a steering knuckle is generally strong component which is capable to withstand if load is but it can fail under fatigue and dynamic load. Depending on the vehicle and suspension design, the steering hub or spindle will also vary slightly. In the design variables of vehicles like all wheel drive and front wheel drive; knuckle joint will be point at which steering is connected.

a>Knuckle joint with Spindle

b> Knuckle joint attached to wheel



c> Components of knuckle joint

Figure 1: a>Knuckle joint with spindle b> Knuckle joint attached to wheel c> components of knuckle joint

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. Steering knuckles come in all shapes and sizes. Their designs differ to fit all sorts of applications and suspension types. However, they can be divided into two main types. One comes with a hub and the other comes with a spindle.

II. LITERATURE REVIEW

Rajkumar Roy et. al. [1] has investigated on optimization techniques which are manual and contemplates on scalability. The author also discussed about challenges for design optimization and also throws light on popular optimization approach known as GA.

S. Vijayaranganet.al. [2] has investigated application of MMC's (Material Matrix Composites) for optimization of steering knuckle as substitute to SG iron. The analysis was conducted using Al-10% TiC using ANSYS software. The findings have shown the maximum stress region of knuckle at strut. The MMC material are able to reduce material of knuckle by 55% as compared to conventional SG iron.

Prof. R. L. Jhala et. al. [3] has analyzed knuckles made from forged steel, cast aluminium and cast iron to determine fatigue life and stresses using finite element analysis. The findings have shown that knuckle made from forged steel has better fatigue life and lower stresses as compared to aluminium and cast iron.

K. H. Chang and P.S. Tang [4] has contemplated on shape optimization using integrated and manufacturing approach for knuckle. The manufacturing cost is the major factor for design optimization of knuckle. The primary objective of optimization should undertake manufacturing cost as objective or constraint.

Rajeev Sakunthala Rajendran et. al. [5] has used topological optimization on knuckle for weight minimization. The optimization is done on design space to derive optimal load path for the load cases. The shape variable is developed using hypermorph and optimization is done using hyper study. The design cycle time is reduced using shape optimization and with the use of intermediate concept models.

III. PROBLEM DEFINITION

The manufacturing cost of steering knuckle depends upon the type and quantity of material used. The current research investigates the stresses generated in steering knuckle along with determination of possible zones for material minimization. The CAD model of steering knuckle, spindle and steering assembly is developed in CATIA V5 software and structural analysis is conducted using ANSYS software. The new designs of knuckle is presented with reduced thickness and subjected to structural analysis.

IV. METHODOLOGY

The analysis involves three stages of analysis which are preprocessing, solution and postprocessing. The preprocessing stage involves CAD modelling. The CAD model is developed using CATIA V5 software. The model is developed using extrude, revolve, chamfer and fillet tools. The CAD model of knuckle with different thickness is developed as shown in figures below.

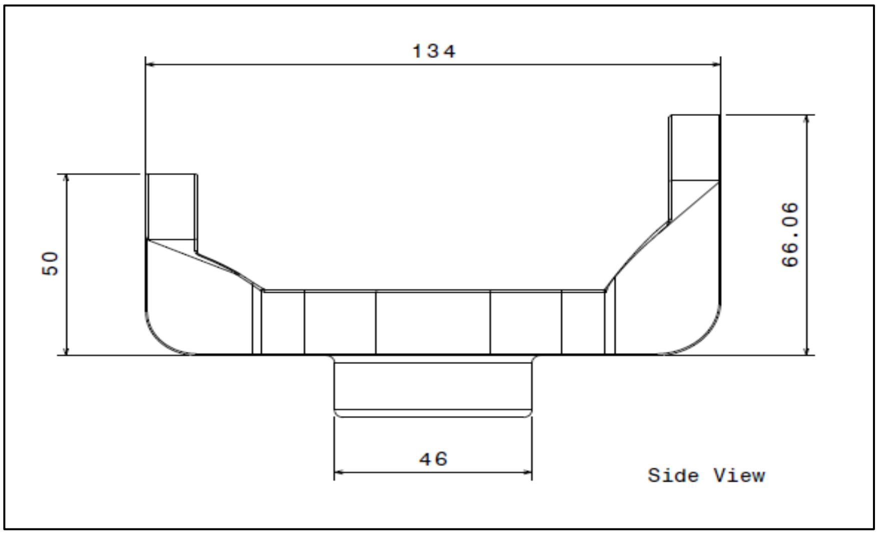


Figure 2: Side view of knuckle

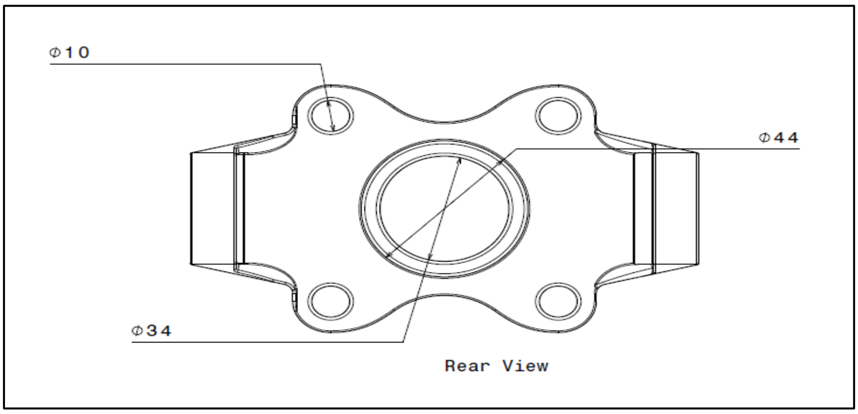


Figure 3: Rear view of knuckle

Figure 2 and figure 3 above shows side view and rear view of knuckle. The CAD model of stub developed is shown in figure 4 and figure 5 below. The geometry has helical cut feature at leftmost portion.

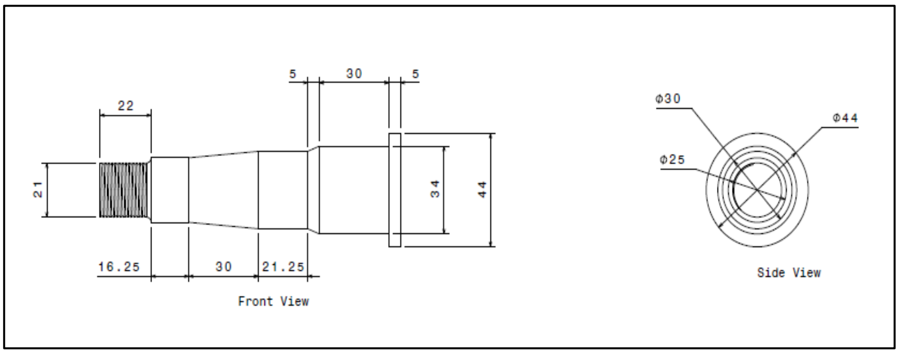
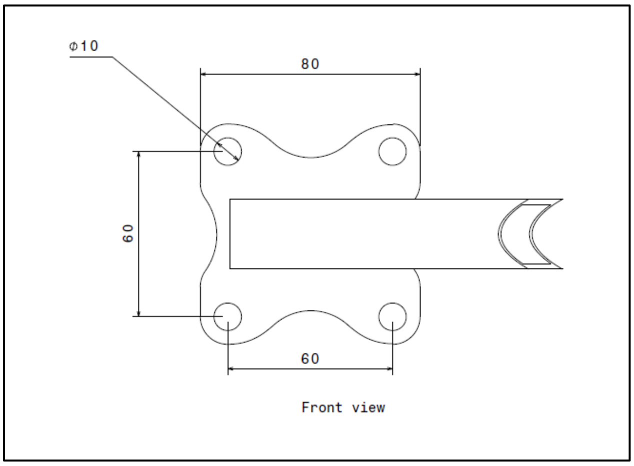
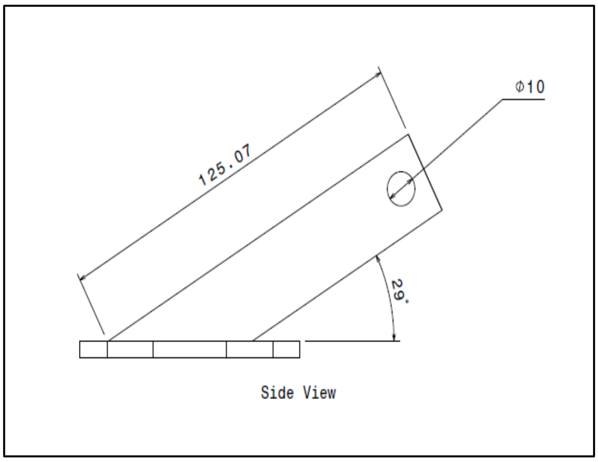


Figure 4: Front view and side view of stub

Similarly, CAD model of arm is developed using sweep, sketch and extrude tool as shown in figure 5 below.



a. Front view of arm



b. Side view of arm

Figure 5: Front and side view of arm

After CAD model is developed the next stage involves material definition and assignment. The material used for analysis is Aluminium 7075 T6. The material properties are described in table below.

Table 1: Material properties of Aluminium 7075 T6

Density (Kg/m^3)	Elasticity (MPa)	Poissons ratio
2810	72000	.33

The material properties as shown in table 1 above is fed to ANSYS material definition module. The CAD model is then imported in ANSYS design modeler where it is checked for any type of geometric errors like hard edges, inflation etc.

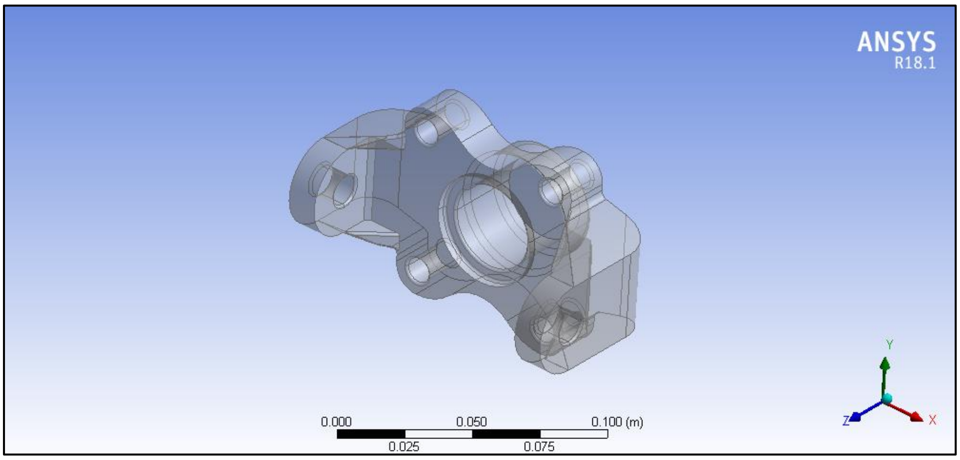


Figure 6: CAD model of knuckle imported in ANSYS workbench

After importing CAD model it is meshed using tetrahedral elements with fine sizing, transition fast, span angle center coarse, smoothing medium and transition ratio set to 0.272, growth rate 1.2. The number of elements generated is 61473 and number of nodes generated is 100657. The meshed model is shown in figure 7 below.

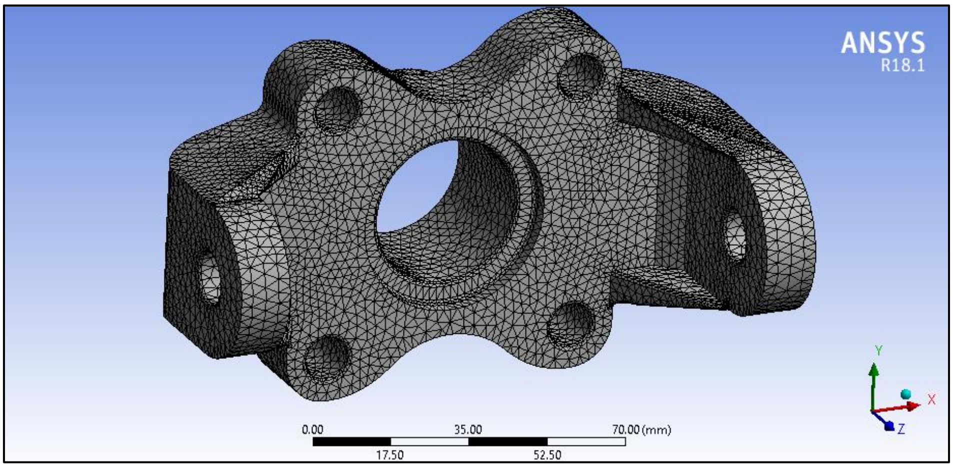


Figure 7: Meshed model of knuckle using tetrahedral elements

After meshing the model is applied with appropriate loads and boundary conditions.

The loading conditions are considered under 3G conditions as discussed below.

Curb weight of vehicle considered – 2100N
Weight of 1 person as per SAE rule – 1110N
Gross weight of vehicle – 3210 N

For analysis, we consider when vehicle is on jump condition when total weight of vehicle comes on spindle.

Therefore, load on spindle – 3210 N

Force on steering arm – 580N

Component of Force in X direction F_x – 1250N

Component of Force in Y direction F_y – 2500N

Component of Force in Z direction F_z– 3600N

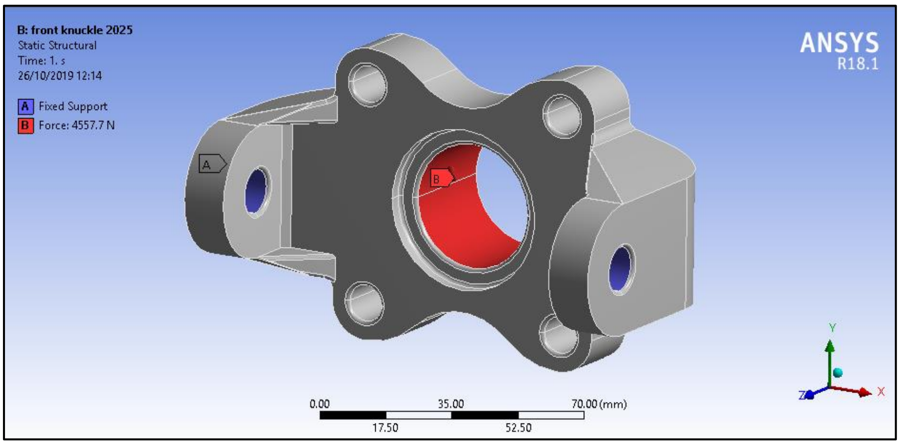


Figure 8: Loads and boundary conditions on knuckle

The loads and boundary conditions applied on knuckle is shown in figure 8 . The fixed support is applied on side faces of knuckle represented by blue color and force components is applied on center circular portion represented by red color.

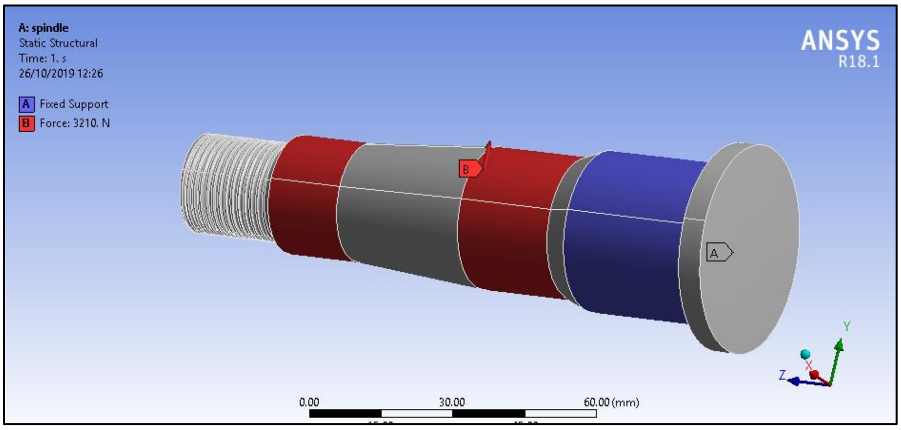


Figure 9: Loads and boundary conditions on spindle

The loads and boundary conditions applied on spindle is shown in figure 9. The fixed support is applied on side faces of knuckle represented by blue color and force components is applied on circular portion represented by red color with magnitude of 3210N.

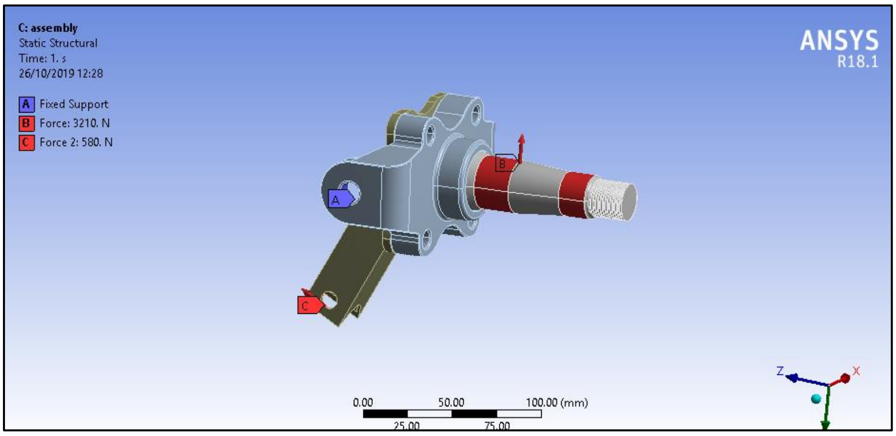


Figure 10: Loads and boundary conditions on steering assembly

The loads and boundary conditions applied on steering assembly is shown in figure 10 above. The fixed support is applied on side faces of knuckle represented by blue color and forces are applied on spindle surface and arm support depicted by red color. In solution stage, the software formulates element stiffness matrices for each element. The element stiffness matrix is assembled together to form global stiffness matrix. The next step involves matrix multiplication and inversions. The results are generated for each node. These results are then interpolated for entire element edge length as per linear or quadratic interpolation function.

V. RESULTS AND DISCUSSION

After conducting FEA structural analysis on knuckle, spindle and steering assembly the contours of equivalent stress and deformation are plotted. The results are discussed below for all components.

SPINDLE RESULTS

The material used for analysis of spindle is structural steel and contour plot of equivalent stress is shown in figure 11 below. The contour shows maximum equivalent stress of 72.822MPa on tapered zone shown by red and yellow colored contours. The stress decreases as we move away from tapered zone shown by light blue color with magnitude of stress ranging from 8.09MPa to 16.183MPa.

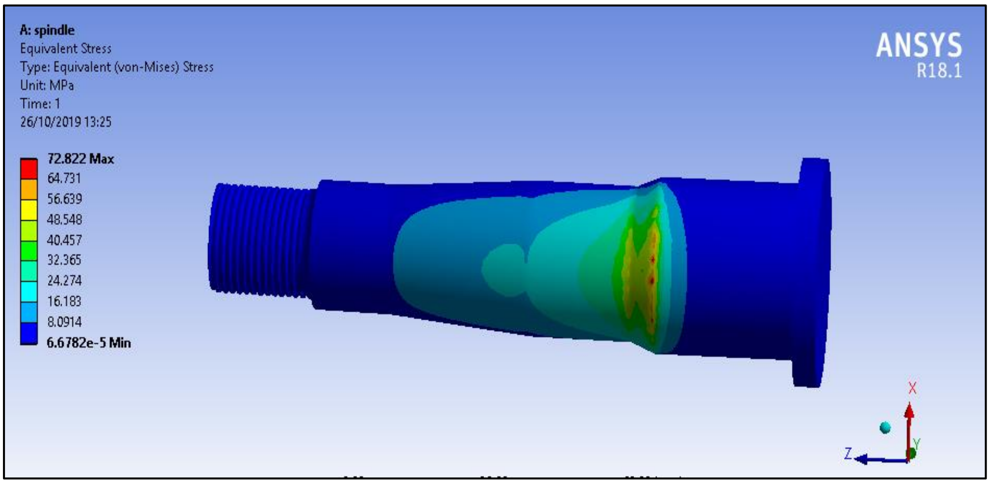


Figure 11: Equivalent stress at different regions of spindle

The total deformation plot is shown in figure 12 below. The maximum magnitude of deformation is .0329mm on left end portion with helical cut and reduces as we move towards the thicker end portion of spindle as shown in by light colors of green, yellow and blue.

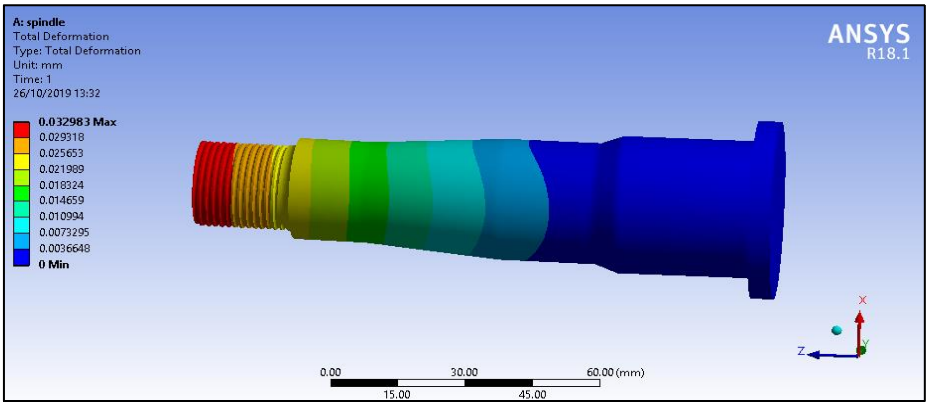


Figure 12: Total deformation at different regions of spindle

KNUCKLE RESULTS

From the structural analysis conducted for front knuckle component equivalent stress plot is generated. The maximum stress is generated near support feature and V shape junction with magnitude of 122.7MPa and reduces as we move away from this junction show by light blue and dark blue color contour.

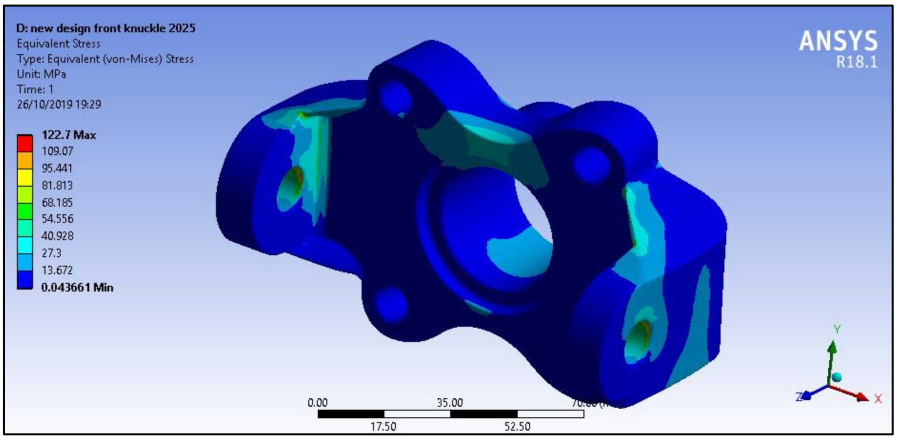


Figure 13: Equivalent stress at different regions of knuckle

The total deformation plot is shown in figure 14 below. The maximum magnitude of deformation is .0689mm on circular portion which is connected to arm and deformation reduces as we move away from circular zone shown by dark orange and green colors. The minimum deformation is noticed at support zones.

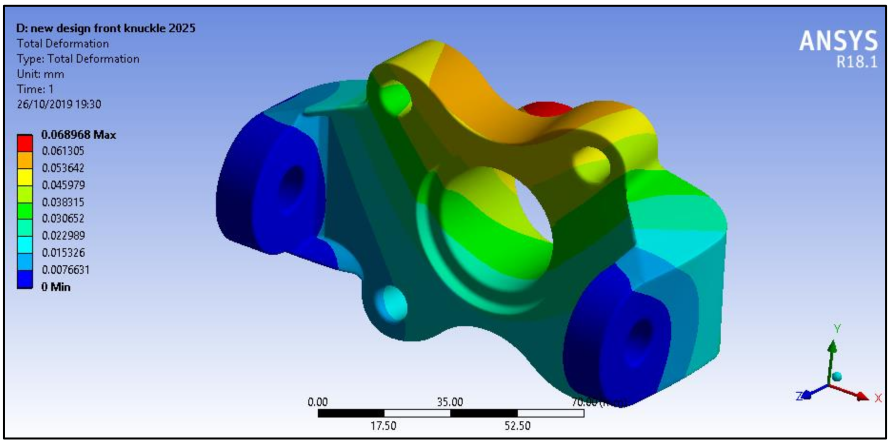


Figure 14: Deformation at different regions of knuckle

ASSEMBLY RESULTS

From the structural analysis conducted for front knuckle assembly equivalent stress plot is generated. The maximum stress is generated near support feature and V shape junction with magnitude of 158.76MPa and reduces as we move away from this junction show by light blue and dark blue color contour.

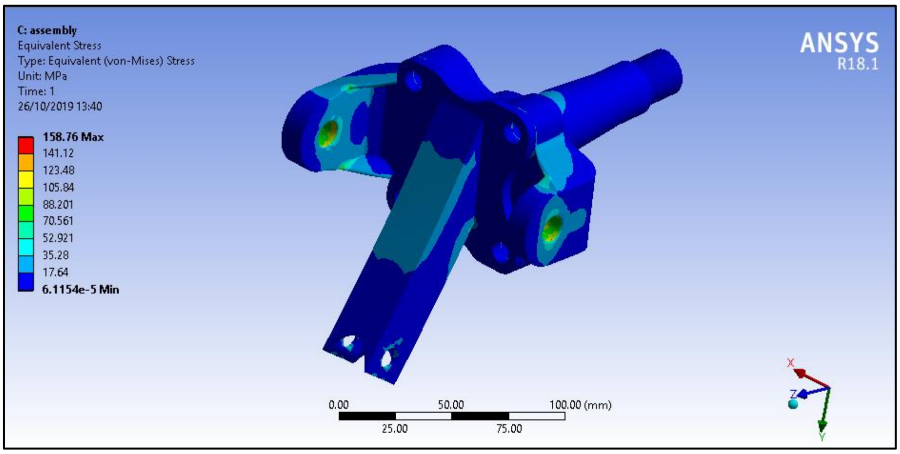


Figure 15: Equivalent stress at different regions of knuckle assembly

The total deformation plot of knuckle assembly is shown in figure 16 below. The maximum magnitude of deformation is .3201 mm on helical portion of spindle and deformation reduces as we move away from helical zone shown by dark orange and green colors. The minimum deformation is noticed at support zones of knuckle.

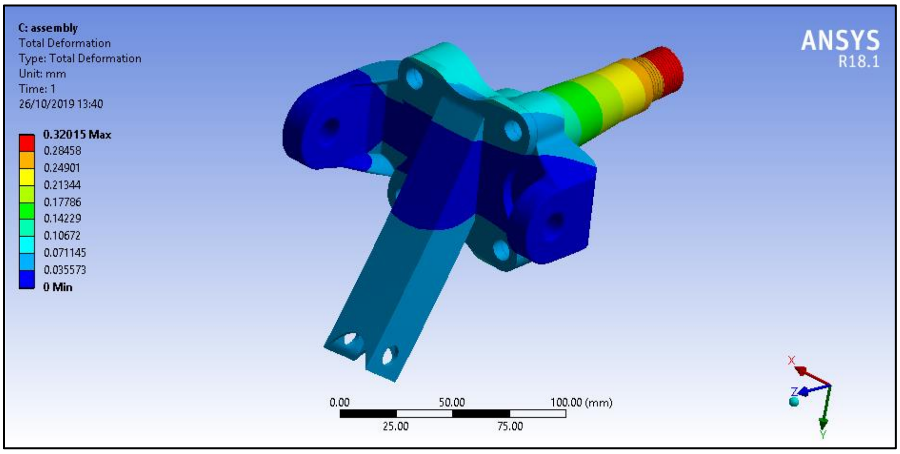


Figure 16: Deformation at different regions of knuckle assembly

Table 2: Results for different components

Component Name	Yield Strength (MPa)	Deformation(mm)	Eq Stress (MPa)	Factor of Safety
Spindle	250	.0329	72.82	3.43
Knuckle	510	.0653	122.7	4.15

Table 2 above shows safety factor and deformation for both components i.e. spindle and knuckle. The safety factor achieved for spindle is 3.43 and for knuckle is 4.15. Further analysis is conducted to reduce weight of knuckle and hence the manufacturing cost involved. The existing design of knuckle has mass of .4561kg for 18mm thickness and further analysis is conducted with reduced thickness of knuckle. The new thickness of knuckle is taken are 15mm and 11mm.

KNUCKLE THICKNESS 15MM

From the structural analysis conducted for front knuckle with 15mm thickness, equivalent stress plot is generated as shown in figure 17 below. The maximum stress is generated near support feature and V shape junction with magnitude of 135.84MPa and reduces as we move away from this junction show by light blue and dark blue color contour.

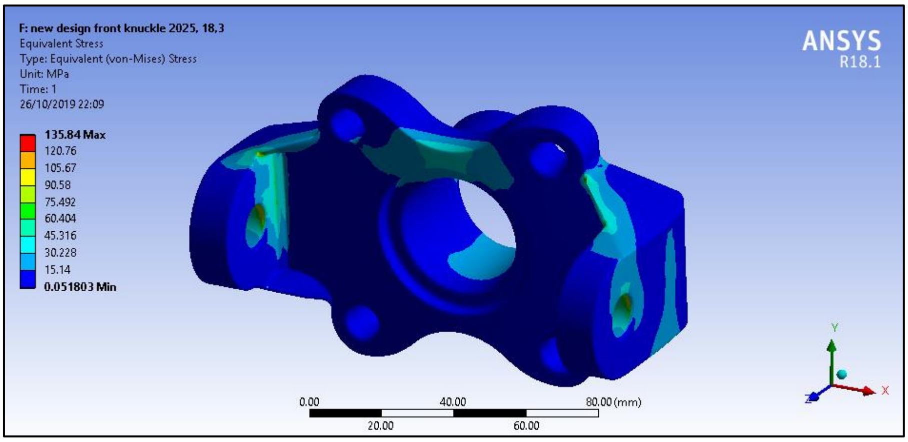


Figure 17: Equivalent stress with 15mm thickness

The total deformation plot is shown in figure 18 below for 15mm thickness. The maximum magnitude of deformation is .0799mm on circular portion which is connected to arm and deformation reduces as we move away from circular zone shown by dark orange and green colors. The minimum deformation is noticed at support zones

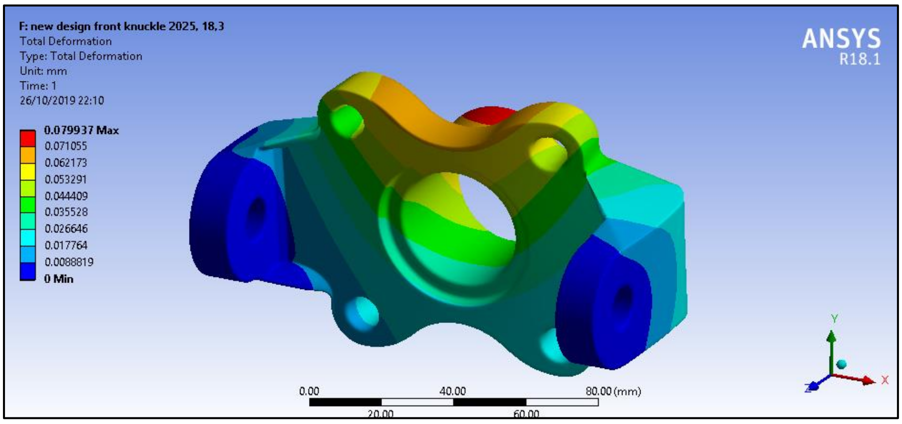


Figure 18: Deformation with 15mm thickness

KNUCKLE THICKNESS 11MM

From the structural analysis conducted for front knuckle with 11mm thickness, equivalent stress plot is generated as shown in figure 19 below. The maximum stress is generated near support feature and V shape junction with magnitude of 149.48MPa and reduces as we move away from this junction show by light blue and dark blue color contour.

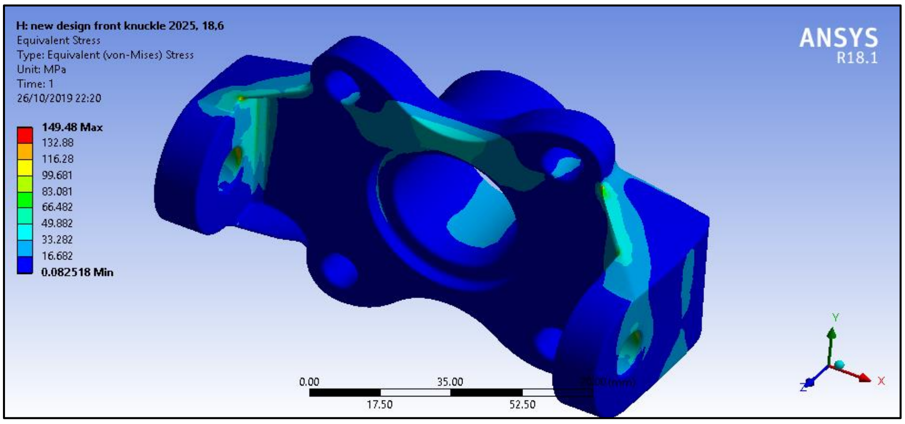


Figure 19: Equivalent stress with 11mm thickness

The total deformation plot is shown in figure 20 below for 11mm thickness. The maximum magnitude of deformation is .10016mm on circular portion which is connected to arm and deformation reduces as we move away from circular zone shown by dark orange and green colors. The minimum deformation is noticed at support zones

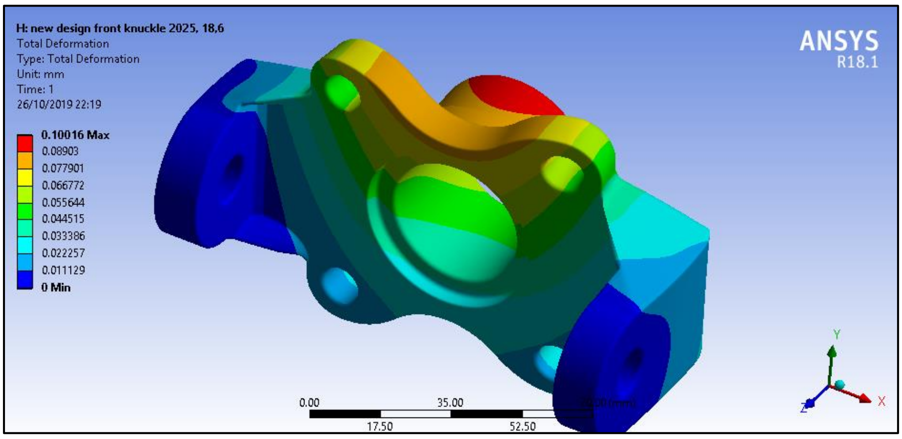


Figure 20: Equivalent stress with 11mm thickness

Table 3: Results for different thickness of Knuckle

Thickness(mm)	Equivalent stress (MPa)	Deformation (mm)	Mass(Kg)	FOS
18	122.7	.065	.450	4.15
15	135.84	.079	.405	3.75
11	149.48	.1001	.360	3.41

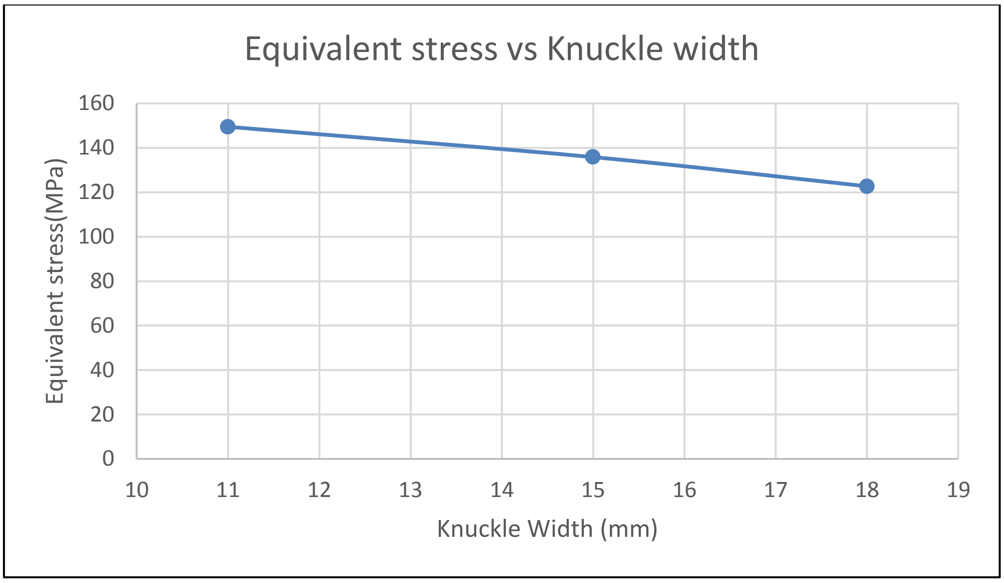


Figure 21: Equivalent stress vs knuckle width

The equivalent stress vs knuckle width graph shown in fig 21 above shows decrease in stress value as knuckle width increase. This is due to increased stiffness of knuckle. The mass of knuckle increases with increase in knuckle width as shown in figure 22 below and maximum mass obtained is .450kg for 18mm thickness of knuckle.

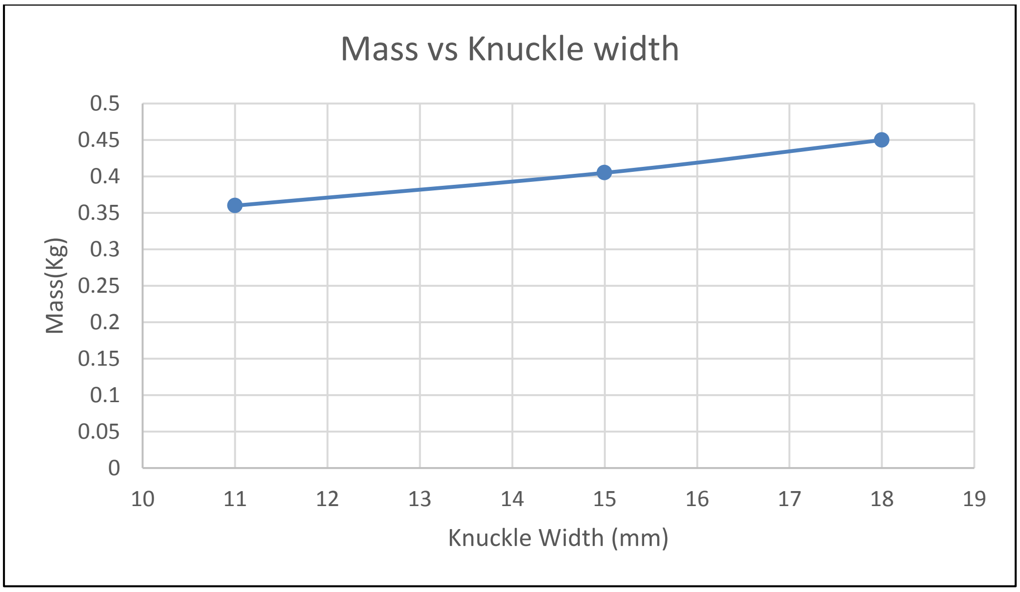


Figure 22: Mass vs knuckle width

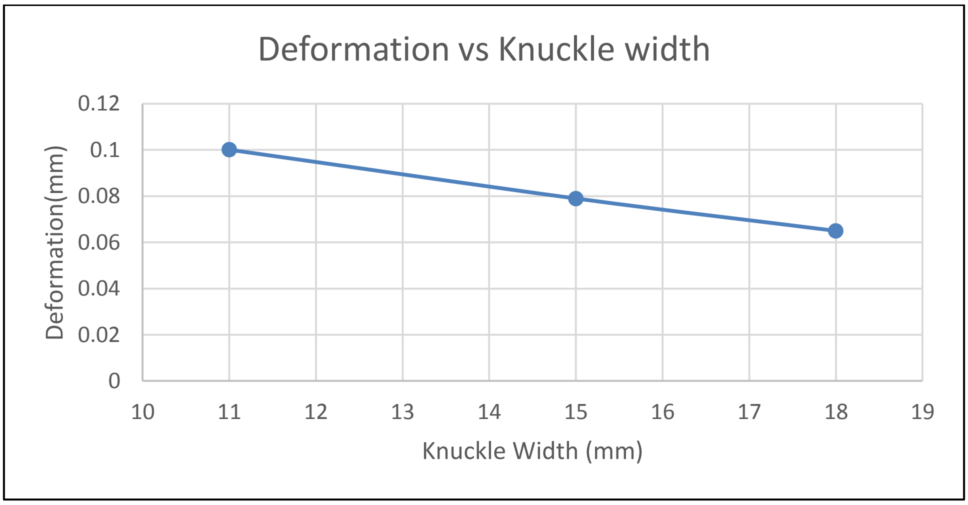


Figure 23: Deformation vs knuckle width

Figure 23 above shows decrease in deformation of knuckle with increase in knuckle width. The maximum deformation is observed for knuckle width of 11mm and minimum deformation is observed for knuckle width of 18mm. Factor of safety increases with increases in knuckle width as shown in figure 24 below. The minimum FOS is observed for knuckle width of 11mm and maximum FOS is observed for knuckle width of 18mm.

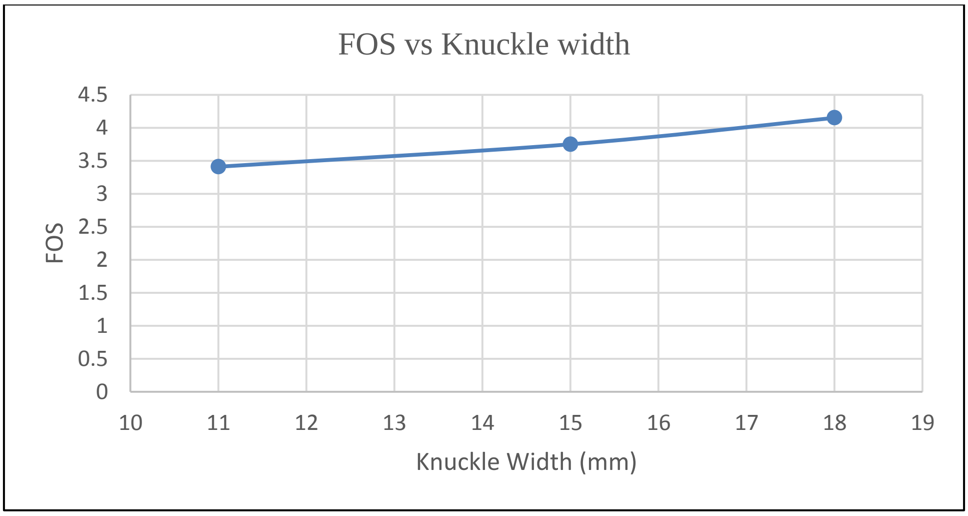
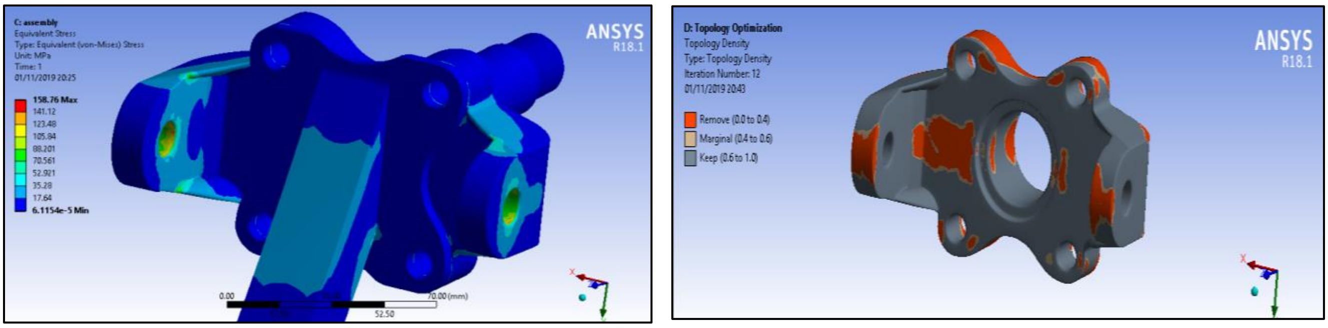


Figure 24: FOS vs Knuckle width

The findings have shown that mass reduction is possible by reducing width of knuckle without much compromise in safety factor of the component. The minimum safety factor is for 15mm width of knuckle and corresponding mass obtained is 3.75Kg. For lot production in bulk quantities a large mass of material can be saved and thus reduce can reduce manufacturing cost of the component also. Further analysis involved topological optimization which is a mathematical method that optimizes material layout within a given design space, for a given set of loads, boundary conditions and constraints with the goal of maximizing the performance of the system.



a>Equivalent stress b> Topological density

Figure 25: (a)Equivalent stress and (b)topological density

The topological optimization density plot shows good possibility of mass reduction at regions of low equivalent stress. The maximum material removal is observed at knuckle thickness as shown in figure 26 below. Similarly, at regions near supports for spindle and also near fixed support as shown by red contour, max material reduction is possible. While for the remaining areas the topological optimization shows to retain (grey colored zone).

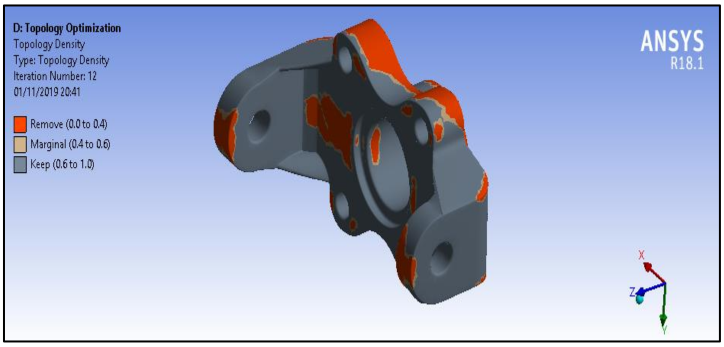


Figure 26: Maximum material removal at knuckle thickness area

VI. CONCLUSION

The structural analysis of knuckle assembly along with components involved like spindle is analyzed employing techniques of Finite Element Method using ANSYS software. On the basis of analysis equivalent stress, deformation and safety factor. The thickness of knuckle is optimized for weight reduction without compromising strength and safety factor of knuckle. The minimum mass achieved for is found to be .360 Kg and corresponding factor of safety is 3.41. The mass reduction achieved is .090kg/unit. Thus, for lot production good amount of material can be saved and this reducing manufacturing cost.

REFERENCES

[1]. RajkumarRoy, Srichand Hinduja and Roberto Teti “Recent advances in engineering design optimization: Challenges and future trends” CIRP Annals – Manufacturing Technology 57 (2008) 697–715

[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]. 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) 532– 541

[4]. K. S. Chang and P.S. Tang “Integration of Design And manufacturing of structural shape optimization” Advances in engineering software32 (2001) 555-567

[5]. Purushottam Dumbre, Prof A.K.Mishra, V.S.Aher, Swapnil S. Kulkarni, “Structural Analysis of steering Knuckle for Weight Reduction”8974April-June,2014, International Journal of Advanced Engineering Research and Studies EISSN2249–/ pp. 86-86.