International Journal of Advanced Research in Science Management and Technology
ISSN(Online)-2454-4159 Volume 3, Issue 2, February 2017

Experimental & Finite Element Analysis of an Automotive Steering Knuckle

Mr. Abhijit Chaudhari[1], Prof. A.V.Patil[2]
PG Scholar, Department of Mechanical Engg., SSGBCOET, Bhusawal, Maharashtra,India[1]
HOD, Department of Mechanical Engg., SSGBCOET, Bhusawal, Maharashtra, India[2]

Abstract— The heaviness of the vehicle is continuing expanding because of extra extravagant and security highlights. The expanding weight of the vehicle influences the fuel efficiency and general execution of the vehicle. Hence the weight decrease of the vehicle is thereal need of today's automotive industry. The steering knuckle on your vehicle is a joint that permits the steering arm to turn the front wheels. The powers applied on this get together are of cyclic nature as the steering arm is swung to move the vehicle to one side or to one side and to the middle once more. For the proposed work over the `steering knuckle arm', the goal is to recognize the nature and magnitude of stresses over the part while thinking about change in the plan. Contemplating static and element stack conditions, basic examination and modular investigation were performed. Limited component model was created in Hypermesh10. 10 hub tetrahedral components were utilized for lattice, giving better outcomes in less time. FEA/CAE software like Abacus solver should be locked in to reproduce and anticipate the level of stresses for the given info loads. Optistruct is utilized as improvement apparatus for discovering the anxiety dissemination, thickness dispersion over the segment. Subsequent to examining the outcome from optistruct the part is altered by expelling the material from low anxiety district. In the wake of playing out the examination on adjusted part it is watched that the anxiety and the removal are diminished. Consequently it is watched that the part is protected under given stacking conditions and stresses are beneath yield stretch. For approval of the venture test setup is to be composed which comprise of Universal testing machine for applying the stacking conditions, strain gage for measuring the uprooting and appropriate apparatus for holding the segment is utilized. In the wake of applying the stacking the anxiety and dislodging are to be recorded.

Keywords— Fuel Efficiency, Automotive Industry, Steering Knuckle Arm, Magnitude of Stresses, Hypermesh10, Universal Testing Machine.

I. Introduction

Automotive producers persistently grow new vehicles with more extravagance, accommodation, execution, and wellbeing. Often, they lessen a vehicle weight which brings about decreased vitality utilization. Auxiliary segments like suspension parts are oftenly thought about for streamlining on the grounds that their mass decrease (unsprung masses) impacts the auto solace and taking care of .Among the vehicle basic segments, the knuckle is one of the vital parts in the suspension framework. It assumes a urgent part in minimizing the vertical and move movement of the vehicle body, which suggests a poor traveler encounter, when a vehicle is driven on a harsh street. A knuckle segment is required to bolster the heap as well as torque incited by knocking, braking, and speeding up and furthermore helps in steering the tire.

Steering knuckle is the most imperative part in the vehicle yet not everybody realizes what this for steering knuckle is a manufacturing that typically incorporates the axle and steering arm, and permits the front wheel to rotate. The knuckle is mounted between the upper and lower rotating conjunctures on a SLA suspension, and between the strut and lower swiveling appendage on a MacPherson strut suspension. The steering knuckle on your vehicle is a joint that permits the steering arm to turn the front wheels. The powers applied on this get together are of cyclic nature as the steering arm is swung to move the vehicle to one side or to one side and to the inside once more. The anxiety examination is completed for the outrageous conditions, and the object is to research the auxiliary quality. In genuine utilize, the structure bears the boundless loads for the most part.

The knuckle exist three types of extreme conditions, namely:

1) The uneven ground conditions

2) Emergency braking conditions.

3) And sideslip conditions.

II. LITERATURE SURVEY

V. Sivananth et al.[1]conducted computer aided design of steering knuckle and the model was developed using modeling package SOLIDWORKS. Finite element analysis model was performed on the developed model of knuckle using HYPERMESH. To simulate the effect under operating load conditions OPTISTRUCT AND RADIOSS solver was used. This work discusses the methodology adopted for static, fatigue, and impact analysis of knuckle.

Purushottam Dumbre et al. [2] in his paper “Structural Analysis of steering Knuckle for Weight Reduction” used steering knuckle for study. Weight reduction of steering knuckle is the objective of this exercise for optimization. Typically, the finite element software like OptiStruct (Hyper Works) is utilized to achieve this purpose. For optimization, Nastran/Ansys/Abaqus could also be utilized. The targeted weight or mass reduction for this exercise is about 5% without compromising on the structural strength. Topology optimization can be used to reduce the weight of existing knuckle component by 11% while meeting the strength requirement, with limited design space given with or without change in material properties.

Mahesh P. Sharma et.al [3] has studied static analysis of steering knuckle. In his study he design a knuckle which accommodates dual caliper mountings for increasing braking efficiency & reducing a stopping distance of a vehicle. CAD modal of knuckle was prepared in CREO2.0. Static analysis was done in ANSYS WORKBENCH by constraining the knuckle, applying loads of braking torque on caliper mounting, longitudinal reaction due to traction, vertical reaction due to vehicle weight and steering reaction. Also, reducing the weight of vehicle component plays vital role in increasing efficiency of vehicle and reducing fuel consumption. The study also illustrates the shape optimization of same knuckle and saved material resource. Shape optimization of knuckle was done using ANSYS WORKBENCH making objective function as reducing weight. These FEA results are verified by comparing with analytical calculations. Considering these results modal is modified.

Patel Nirala and Mihir Chauhan et.al [6] carried out the topology optimization of clamp cylinder using CAE tools to reduce weight with the constraints of standard operating condition. The new optimized design of configuration is proposed. FEA of optimized cylinder is also carried out and compared with acceptance criterion. The optimized model is equally strong and light in weight compared to existing model. The topology optimization of the component is carried out and substantial reduction in weight about 70 kg is obtained and also obtained stress and deformation within acceptance criteria.

Chang et.al [8] discussed reliability-based design optimization (RBDO) of an automotive knuckle component under bump and brake loading conditions. The paper explores an application of the moving least squares response surface method that ensures the constraint feasibility in meta-model-based RBDO of a wishbone-type knuckle component.

Prof R. L. Jhala et.al [9] published a paper in which they used forged steel steering knuckle for optimization so that reduction in weight and manufacturing cost of the steering knuckle can be achieved. In this study he assesses fatigue life and compares fatigue performance of steering knuckles made from three materials of different

manufacturing processes. These include forged steel, cast aluminum, and cast iron knuckles. Finite element models of the steering knuckles were also analyzed to obtain stress distributions in each component. Based on the results of component testing and finite element analysis, fatigue behaviors of the three materials and manufacturing processes are then compared. They conclude with that forged steel knuckle exhibits superior fatigue behavior, compared to the cast iron and cast aluminum knuckles.

Fan Pingqing et.al [10] analyzed the steering knuckle of the main operating conditions. The structural analysis of ultimate working conditions is studied by the use of MSC Patran/Nastran software for the steering knuckle. The three different loading conditions are analyzed in this paper, i.e. the uneven ground conditions, emergency braking conditions and sideslip conditions.

K. H. Chang and P.S. Tang et.al [11] discuss an integrated design and manufacturing approach that supports the shape optimization. The main contribution of the work is incorporating manufacturing in the design process, where manufacturing cost is considered for design. The design problem must be formulated more realistically

by incorporating the manufacturing cost as either the objective function or constrain function.

S. Vijayarangan et.al [12] used the different material for optimization of the steering knuckle so that it can work better than existing steering knuckle. Currently, spheroidal graphite (SG) iron is widely used to manufacture steering knuckle in the commercial automobile sector. Also he uses the different material than regular material for optimization of steering knuckle. They use Metal Matrix Composites(MMCs) as it has potential to meet demanded design requirements of the automotive industry, compared with conventional materials. Structural analysis of steering knuckle made of alternate material Al-10wt% Tic was performed using commercial code ANSYS. It is found from the analysis; the knuckle strut region has maximum stress and deflection during its life time. The results obtained from numerical analysis and experimental testing using particulate reinforced MMCs for steering knuckle with a weight saving about 55% when compare with currently used SG iron.

Murali M. R. Krishna et.al[13] presented a paper considering a case study of the application of shape optimization techniques to reduce the weight of a steering knuckle of a heavy truck suspension system. A baseline analysis confirmed that the knuckle was overweight. A shape optimization analysis was undertaken to bring down its weight and at the same time keep the stresses within design limits.

III. PROBLEM IDENTIFICATION

It has been seen from the knuckle producers that weight lessening and propelled materials are the genuine requirement for the present vehicle industry. The steering knuckle represents most extreme measure of weight of all suspension parts, which requires high need of weight decrease. Under working condition is subjected to element powers transmitted from strut and wheel. The weight lessening of steering knuckle is done to such an extent that the quality, firmness and life cycle execution of the steering knuckle are fulfilled. Topology advancement is completed to deliver lighter, less costly and more effective steering knuckle that displays exact measurements, unnecessary machining and requires less part handling.

Because of its huge volume generation, it is just intelligent that improvement of the steering knuckle for its weight or volume will bring about substantial scale reserve funds. It can accomplish the target of decreasing the heaviness of the vehicle part, in this way lessening idleness loads, diminishing vehicle weight and enhancing vehicle execution and fuel economy. So considering car improvement and significance of relative angle, for example, fuel utilization, weight, riding quality, and taking care of, subsequently advancement of new material is vital in the vehicle industry.

Regardless of various arrangements of the steering knuckle/axle get together for every sort of vehicle suspension, the get together is planned to assume a typical part in all sort, and that is to suit the administration stacking. Mass or weight decrease is getting to be distinctly vital issue in auto fabricating industry. Weight diminishment will give significant effect to fuel efficiency, endeavors to lessen emanations and hence, spare environment. Weight can be diminished through a few sorts of mechanical changes, for example, propels in materials, outline and investigation strategies, manufacture procedures and streamlining systems, and so forth. There are four controls for improvement prepares:

a. Topology optimization.

b. Shape optimization.

c. Size optimization.

d. Topography.

IV. METHODOLOGY

For the advancement different CAE instruments are utilized. For creating the 3D display there is software like Catia, Pro-E, strong edge yet Catia is favored in light of the fact that our geometry containing the surface components and Catia is generally utilized for surface elements. Cross section can done by different software like Hypermesh, Ansys. We have chosen Hypermesh in light of the fact that it is having more easy to use interface for complex geometry highlights. For examination software like Ansys, Abacus, Nastran, Radioss, Patran are utilized. Preparatory examination will be directed to the underlying model utilizing OptiStruct to get beginning required data (push conveyance, uprooting, most extreme anxiety, and so forth). That data would be utilized as reference for improvement prepare setting and contrasted with the enhanced model for evaluating advancement handle execution. There are a few emphasess and assessments amid the advancement procedure to accomplish an enhanced model.

In the wake of getting required anxiety conveyance through the part we can discover least and most extreme weight on the segment. Henceforth we can expel the material from low anxiety district. In the wake of advancing the geometry run the examination through Optistruct and view the outcome in Hyperview. Shape improvement was connected to diminish volume of steering knuckle show. OptiStruct was utilized to play out the procedure. Moreover, Hyperview and Hypergraph were utilized to show and plot the information for results elucidation.

Modelling Of Steering Knuckle-Computer aided design model of steering knuckle is produced in 3D demonstrating software CATIA V5 R19. It comprises of stub gap, brake caliper mounting focuses, steering tie-pole mounting focuses, suspension upper and lower An arm mounting focuses. Knuckle configuration for the most part relies on upon suspension geometry and steering geometry. For this review steering knuckle of Macpherson front suspension framework has been chosen. Computer aided design model is produced utilizing summons Pad, Pocket, Filet and geometrical determinations to a limited extent plan module.

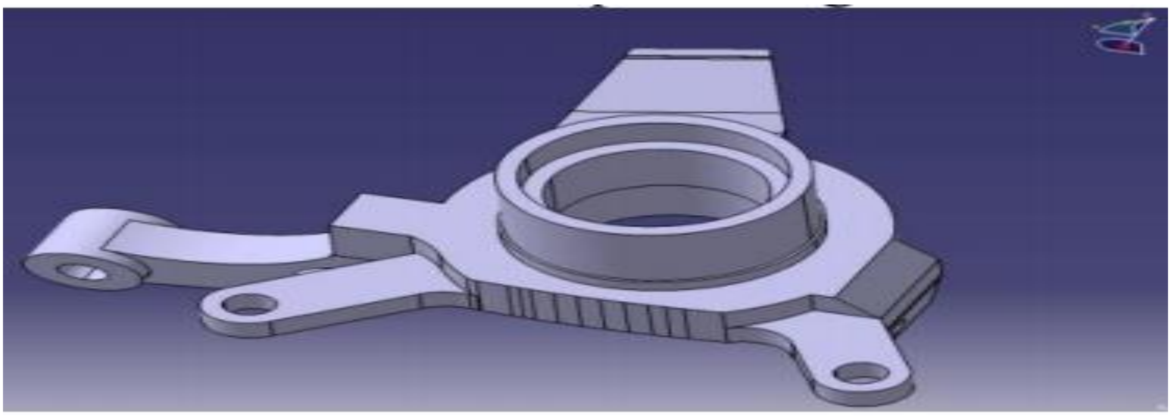


Fig.1 CAD Model of Steering Knuckle

Material Selection:

There are several materials used for manufacturing of steering knuckle 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 Ferrite ductile iron is used. The material used for the production of steering knuckle is S32-HSS/ EN42 (Ferrite Ductile iron). EN42is supplied in accordance with our ISO 9001: 2008 registrations.

Table 1 Composition of Materials

Content	Percentage composition	Content	Percentage composition
Carbon	0.42-0.44%	Silicon	0.10-0.35%
Nickel	3.90-4.30%	Manganese	0.40-0.60%
Molybdenum	0.20-0.40%	Phosphorous	0.05%max
Chromium	1.10-1.40%	Sulphur	0.05%max

Table 2 Mechanical Properties of Materials

Sr.No.	Parameter	Description
1	Material	EN42
2	Young's modulus	200 Mpa
3	Density	7.7×10^{-3} Kg/m^3
4	Poisson's ratio	0.3
5	Tensile stress, yield	350 Pa
6	Tensile stress, ultimate	650 Pa

V. EXPERIMENTAL ANALYSIS

The segment delivered for the experimentation is given by the organization which is utilized for the testing. The info conditions are reproduced in the lab while the part is being tried for execution. The stacking and the limit conditions will coordinate the down to earth working conditions in which the vehicle is relied upon to perform. For effortlessness, a Universal Testing Machine alongside an appropriate apparatus for the segment might be locked in for testing purposes and strain gages are being utilized for recording the strain in the segment while stacking. The heap connected is5000N which is the most extreme load acting along the strut locale.

Experimental set up consists of following components

1. UTM (universal testing machine)
2. Steering knuckle
3. Plunger
4. Strain gauges
5. Data logger
6. Fixtures for holding the component.

Fig.2 Photograph of Actual Model

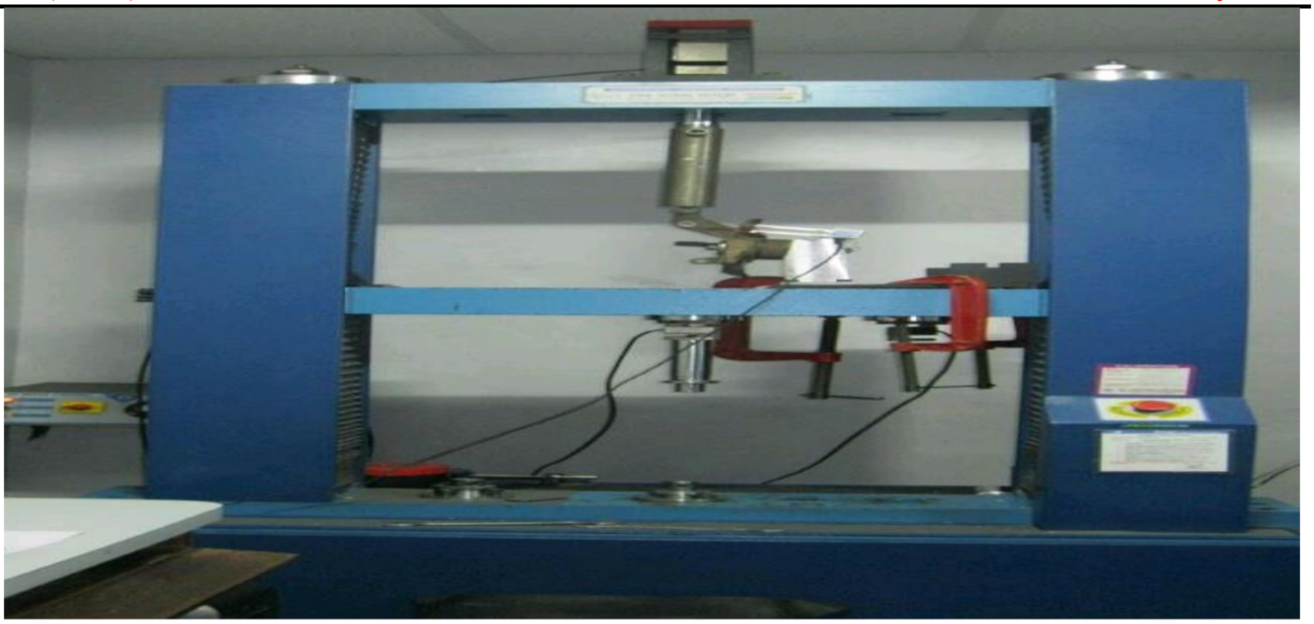


Fig.3 Experimental Set Up

Figure demonstrates the run of the mill test setup for deciding the firmness of the steering knuckle. The progressively expanding burden will be connected and relating disfigurement is resolved. The heap from the heap cells exhibit on the UTM machine will be connected progressively. Show joined to the machine will give a relating plot for load V/s dislodging i.e. solidness of the segment as appeared in chart beneath.

Preliminary Analysis: In order to observe maximum deformation and stresses analysis carried out using HYPERWORKS and OptiStruct solver for conventional model.

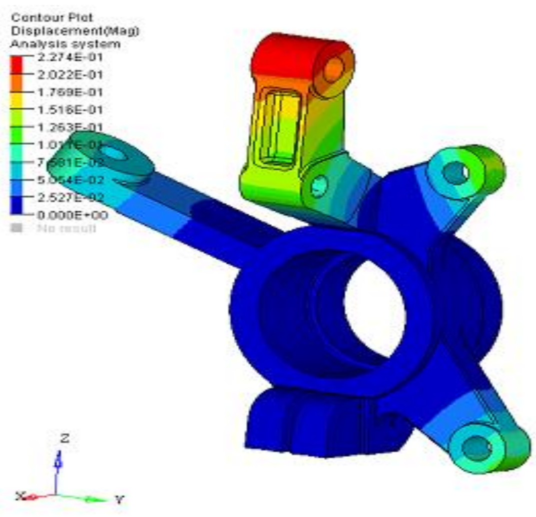


Fig.4 Displacement Analysis of Existing Component

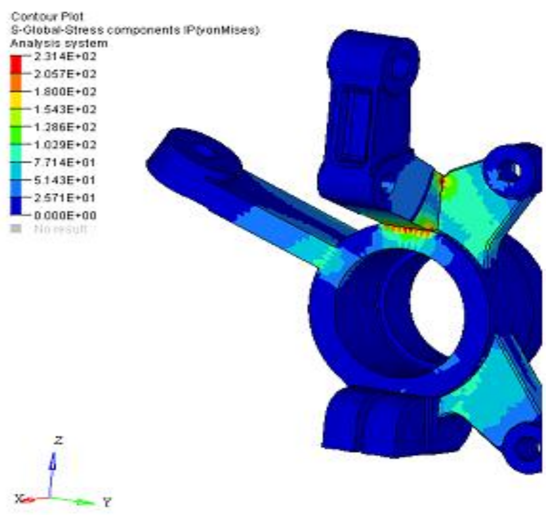


Fig.5 Stress Analysis of Existing Component

From the Fig.4 the nature and extent of displacement can be studied over the contour plot. The max displacement of 0.22 mm is found along one of the arm.

Fig.5 depicts the distribution of stresses along the geometry of the component. The maximum stress recorded as 231.4 MPa is limited to a very insignificant span of the geographical region within the component.

After the analysis of base model the maximum displacement is found to be 0.2mm in the brake clamping holding area as shown in fig 4.The maximum stress is found in junction area brake clamping and shock absorb connecting arm as shown in figure.

International Journal of Advanced Research in Science Management and Technology
IJARSMT
ISSN(Online)-2454-4159 Volume 3, Issue 2, February 2017

The material density distribution is observed for finding the low stress area where we can modify the geometry. Fig6 shows the low stress area where we can remove the material. For removal of material we have to consider the manufacturing aspect and some functional constraint.

The feasibility of tooling required for modified geometry should check at the time of material removal. The geometry after material removal is shown in fig. below

The new geometry is analyzed for the same loading conditions and observed for displacement and stress pattern.

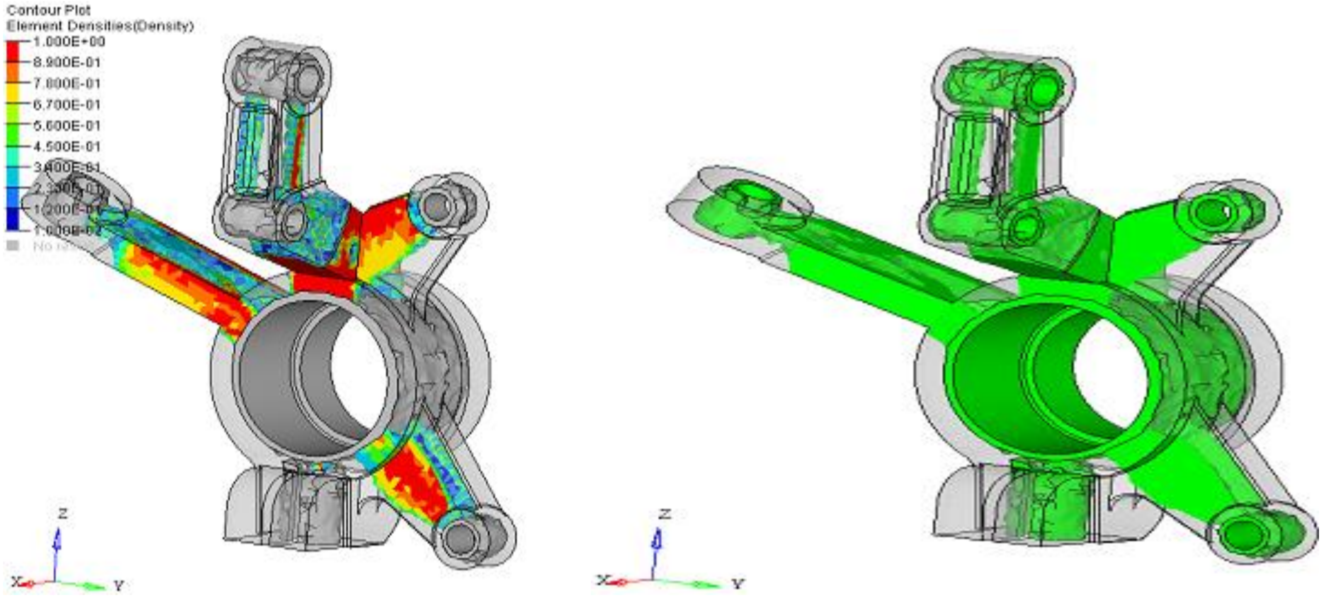


Fig.6 Material Stress Density Distribution

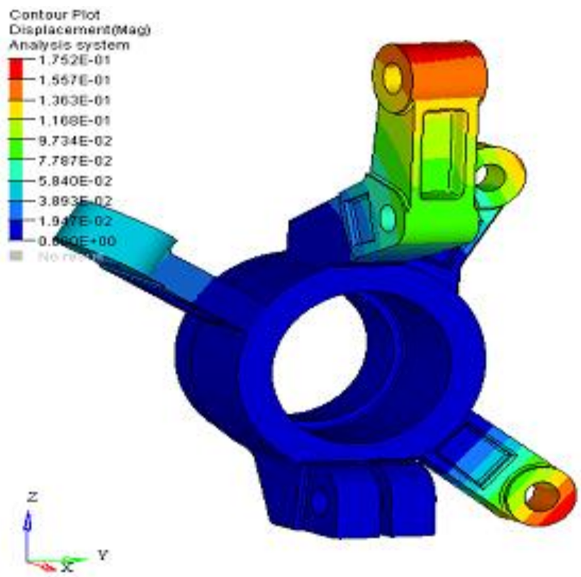


Fig.7Displacement Analysis of Optimized Geometry

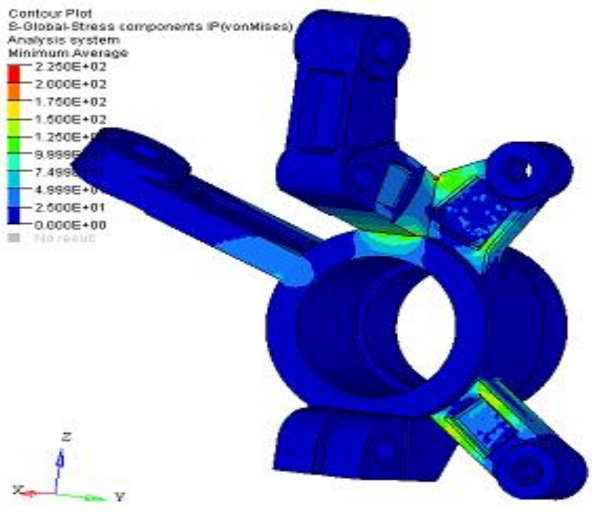


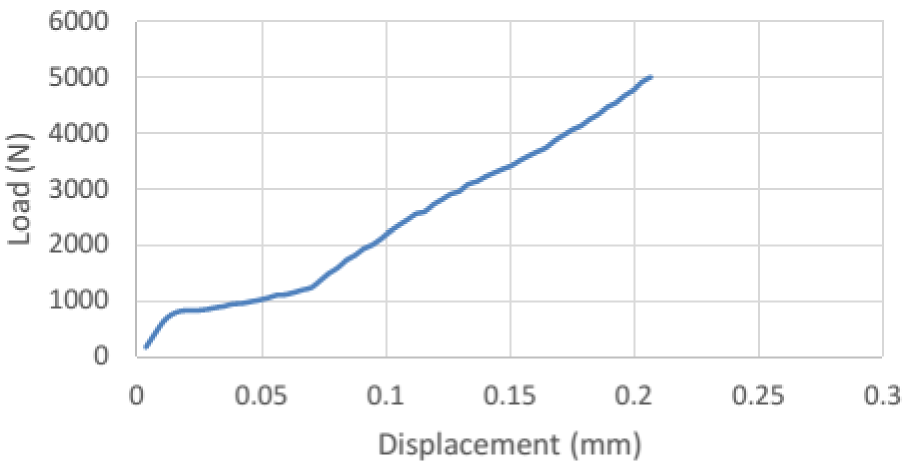
Fig.8Stress Analysis of Optimized Geometry

From Fig 7 the static analysis of the modified geometry it is observed that the maximum magnitude of the displacement is 0.1752mm. From Fig 8the static analysis of the modified geometry it is observed that the maximum magnitude of stress 225 Mpa observed as shown in encircled region.

Table 2 Comparison of FEA Results of Initial and Optimized Model

Parameters	Initial design	Proposed design	% Reduction
Displacement in mm	0.22	0.1752	20.36
Stress in Mpa	231.4	225	2.7
Mass in Kg	3.25	2.9	10.76

Graph 1 Load Verses Displacement



From the above results of graph and display attached to the machine the magnitude of maximum stress obtained is 233.8 Mpa.

Table 4 Result of Experimentation

Parameters	Stress recorded during Expt.
Max. Stress (Mpa)	233.8

VI. RESULTS AND DISCUSSIONS

The Test Report for the segment for checking the outcomes with the Analytical strategy for investigation are thought about it is watched that the outcomes are much closer. Regularly, contingent upon the sort of Test and the application, a mistake edge or around 5 to 10% could be viewed as close towards approving the proposed outline. The anxiety dictated by FEA is 231.4 Mpa and the anxiety controlled by the test is around 233.8 Mpa. Henceforth the rate variety in result is 1.02 %.

From results result through FEA both for existing part and upgraded segment is indicated which comprehends correlation of conduct of segment before &after improvement. The correlation is as appeared in underneath table.

Table 4 Result Comparison between FEA and Experimental

Parameter	Stress Determined By FEA	Stress Recorded During Expt.	% Variation In Results
Max Stress (Mpa)	231.4	233.8	1.02

1. Part is safe under the given loading conditions.
2. The working stresses are less than the yield stress by around 22% to 24% which improves the design life of steering knuckle.
3. To improve performance geometry has been modified using topology and free size optimization which enables to reduce stress levels marginally below yield limit.
4. Less process time will required to melt the material of modified component is reduced.
5. As the cost of component is reduced, market demand and profitability will increase which makes the product competitive in the market.

VII. CONCLUSON

The outcomes got are very good which was normal. Limited component examination is successfully used for tending to the conceptualization and definition for the outline stages. The stresses inferred amid the examination stage ordinarily show the potential arrangement. The cycles are completed in the investigative stage which yields the reasonable qualities for outline parameter. The review has set up a technique for weight improvement of mechanical segments utilizing limited component apparatus i.e. Optistruct. Streamlining technique utilized as a part of this venture is prevailing with regards to decreasing the mass of existing steering knuckle segment by 10.76 %. Despite the fact that there is weight decrease, most extreme anxiety and relocation has not changed fundamentally. Consequently it is sheltered to reason that the streamlined plan is alright for operation.

REFERENCES

[1] Mr. V. Sivananth Dr. S.Vijayarangan Mr. R.Aswathaman "Fatigue and Impact Analysis of Automotive Steering Knuckle under Operating Load Cases" Altair Technology conference 2015 India.

[2] Purushottam Dumbre, A. K. Mishra, V. S. Aher, "Structural Analysis of Steering Knuckle for Weight Reduction" ISSN 2250-2459, ISO 9001:2008 Certified Journal, Volume 4, Issue 6, June 2014.

[3] Mahesh Sharma.P, DenishMevawala.S, Harsh Joshi, DevendraPatel.A (2014), 'Static Analysis of Steering Knuckle and Its Shape Optimization',Journal of Mechanical and Civil Engineering Vol. 3, PP.34-38

[4] Viraj Rajendra Kulkarni and Amey Gangaram Tambe et al „Optimization and Finite Element Analysis of Steering Knuckle" Altair Technology conference 2013 India.

[5] Razak I.H.A1, Yusop M.Y.M2, Yusop M.S.M3, Hashim M.F4, ― "Modelling, Simulation and Optimization Analysis of Steering Knuckle Component for Race Car", ‗ IJRET: International Journal of Research in Engineering and Technology eISSN:2319-1163 | pISSN: 2321-7308, Volume: 03 Issue: 11 | Nov- 2013.

[6] Patel Niral and Mihir Chauhan "FEA and Topology Optimization of 1000T Clamp Cylinder for Injection Molding Machine" Procedia Engineering 51 (2013) 617 – 623.

[7] Wan Mansor Wan Muhamad, Endra ujatmika, Hisham Hamid, & FarisTarlochan. "Design Improvement of Steering Knuckle Component Using Shape Optimization". International Journal of Advanced Computer Science, Vol. 2, No. 2, pp. 65-69, Feb. 2012.

[8] Chang Yong Song, Jongsoo Lee. "Reliability-based design optimization of knuckle component using conservative method of moving least squares meta-models". Probabilistic Engineering Mechanics 26 (2011) 364–379.

[9] Prof R. L. Jhala, K. D. Kothari, Dr. S.S. Khandare. "Geometry and Size Optimization for a Steering Knuckle with no change in attachment Geometry by reducing Production Cost and Weight". International Journal of Advanced Engineering & Application, January 2010.

[10] Fan Pingqing, Zhao Bo, Qiu Long. "The Analysis on Destruction Forms of Steering Knuckle". 978-0-7695-4296-6/11 © 2011 IEEE.

[11] K.H. Chang and P.S. Tang "Integration of Design And manufacturing of structural shape optimization" Advances in engineering software32 (2001) 555- 67.

[12] S. Vijayarangan, N. Rajamanickam, 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.

[13] Murali M. R. Krishna. "Finite Element Shape Optimization of a Steering Knuckle for a HeavyTruck– A Case Study". SAE 2001-01-063.

[14] Mehrdad Zoroufi and Ali Fatemi. "Fatigue Life Comparisons of Competing Manufacturing Processes: A Study of Steering Knuckle". SAE 2004-01-0628.

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

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

[17] Vivekananda Reddi, " Finite Element Analysis And Optimization of The Design Of Steering Knuckle", 'International Journal Of Engineering Research, 2016, Vol No.4, Issue.1.

[18] Rao, S.S., "Engineering Optimization Theory and Practice", John Wiley & Sons, Inc., 2009, 4th edition.

[19] A text book of "Machine Design" by Khurmi and Gupta.

[20] A text book of "Design of Machine Element" by V.B. Bhandari, Tata McGraw Hill Education Pvt Ltd.

[21] A text book of "Finite element in engineering" by Tirupti R. chandrupatla and Ashok D. Belegunda, Third Edition, PHI learning Pvt. Ltd.