SSRG International Journal of Mechanical Engineering
ISSN: 2348 – 8360 /doi:10.14445/23488360/IJME-V7I6P103

Volume 7 Issue 6, 20-23, June 2020

Weight Optimization of Hub and Knuckle Using Topology Optimization

P. Chaudhari[1], R. Khairnar[2]

[1]*Aassociate professor,Department of mechanical engineering, Chhatrapatishahu college of engineering Aurangabad.*
[2] *Student of Department of mechanical engineering, Chhatrapatishahucollege of engineering Aurangabad.*

Abstract: *Steering knuckle is the most stress sustaining and critical component of All-Terrain Vehicle (ATV). The lightweight and high strength components are highly necessary for race car applications. A lightweight and optimized steering knuckle design is proposed to use in a BAJA SAE INDIA off-road race car. A steering knuckle is a component over which a wheel hub is mounted for supporting the steering and braking, which operates under very high-stress conditions. THE 3D CAD model was created by using Cero 3.0. Static and model analysis was carried out in HYPERMESH and ANSYS software to understand its behavior under operating conditions. A CAD model of the existing steering knuckle and wheel hub is applied in the race car. The finite element analysis (FEA) method was used to predict product reactions to real-world forces, vibration, and other physical effects to prove whether a product worked the way it was designed to. At the same time, topology optimization was used to minimize the weight of the component. There was a 24.09 % weight reduction for Hub and 16.30% for knuckle after weight optimization performed using topology optimization. FEA analyzed that the knuckle models are below the stress values and very less deflection under the applied loads.*

Keywords:*Topology optimization, Hub, knuckle, FEA*

I. INTRODUCTION

SAEINDIA, a professional society of automotive, commenced the 11th edition of the BAJA series. Mahindra & Mahindra was the title sponsor, and the event is known as Mahindra presents BAJA SAEINDIA 2018. This competition was conducted in various parts of the world every year, and about 80 universities participate every year from over the world. It is a student competition wherein students are supposed to design, manufacture, and run a prototype of an open-wheel racing car. The competition's objective was to simulate real-world engineering design projects and related challenges and have their design accepted for manufacturing by a fictitious firm. The team's goal was to design and build a single-seat, all-terrain, sporting vehicle, including an assembly containing the driver.

The thesis aims to design and manufacture the front wheel upright assemblies, a steering knuckle over which a wheel hub is mounted to support the steering and braking, operating under very high-stress conditions. The steering knuckle is not a standard part of the terrain vehicle component, but it may be changing for every race car. Thus, the design may vary to fit all sorts of applications and suspension types. The goal is to produce a lighter design compared to the highly successful previous designs of vehicles and not sacrifice performance in stiffness. It was thereby contributing to making the 2018 vehicles better than its predecessor. Reducing the mass of vehicle components helps save material resources and manufacturing costs and improves fuel efficiency.[1]

The function of a steering knuckle is to steer the All-Terrain Vehicle (ATV). According to B. Babu et al., Steering Knuckle plays a major role in many direction controls of the vehicle; it is also linked with other linkages and supports the car's vertical weight.[2]

For the application on a high-performance racing vehicle, it has to meet the following criteria:

- Lightweight to maintain good performance to weight ratio of the race car.
- Optimum stiffness to ensure low system compliance and maintaining designed geometries.
- Ease of maintenance for enhancing serviceability and setup repeatability.

For this team's purpose, the ability to manufacture the house components reduces turnaround time and outside dependability.[3]

II. METHODS AND MATERIALS

This thesis aims to design a Steering Knuckle and Wheel Hub having minimum weight and maximum strength using topology optimization. The topology optimization process's main application involves weight reduction, lightweight to maintain good performance to weight ratio, and increasing the component's strength. As the weight of the components, i.e., Hub and Knuckle,increase,the vehicle's speed reduces. If this component's strength is not good, it can affect the component's life in the vehicle's dynamic conditions; different forces are developed, such as cornering force and bump force. It may fail, which creates a problem in the wheel assembly. Ultimately for the reduction in

manufacturing cost and increasing the life of components.
To satisfy this requirement, Al-6061 T6 alloy is used for Wheel Hub and EN8 for Knuckle. It has a low density and compatible yield strength. Considering the above facts, a CAD model of the Steering Knuckle and Wheel Hub was prepared using Creo 3.0. The model was designed considering general suspension geometry parameters of an off-road vehicle. In the wheel, Hub fitted in bearing and tire mounted on the wheel hub. In this type of knuckle, the mounting of brake caliper and steering arm is directly attached without an external joint. To reduce the cost of manufacturing and required raw material, the knuckle design includes forces exerted by frame, steering arm, brake caliper mounting.
A variety of insights were obtained from the wheel hub review, and upright assembly research and Finite Element Analysis were performed on the front hubs to validate the design.The procedure began with importing the model as per the file format through the import menu and meshing the component or model as per the type of mesh required using different mesh options such as AUTOMESH, TETRAMESH, HEXAMESH, etc. We then applied material, property, boundary condition, created load step, and Optistruct Solver to analyze the results. Then further in the optimization solver applied properly, all parameters such as design variable, responses, design constrain, and objective were determined. Run the result in optistruct Solver.
For testing on the field, we had built a ramp of approximately 45⁰ to test the vehicle at a speed of 40-45 km/hr. The force which was calculated theoretically, according to considered conditions and it was 98.895 KN. Various loads and constraints are added to the model to represent the loading conditions that the parts are subjected to different load cases can be defined to represent different loading conditions on the same model. Solver information is also added to tell the Solver what kind of analysis is being run, which results in export to determine the load from a static or dynamic event. AMultibody Simulation (MBD) might be helpful. The FEM model (consisting of nodes, elements, material properties, loads, and constraints) is then exported from within the pre-processor Hyper Mesh.

Once the solution has ended successfully, post-processing (in Hyper View for contour plots and HyperGraph for 2D/3D plots) of the simulation results is done next. Stresses, strains, and deformations are plotted and examined to see how they responded to the various loading conditions. Based on the results, modifications may be made to the part, and new analysis may examine how the modifications affected the part.
This eventually completes the FEM process.4

Optimization: These methods are developed for manufacturing lighter components. Optimization can be defined as the automatic process to make a system or component as good as possible based on an objective function and subject to certain design constraints. The topology optimization of structures is a rapidly growing field of particular interest to the automotive and aerospace industries. Instead of shape optimization, topology optimization allows the introduction of holes or cavities in structures,resulting in great savings in weight or improvement of structural behavior such as stiffness, strength, or dynamic response.
Response for OptiStruct is any value or function dependent on the Design Variable and is evaluated during the solution. OptiStruct allows the use of numerous structural responses, calculated in a finite element analysis, or combinations of these responses to be used as objective and constraint functions in structural optimization. Responses are defined using DRESP1 bulk data entries.
Responses considered for topology optimization were:

1. Mass response.
2. Static displacement response for caliper mounting points.
3. Static displacement response for tie rod mounting point.

Design Constraint:On every engineering design,some constraints need to be satisfied. These constraints can be defined as a lower bound or an upper bound on any response dependent on the design variable.
Design constraint used for the model-
Upper bound for static displacement response for caliper mounting points.
Upper bound for static displacement response for tie rod mounting points.

Design Variable:We define the variable name and define which type of component it is, i.e., solid, shell, etc.

A. Analysis of knuckle

Different forces acting on knuckle:
Cornering force- The force acting on the upper and lower ball joints during turning or cornering is called cornering force.
Breaking force- The force applied on the mounting point of the Hub during the braking condition is known as braking force.
Force values:-
Bump force-588.6N
Breaking force-1089.96N
Cornering force-27000N
Output
The original weight of the component was 1.19kg.
After optimization, the weight of the component is 0.996kg

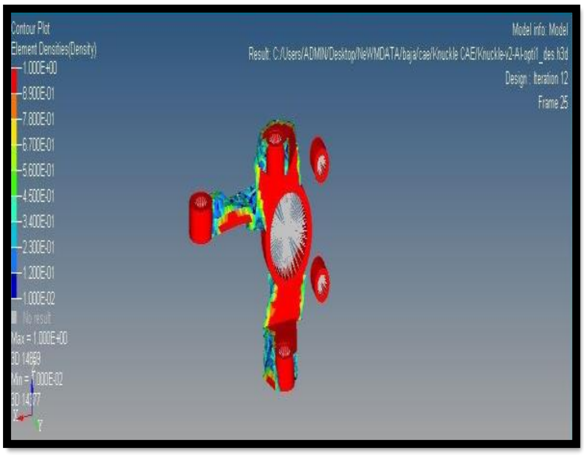

Fig. 1 Optimized Knuckle

B. Analysis of Hub

Different forces acting on Hub:

Bump Force- When a vehicle is at a bump condition, and the vertical force act on the Hub during the bump is called a bump force.

Moment- During the moment, all four corners of the wheel hub were constrained, and the moment was applied at the center.

Force values:-

Moment-318.38N-M

Before, the weight of the component was 1.66kg.

After optimization, the weight of the component is 1.26kg.

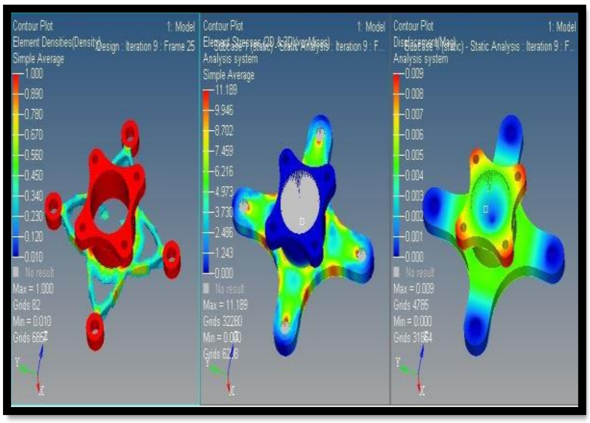

Fig. 2 Optimized Hub

C. Model analysis

The model analysis was carried out to check the component's behavior at different frequencies ranges to avoid resonances.

Object Name	*Total Deformation*	*Total Deformation 2*	*Total Deformation 3*	*Total Deformation 4*	*Total Deformation 5*	*Total Deformation 6*	*Total Deformation 7*	*Total Deformation 8*	*Total Deformation 9*	*Total Deformation 10*	*Total Deformation 11*
State	Solved										
Scope											
Scoping Method	Geometry Selection										
Geometry	All Bodies										
Definition											
Type	Total Deformation										
Mode	1.		2.	3.	4.	5.	6.	7.	8.	9.	10.
Identifier											
Suppressed	No										
Results											
Minimum	0. mm										
Maximum	82.082 mm		87.755 mm	81.332 mm	84.844 mm	52.723 mm	87.05 mm	117.53 mm	119.16 mm	82.389 mm	85.364 mm
Minimum Occurs On	MSBR										
Maximum Occurs On	MSBR										
Information											
Frequency	22104 Hz		22181 Hz	22205 Hz	22294 Hz	25253 Hz	25719 Hz	26394 Hz	26419 Hz	26954 Hz	26970 Hz

Fig. 3 model analysis

III. TESTING ON FIELD

We had built a ramp of approximately 45⁰ for vehicle testing at a speed of 40-45 km/hr—the force which was calculated theoretically according to considered conditions 98.895 KN.

Testing Results:

Damage of knuckle

The welded part that is steering point was broken from the knuckle.Bending of the Steering Tie rod and Ball joint occurred on-field and Bending of the brake disc.

A. Analysis of failure part

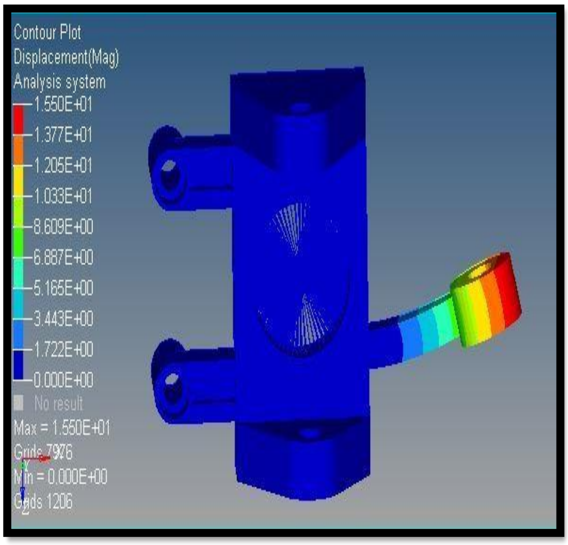

Fig. 4 Analysis of failure part

IV. CALCULATIONS FOR 2ND DESIGN WITH IMPROVED REINFORCEMENT

Upon a failure in optimized design the knuckle, it was reinforced, and calculation for the same was done, which gave results as:
Failure displacement **15.50 mm**
After reinforming displacement**0.7746 mm**

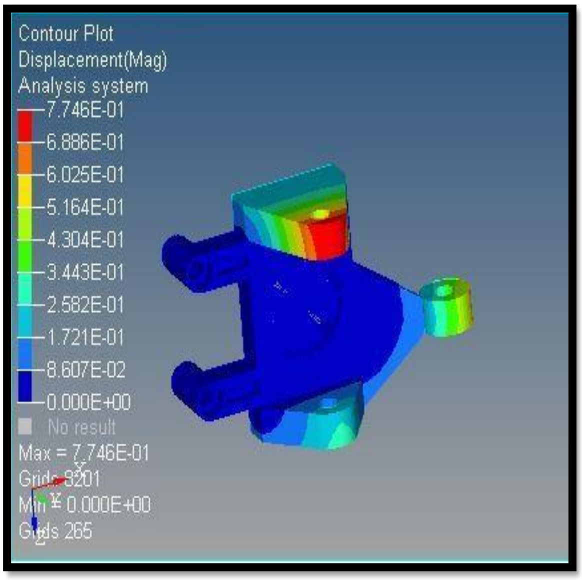

Fig. 5 Analysis of Reinforced Knuckle

V. FINAL RESULTS OF ASSEMBLY

Tyre, knuckle, and Hub arrangement problem was solvedbecause of flange.No damage to knuckle after reinforcement.Successfully ran the vehicle for a 4hours BAJA endurance race.

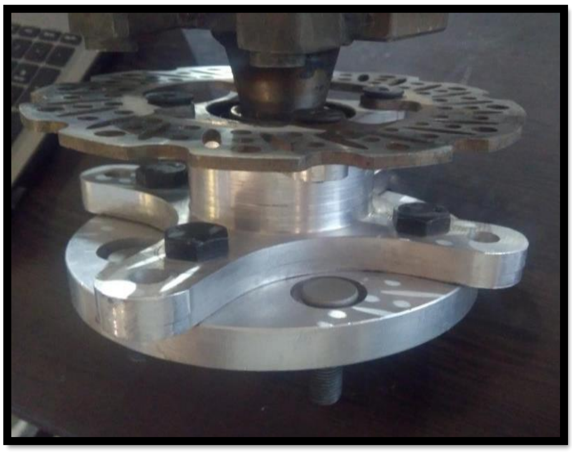

Fig. 6 Actual assemblies of Knuckle andHub

VI. CONCLUSIONS

The study's ultimate goal is to design and produce the steering knuckle and Wheel Hub, capable of bearing loads at the dynamic and lightweight loads. Aluminum 6061- T6 alloy and EN8 was found to be the best material for the component due to the good physical and mechanical properties as well as lightweight. It was analyses through FE analysis that the models of the knuckle and Hub are below the stress values and very less deflection under the applied loads.

ACKNOWLEDGMENT

I want to express my deep sense of gratitude and respect to our guide Prof. P.B Chaudhari for his excellent guidance, suggestion, and constructive criticism. He has been not only a wonderful guide but also a genuine person. I consider myself extremely lucky to be able to work under the guidance of such a dynamic personality.
I am also thankful to Dr. R.P Chopde, HOD of Department of Mechanical Engineering, CSMSS Chh. ShahuCollege of Engineering, Aurangabad, for his constant support and encouragement.
Finally, I extend my sincere thanks to my friends for making this project successful one, for their valuable advice in every stage, andforgiving me an absolute working environment. I would like to thanks all whose direct and indirect support helps me complete my project on time.

REFERENCES

[1] Joost, W. J, Reducing vehicle weight and improving US energy efficiency using integrated computational materials engineering. Jom, 64(9) (2012)1032-1038.
[2] Dusane, S. V, Dipke, M. K, and Kumbhalkar, M. A, Analysis of steering knuckle of all terrain vehicles (ATV) using finite element analysis, In IOP Conference Series: Materials Science and Engineering, 149(1) (2016). 012133. IOP Publishing.
[3] Kokate S.B and Kulkarni G. R, Material Optimization of Wheel Hub using Finite Element Analysis, (2019).
[4] Kavita,RakeshSaxena, LalitRanakoti, AshishBedwal,Topological Optimization using Guide Weight Method, SSRG International Journal of Mechanical Engineering. 2(7) (2015)18-2.
[5] Saxena G, Chauhan A. S, Jain R, and Gupta I,Simulation and Optimization of wheel Hub and Upright of Vehicle: A Review.
[6] Andrew Salzano and Eric Klang, Design, Analysis andFabrication of a Formula SAE Chassis, College of Engineering,North Carolina State University, (2009).