Analysis of Steering Knuckle of All Terrain Vehicles (ATV) Using Finite Element Analysis

2016 IOP Conf. Ser.: Mater. Sci. Eng. 149 012133

(http://iopscience.iop.org/1757-899X/149/1/012133)

Analysis of Steering Knuckle of All Terrain Vehicles (ATV) Using Finite Element Analysis

S V Dusane[1], M K Dipke[2] and M A Kumbhalkar[3]

[1,2]Department of Mechanical Engineering, G. H. Raisoni Institute of Engineering and Management, Jalgaon, Maharashtra, India.
[3]Department of Mechanical Engineering, JSPM Narhe Technical Campus, Pune, Maharashtra, India.
Email: [1]saurabhdusane001@gmail.com

Abstract. Steering knuckle is the most stress sustaining and critical component of All Terrain Vehicle (ATV). Steering knuckle is the pivot point of the steering and suspension system, which allows the front wheels to turn and also allow the movement of suspension arms motion. The light weight and high strength component is always in demanding for racecar application. Lightweight and optimized design of steering knuckle is proposed to use for a BAJA SAE INDIA off road racecar. Due to the failure in knuckle in terrain vehicle after some instances, it has to be modified for better performance. The 3D CAD model created by using CATIA V5 and static as well as model analysis carried out in ANSYS 12 to understand its behavior under operating conditions. All test for frame was carried out on aluminum alloys 6061-T6 & for spindle EN8. The paper discusses the FE analysis of existing and modified Steering Knuckle.

1. Introduction

All-terrain vehicle is design for BAJA event is organized by SAEINDIA and Mahindra & Mahindra in the 2015. The object of the competition is to simulate real-world engineering design projects and their related challenges. Each team is competing to have its design accepted for manufacture by a fictitious firm. Each team's goal is to design and build a single-seat, all-terrain, sporting vehicle whose structure contains the driver [1].

A steering knuckle is a component over which a wheel hub has been mounted and support for steering and braking which operating under very high stress condition. Steering knuckle is not a standard part of terrain vehicle component but it may be change for every racecar. Thus, the design may vary to fit all sorts of applications and suspension types. A CAD model of existing steering knuckle is shown in figure 1 is applied in the racecar as shown in figure 3. Finite element analysis (FEA) is a method for predicting how a product reacts to real-world forces, vibration and other physical effects. FE analysis shows whether a product will wear out, break, or work the way it designed.

The advanced optimization techniques help to explore the light weight architecture. Rajendran et al discusses the process of designing light weight Knuckle from scratch which can be applicable for many casting components. To derive the optimal load path required for the major load cases a topology optimization is performed on the design volume and prepare a concept model from the topology results generated. The model is verified for all the required extreme loads & the durability load which helps for significant mass reduction from model [2].

Chang and Tang presented an integrated design and manufacturing approach that supports shape optimization of structural component. The approach starts from a primitive concept stage, where boundary and loading condition of structural component are given to the designer. Topology optimization is conducted for a structural layout. A 3D tracked vehicle roadarm is employed to illustrate overall design process and various techniques involved [3].

According to B.Babu et al, Steering Knuckle plays major role in many direction control of the vehicle it is also linked with other linkages and supports the vertical weight of the car. Study involves modelling of the steering knuckle with the design parameters using the latest modelling software, and also it includes the determination of loads acting on the steering knuckle as a function of time. This is done for finding out the minimum stress area [4].

Chang Yong Song discusses reliability-based design optimization (RBDO) of an automotive knuckle component under bump and brake loading conditions. The probabilistic design problem is to minimize the weight of a knuckle component subject to stresses, deformations, and frequency constraints in order to meet the given target reliability. The initial design is generated based on an actual vehicle specification. The finite element analysis is conducted using ABAQUS, and the probabilistic optimal solutions are obtained via the moving least squares method (MLSM) in the context of approximate optimization. [5].

In the present study, design of a durable and reliable steering knuckle for a racecar being an ultimate aim to be achieved. Development of racecar components tied with the regulations drawn by the organizer. In existing design of knuckle, which has very less weight, but at the time of race a steering arm has separated from the knuckle due to low strength of knuckle and high strength of bolted joint as shown in figure 2. While steering an arm pulls knuckle towards a car for turning, so it should be assembled with the knuckle to strengthen a steering and braking mechanism. In proposed design, a single piece of knuckle includes the steering arm and brake caliper and not provide any bolted joint to improve its strength and attached to knuckle directly.

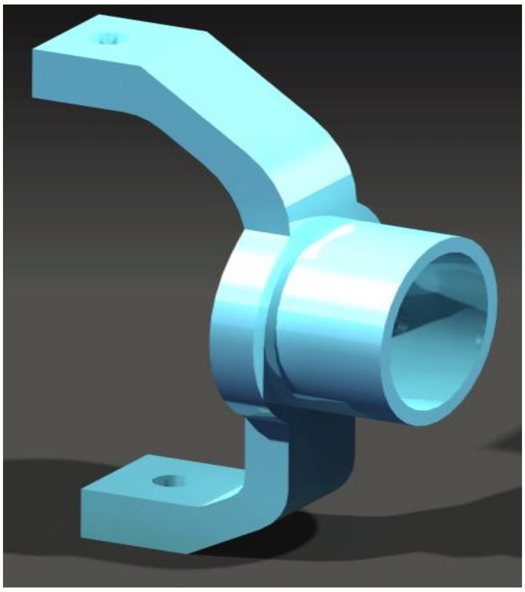


Figure 1.CAD model of existing knuckle

Figure 2. Existing KNUCKLE

Figure 3.SAE INDIA racecar

2. Design of Existing Knuckle

The objective of the study is to design a steering Knuckle have minimum weight as well as maximum strength. To satisfy this requirement, aluminium 6061-T6 alloys are the best option for nowadays-automobile industry due to light weight as well as has low density and compatible yield strength. Table 1 shows the physical and mechanical properties of the Aluminium 6061 alloy. Considering above facts, a CAD model of the steering Knuckle was prepared using CATIA V5 as shown in Figure 1. The model was designed, considering general suspension geometry parameters of an off-road vehicle. The existing knuckle is a Hub type is as shown in figure 2. In which the wheel hub fitted in bearing and tyre mounted on wheel hub. In this type of knuckle the mounting of brake calliper and steering arm is directly attached without an external joint but, to reduce the cost of manufacturing as well as required raw material, design of knuckle includes force exerted by three parts.

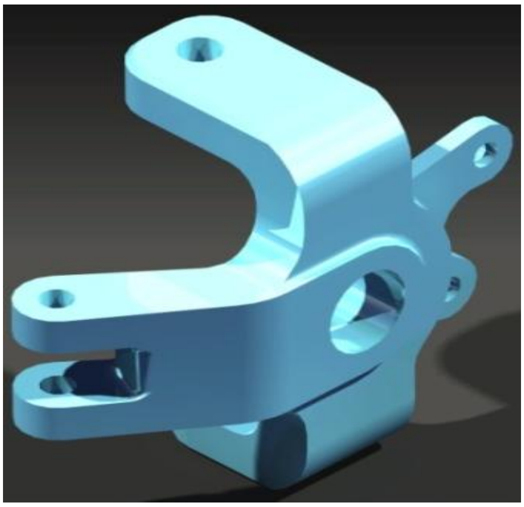
i. Frame
ii. Steering arm
iii. Brake Calliper mounting

Loads consider as per weight biasing and resolve in three component x, y and z direction. G's consideration for force calculation in x, y, and z direction is 6.6, 4.6 and 2.3 respectively.

3. Structure and Design of Modified Knuckle

The aim of design is produce reliable and durable steering knuckle for a racecar to overcome the previous year's failure. Proposed design of knuckle is spindle type, in which, the frame and spindle is is made up of same or different material. The material is used to design a frame of vehicle is AL6061-T6 whose properties are given in table 1. To restrict the lateral movement of wheel hub, lock nut arrangement is provided on end of spindle and to increase the strength of bolted joint, spindle is made-up of material EN8. The design process was started with preliminary study on the existing steering knuckle constituent used for the previous racecar including investigating the existing knuckle design.

The design also needs to follow the criteria and regulations drawn by BAJA organizer, mainly depends on suspension as well as steering geometry. In general, a steering knuckle has three connections on the body part connecting to the upper arm, lower arm and tie rod. Therefore, the design needs to stress on these three connections, as well as one side of connectors where brake caliper attached. Then final design evaluated through FEA simulation in order to estimate the deflections, stress distribution as well as the weight. According to the result the optimized the design by thickness of material or applying fillet and chamfer on corners. The proposed design is shown in figure 4.



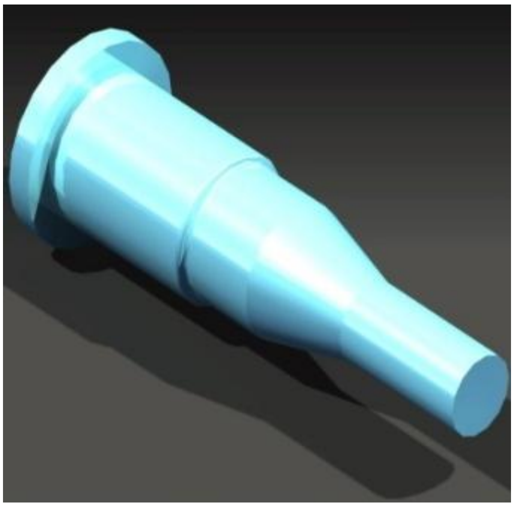


Figure 4. Modified Knuckle Design (Frame) **Figure 5.**Modified Knuckle Design (Spindle)

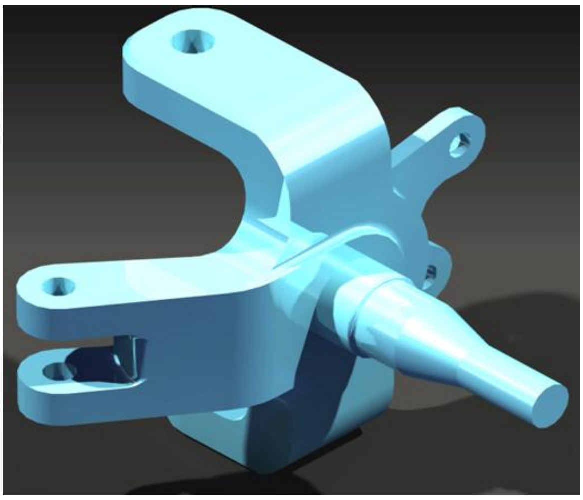


Figure 6. Modified Knuckle Design (Spindle)

3.1 Material Selection

Various types of materials are currently used for the steering knuckle component like grey cast iron, white cast iron. These materials have high yield strength but the weight of the material is more which is the limitation of racecar. Hence searching for the alternate material with nearest yield strength and light in weight, for this factor and cost consideration the Aluminium alloy 6061-T6 is used.

Table 1. Material Input Data for Al 6061-T6 and EN-8

Property	Unit	Al 6061-T6	EN-8
Density	Kg/m^3	2700	2700
Tensile yield strength	MPa	276	276
Tensile ultimate strength	MPa	310	310
Young Modulus	GPa	69	71
Poisson ratio		0.3	0.33

3.2Load Distribution

- Curb weight of vehicle considered = 2500N
- Weight of One person As per SAE rule = 1110 N
- Gross Weight of Vehicle = 3610 N

For analysis consider when vehicle is jump and landing on single front wheel so, total weight of vehicle comes on spindle

I. Load on spindle = 3561 N
II. Braking Force = 3500 N
III. Braking Torque = 331.5 N-m
IV. Force On Steering Arm = 600 N

4. Finite Element Analysis

For the FEA of existing and modified Knuckle, 3D model is created in CATIA v5 and save in IGS format and imported in ANSYS 12.0. The material properties as shown in table 1 have been assign in engineering data. Model is mesh with Solid 187 hexahedral 10-node element. The solid elements has three degree of freedom i.e. translation in X, Y and Z direction.

The finite element analysis of knuckle has been carried out for different boundary condition and observed the stress level as per material property of material. All the results carried out using ANSYS 12.0 are shown in tabulated format.

4.1. On Existing Design

An existing design is hub type steering knuckle, to observe that maximum stress produce into steering knuckle, model subjected to extreme conditions. Steering knuckle was constraint at upper and lower ball joint mountings. As per loading conditions, the weight biasing on front side on each wheel 60kg weight considered. According to speed of vehicle, three component of force was considered on x, y, z direction. Apply load of1400 N, 2800 N and 4000 N on X, Y and Z direction respectively, as shown in figure 7. A mesh model of existing knuckle is shown in figure 8 having 181775 Nodes and 120120 elements. Referring to the analysis results, the maximum stress on designs are less than material yield strength and very less deflection under the assigned loads. In addition, mass of the models are also evaluated in the CATIA software since the ultimate aim of the current project is to design a lightweight steering knuckle.

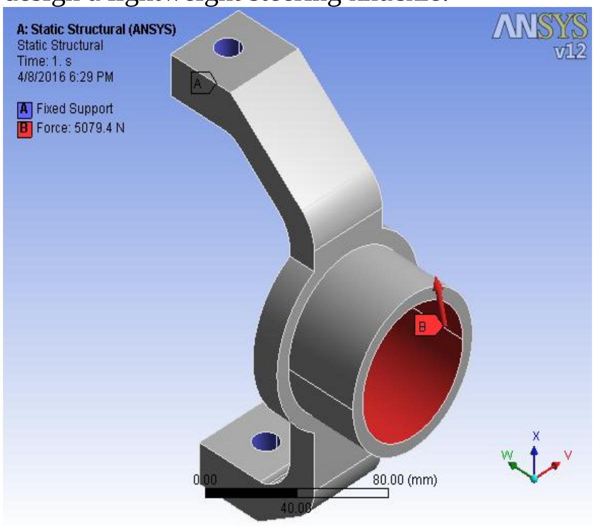


Figure 7. Boundary Conditions

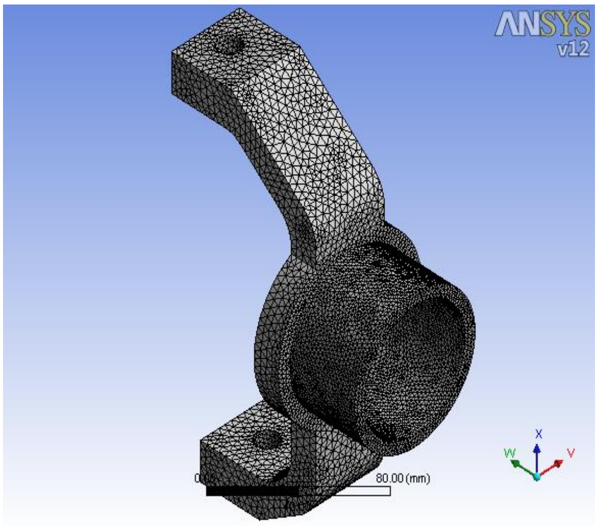


Figure 8. Mesh model of Existing knuckle

IOP Conf. Series: Materials Science and Engineering **149** (2016) 012133 doi:10.1088/1757-899X/149/1/012133

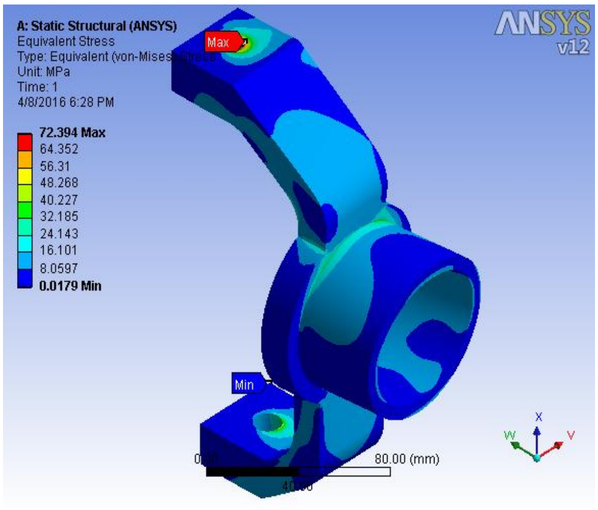


Figure 9. Equivalent Stress

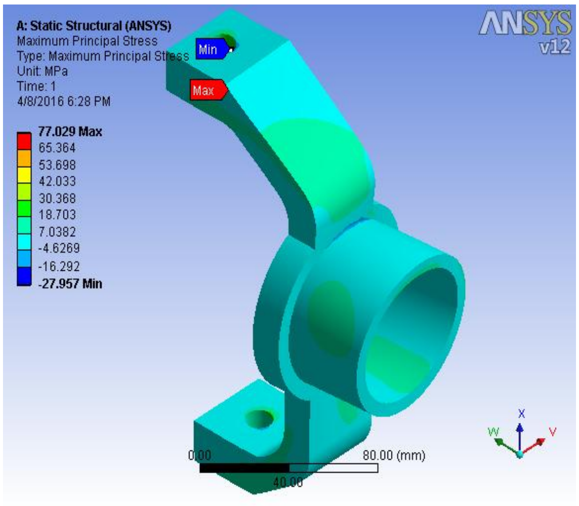


Figure 10. Maximum Principle Stresses

4.2. On Modified Design

The modified design of knuckle is spindle type. In this type, wheel hub rotated on spindle and spindle fitted in frame by interference fit. For reducing material and according to loading condition, the modified knuckle is designed in two parts.

I. Frame II. Spindle

4.2.1. Frame

The frame is a structure in which the upper and lower suspension arm pivoted and it consists of mounting for steering arm and brake calliper. For FE analysis, the boundary conditions apply as discuss in load distribution section and as shown in Figure. 11 and mesh model is shown in figure 12 having 196446 nodes and 127590 element. By observing result, it is found that it has very less deformation as 0.14115 mm.

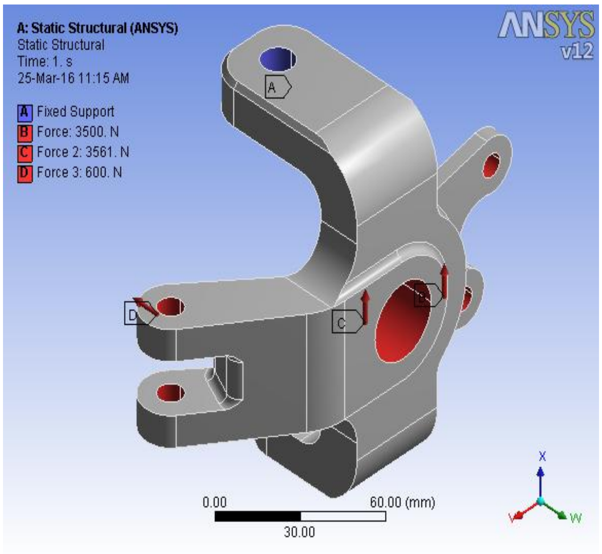


Figure 11. Boundary conditions for Frame

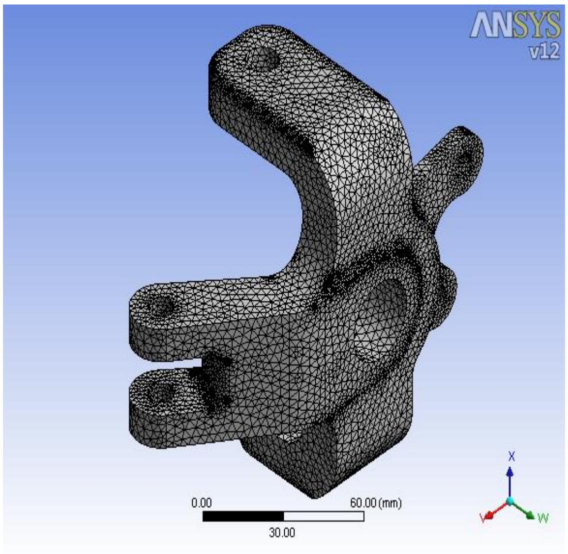


Figure 12. Mesh model of modified knuckle

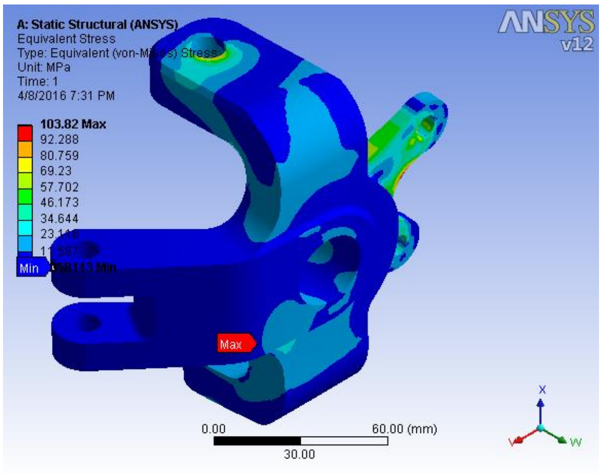


Figure 13. Equivalent von-meshes stress

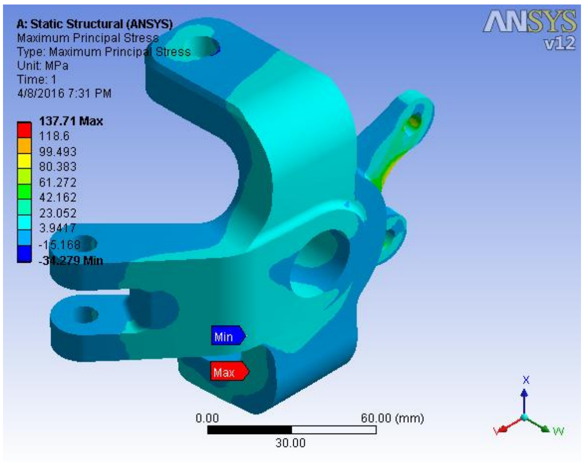


Figure 14. Maximum principle stresses

4.2.2. Spindle

The wheel hub rotated around the spindle, hence vertical force acting on spindle because of dynamic weight transfer at the time of landing of car after jump. Therefore, the spindle has capacity of sustain all forces, which experience during dynamic condition. Keep this fact in mind decided to use the EN8 material for spindle. EN 8 is easily available in local market and it is cheaper. The boundary condition for FEA is as shown in figure 15. After meshing of spindle 147630 nodes and 101000 elements are generated.

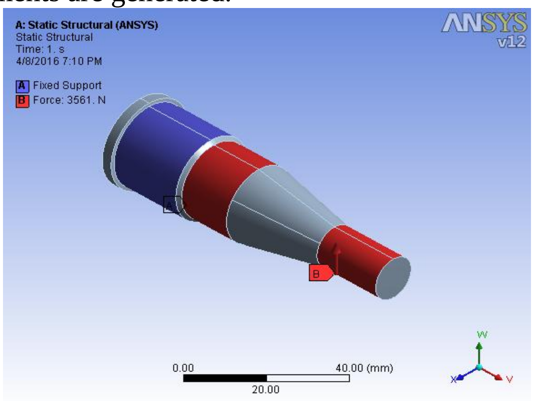


Figure 15. Boundary condition

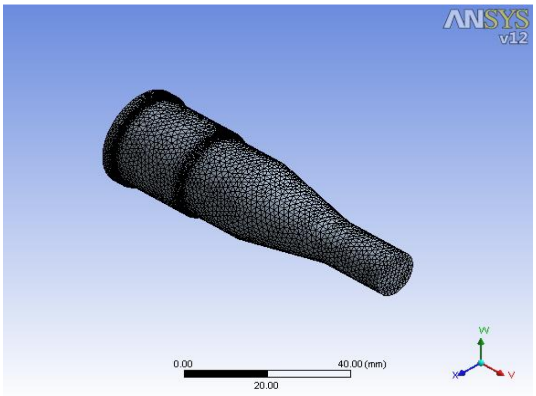


Figure 16. Mesh model of spindle

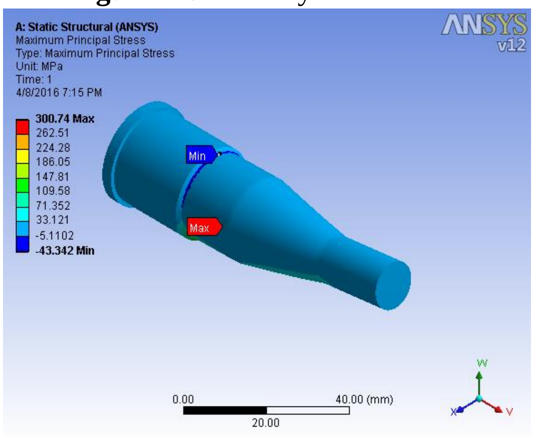


Figure 17. Normal Stresses

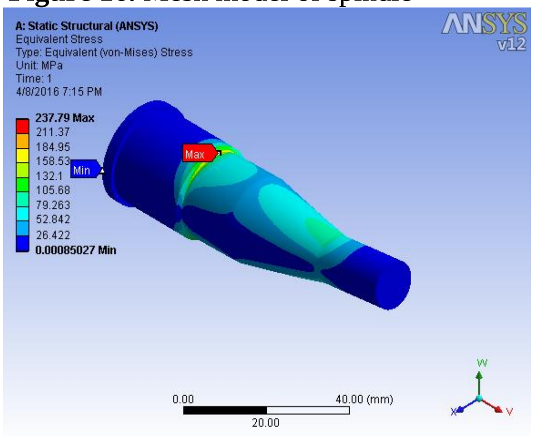


Figure 18. Equivalent Stresses

5. Result & Discussion

From the analysis of existing and modified knuckle, it has been observed that the stress is quite changes in the modified design but the location of maximum stress have same. In the existing design the maximum stress is observed where the upper and lower ball joint are pivoted. The existing design has a maximum stress of 72.394MPa is occurred near pivot.

Modified design is complete one piece of knuckle steering where holes for bolting is not required as it has separate arrangement. Due to this, for the same boundary condition, the stress location has been changed and it is observed that it maximum at mounting of steering. The complete FEA results for the existing and modified design areas shown in table 2.

Table 2. FEA Results Compression for Existing and Modified Knuckle

Parameter	**Unit**	**Existing Design**	**Modified Design**	
			Frame	**Spindle**
Displacement	mm	0.30999	0.14115	0.0817
Von-Mises Stress	MPa	72.394	103.82	237.79
Max. principle stresses	MPa	77.029	137.71	300.74
Factor of Safety	-	3.8677	2.5255	1.18
Nodes	-	181775	196446	147630
Elements	-	120120	127590	101000

6. Conclusion

The Existing design of knuckle is to be applied for an SAEINDIA BAJA 2015 car was successfully fulfils the load-bearing requirement. The external arrangement for steering arm and Brake calliper is failed in Dynamic condition. The ultimate goal of study is to design and produce the steering knuckle, which capable to bear loads at dynamic condition as well as light Weight. Aluminium 6061-T6 alloy was found to be the best material for the component due to the good physical and mechanical properties as well as lightweight. It was analysed through FE analysis that the models of the knuckle are below the stress values and very less deflection under the applied loads. The model to be analysed further taking consideration of the good stress results. Hence for further modification in terrain vehicle for SAEINDIA BAJA 2016 racecar an existing knuckle has to be replaced with modified knuckle to improve its strength to reduce failure of joint.

Acknowledgment

The authors would like to thank to all technical staff of mechanical engineering department for their support in all possible ways. We would like to extend a special thanks to the management and Principle Dr. Prabhakar Bhat for financial support from G.H. Raisoni Institute of Engineering and Management, Jalgaon, India.

References

[1] SAE India Baja Collegiate Design Series 2015 *Baja SAEINDIA Rules*.
[2] Rajeev Sakunthala Rajendran, Subash Sudalaimuthu and Mohamed Sithik 2013 Knuckle Development Process with the Help of Optimization Techniques *Altair Technology Conference, India*.
[3] K. H. Chang and P.S. Tang 2001 Integration of Design And manufacturing of structural shape optimization *Advances in engineering software* 32 **555-567.**
[4] B.Babu, M.Prabhu, P.Dharmaraj, R.Sampath 2014 Stress Analysis of Steering Knuckle of Automobile Steering System *International Journal of Research in Engineering and Technology*.
[5] Chang Yong Song and Jongsoo Lee 2011 Reliability-based design optimization of knuckle component using conservative method of moving least squares meta-models *Probabilistic*

Engineering Mechanics 26 **364–379**.

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

[7] Tim Palmer and Eric Nelson 2011 Optimizing Next-Generation Automotive Structures Using Altair OptiStruct *Plastics in Lightweight and Electric Vehicles Livonia, Michigan* **7 - 9.**