International Journal for Multidisciplinary Research (IJFMR)
E-ISSN: 2582-2160 ● Website: www.ijfmr.com ● Email: editor@ijfmr.com

Design and Optimization of Vibrating Load Analysis Steering Knuckle with Ball Joint by Using FEA

V. Elavarasan[1], M. Shankar[2], T. Sridhar[3], R. Ranjithkumar[4]

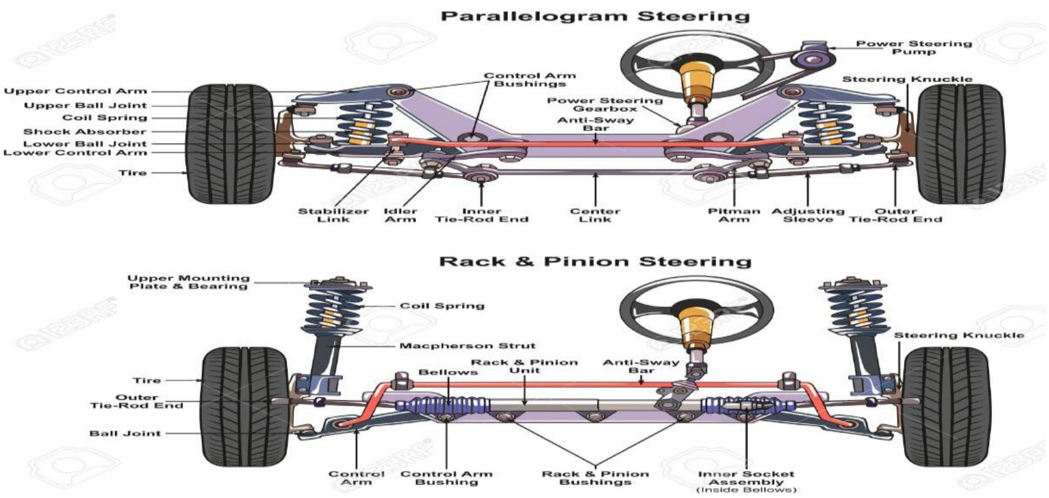
[1]Department Of Mechanical Engineering, Surya Group Of Institutions, Vikravandi, (Villupuram)
[2]Associate Professor, Department Of Mechanical Engineering, Surya Group Of Institutions, Vikravandi, (Villupuram)
[3,4]Assistant Professor Department Of Mechanical Engineering, Surya Group Of Institutions, Vikravandi, (Villupuram)

Abstract:
Steering Knuckle with Ball Joint is a component, to linking the suspension of steering wheel hub to the chassis of a vehicle. Its aims to remodeling the steering knuckle to connect the ball joint hole to provide the internal bush is applied for reduce the oval shape of steering knuckle in order to reduce the vibration of steering wheel in safety for better performance of the vehicle. In this process has been used for initially modeling the knuckle as per the structural design constraints determination of loads acting on the knuckle. The next step is stress analysis using finite element analysis for reducing oval shape without compromising on the structural strength. To analysis the results, material can be added to parts that are subjected to higher stress than the safety factor permits.

Keywords: Steering Knuckle, vehicle, design, modeling, analysis

INTRODUCTION

A steering knuckle is one of the supporting on vertical weight of vehicle's, It has connected to the suspension, brake, and steering subassembly of a vehicle, It serves multiple essential functions, including mounting the wheel hub and bearing assemble to enable the steering arm rotation around the wheel and securing the brake calipers for the disk braking system. To performance of steering knuckle mainly strength, stiffness, and durability. A reduction in the weight of suspension components also improves the vehicle handling performance. The wheel bearing is mounted with a hub to bolt on bearing flange. It delivers all the forces generated at chassis on the suspension system. The design is usually done considering the various forces acting on it which involves all the forces generated by the road reaction on the wheel, The project deals with creation of geometric model of steering knuckle in Autodesk Fusion 360 after that that model will be imported to simulate the finite element modeling analysis, where the meshing properties, element properties will be generated. They added the brass material into the internal bush is applied to steering knuckle with ball joint. To loads and model conditions applied to generating static structural analysis and vibration analysis.



PARTS OF THE STEERING KNUCKLE

The part of the steering knuckle component are given below,

1. Upper control arm
2. Lower control arm
3. Steering arm
4. Brake caliper

The shape and size of steering knuckle component depends upon the vehicle because vertical load of the vehicle is directly act on it and hence the steering knuckle component subjected to power thrust from tie rod to the steering knuckle component subjected to power thrust fro tie rod to the stub axle and hence it must be strong and inflexible in nature. The steering knuckle in the automobile industry can be made either by forging or from casting.

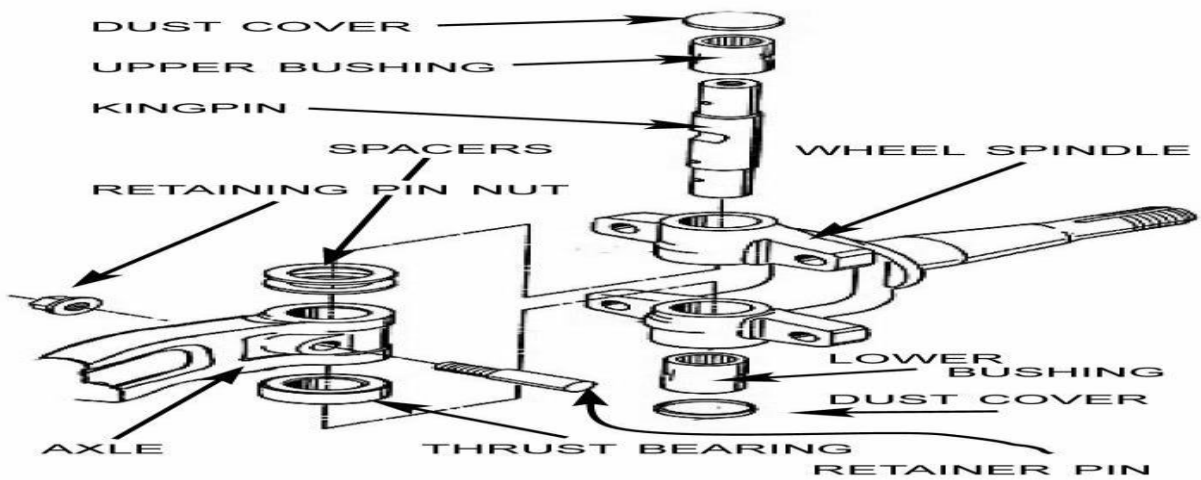


Fig 1.2 Parts of steering knuckle component

The previous authors to study design and analyzing the steering knuckles given below, In this paper, author studied have done static analysis of steering knuckle component. The design of Steering Knuckle component is done with the help of Solid works. Steering Knuckle model is prepared in solid works

(2022) and the static analysis is done in Ansys. Its use of material for the steering knuckle component and compare it, made from Aluminum 6061 T6 material. Keywords: Steering Knuckle, Aluminium 6061 T6, Solid Works (2022), Ansys Student Version.

In this paper author had identified the problem an unconventional shape and it not only act as the connecting entity between steering, brake and suspension system, but also links chassis and wheel hub. Even it is the high stress bearing and critical component of an All-Terrain Vehicle (ATV). Shape optimization was carried over in ANSYS Workbench, whose objective function is to reduce weight, in order to save the material resource.

(2018) in this paper have done static analyses of steering knuckle component. The design of Steering Knuckle component is done with the help of Computer Aided Engineering. Steering Knuckle model is preparing in solid works and the static analysis is done in ANSYS15.0 ,the material for the steering 1 International Journal of Pure and Applied Mathematics Volume 120 No. 6 2018, 3845-3852 ISSN: 1314-3395 (on-line version) url: http://www.acadpubl.eu/hub/ Special Issue http://www.acadpubl.eu/hub/ 3845 knuckle component made from AL 7075 which is suggested material.

In futures, to conduct model analysis to understand the dynamic behavior of the structure and there followed by transient structural response analysis. To improving their overall performance and efficiency. The suspension system depends on design requirements, to requiring the optimization of the variable density topology in the car's design. This may they analysis is used to assess the component's durability under varying forces and vibrations, to simulating different loads to understand its performance. This analysis helps designers to changing the internal dimensions and added the new brass bush components, it's before potential failure, preventing structural failures, a static stress analysis is captured from the component's performance. A component which is connected to steering, suspension and brake to chassis of vehicle. It plays a role in minimizing the vertical motion of the vehicles, to utilize a poor performance of passengers experience in rough roads on highly vibrating on steering wheel. To maximum amount of weight will be calculated to analyze the proper simulating in failure parts are importing in all suspension components.

Table.1-Bush Dimensions

SI No	BUSH DIMENSIONS	MATERIAL	DIMENSIONS IN MM
1	INTERNAL DIAMETER	BRASS	12
2	OUTER DIAMETER	BRASS	18
3	THICKNESS	BRASS	6
4	HEIGHT	BRASS	25

DESIGN OF STEERING KNUCKLE

In this study, to observe the maximum stress to produce steering knuckle, model is subjected to extreme conditions and static and force analysis is carried out in Autodesk Fusion 360. "Table 1 and Table 2" Depicts data collected from off-road design team for the forces calculation on knuckle.

Mass of vehicle	m	3600 kg
Weight Distribution	W.D	40:60
Turning Radius	R	8 m
Wheel Track	C	1422 mm
Wheel Base	B	1524 mm
Height of C.G.	H	635 mm
Velocity	V	45km/hr
Suspension Spring Stiffness	K	25N/mm
Suspension Max. Travel on Bump	X	101.6 mm
Steering Wheel Input Force	Fsi	48N
Steering Gearbox Ratio	G	13.24
Lateral Acceleration During Turning	At	19.53m/s2

Table1.Design Constrains

Table2.LoadingCondition

Moment Due To Braking Force	169029Nmm
Moment Due To Steering Force	49682.72Nmm
Longitudinal Force	2017.96N
Lateral Force	2325.50N
Vertical Force Due To Weight	726.32N
Vertical Force Due To Longitudinal Weight Transfer	1023.38N
Vertical Force Due To Lateral Weight Transfer	3181.80N
Vertical force Due To Bump	2539.00N

3D MODELING & ANALYSIS

To analyze the Solid Modeling, Meshing and FEA A knuckle with internal brass bush was designed for the calculated loads, taking all the mounting points and surfaces as critical points. The material was considered to be Brass and Cast Iron. The solid modeling was done in Autodesk Fusion 360. The weight of the knuckle was 4.212 kg. Stress and Vibration Analysis is performed on the knuckle by applying the various loads calculated in the previous section. Loads are applied in simulate real-time road conditions. The analysis was done using Autodesk Fusion360.

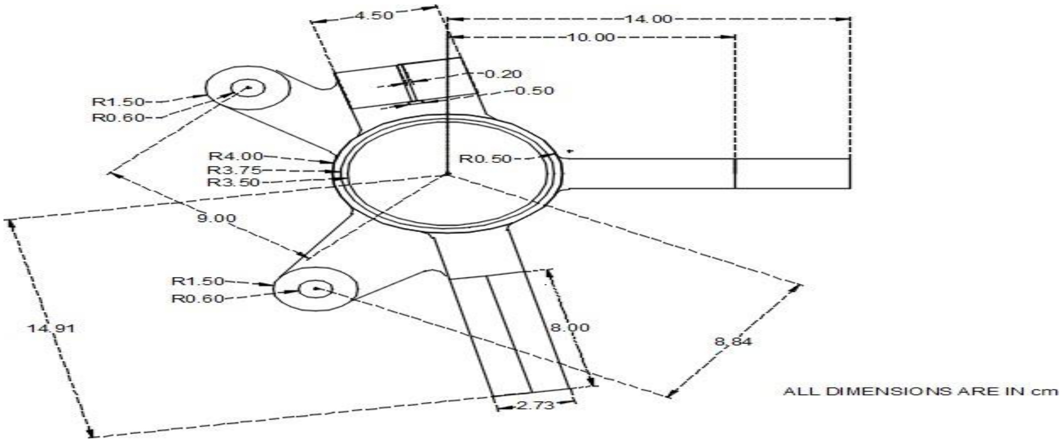
Maximum Stress was found out to be 113.56 MPa. Hence the factor of safety (FOS) is 1.95. The displacement analysis shows that maximum displacement in the component 0.75184mmwhich is well within limits. The Boundary conditions for a FEA problem are basically the point so application of the constraints as well as the forces on the knuckle. The boundary conditions depend mostly on the geometry of the knuckle. Generally the suspension mountings are constrained as they are in direct contact with the chassis of the vehicle.

DESIGN MODELING

GEOMETRIC 2D MODEL BY AUTOCAD

The 2D model of the steering knuckle is designed by using AutoCAD software. Here the dimensions of the steering knuckle component extracted from the TATA indica eV2 model vehicle by using the reverse engineering concept.

Fig 2.1 model of the steering knuckle component



GEOMETRIC 3D MODEL BY AUTODESK FUSION 360

The AutoCAD model of steering knuckle component is generated as 3D modeling with the same software. It does consist of upper control arm lower control arm, tie rod or steering arm and brake calipers. The design is mainly designed depends on the suspension upper control arm, lower control arm, and brake calipers geometry.

A. Study Report by Autodesk Fusion 360

<u>I. Steering Knuckle</u>

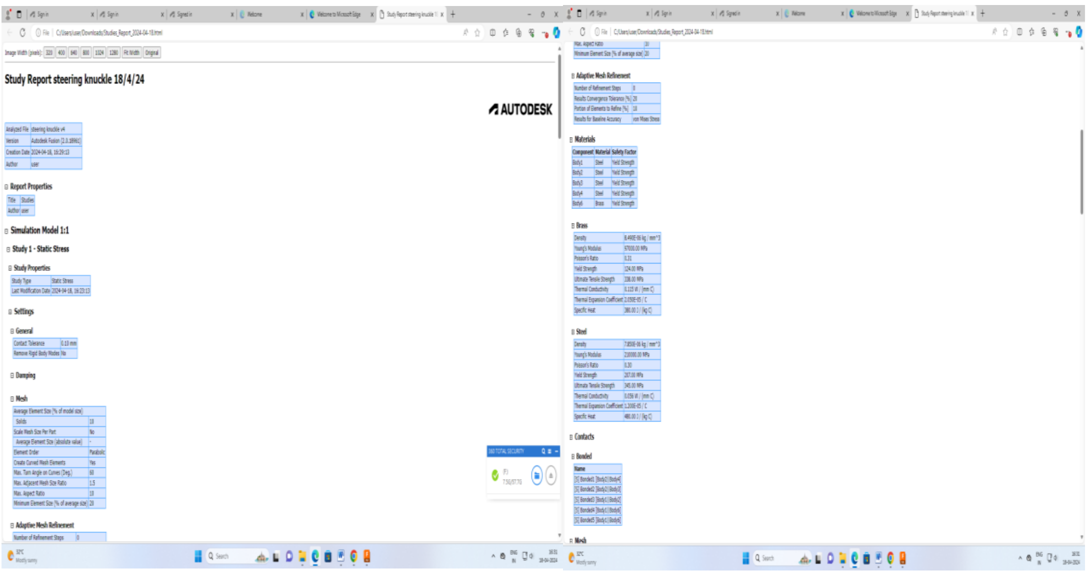


Fig 3.1 Autodesk report of steering knuckle component

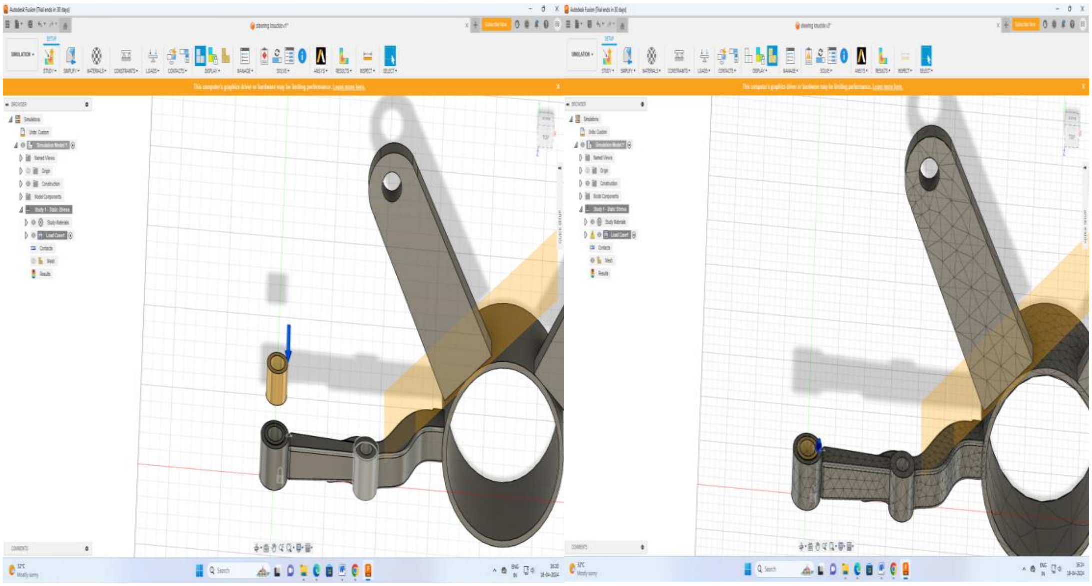


Fig3.2 Solid and Meshed Model of Steering Knuckle with brass bush

II. Steering knuckle with Ball joint

Steering knuckle with ball joint 23/4/2024

Analyzed File	ball joint simulation v1
Version	Autodesk Fusion (2.0.18950)
Creation Date	2024-04-23, 11:39:30
Author	admin

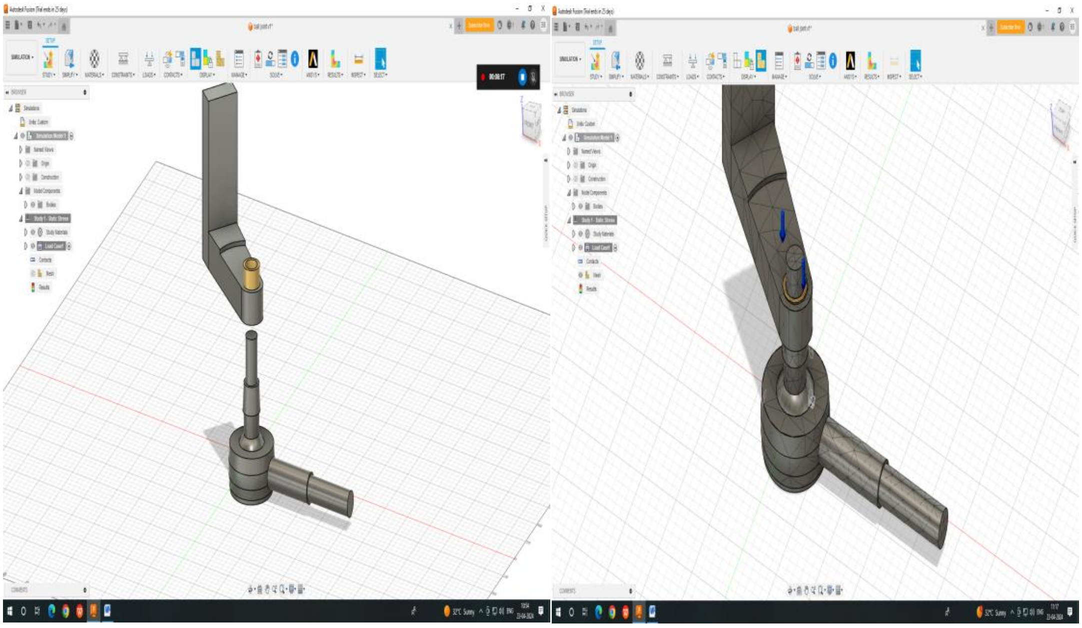


Fig3.3 Solid and Mesh model of Steering Ball joint with brass bush

The preliminary study on the current TATA indica eV2 vehicle steering knuckle component used for the investigation as published in the market as a product. The design also needs to follows the criteria and boundary and loading conditions. Which the size should be mainly depends on the suspension and brake calipers. The model is consisting the material as grey cast iron with yield strength of 240Mpa and the mass of the steering knuckle component is 3.536kg.

DESIGN CALCULATION

Boundary Conditions

Moment at A hub is to acting moment N-mm in clockwise direction.
Moment at B brake pad 1 is to be act moment N-mm in clockwise direction.
Moment at C brake pad 2 is to be act moment N-mm in clockwise direction.
Fixed support at D hub outer is to be fixed at bottom and inner surface ofthe hub.
Force (F_u) acting on E the lower control arm downward force (Y) in N.
Force (F_L) acting on F the lower control arm downward force (Y) in N.
Force (F_G) acting on G the steering arm the rod (-Z) in N.

Loading Conditions (Mass Acting On The Steering Knuckle**)**

There are two types of load acting on knuckle i.e. force and moment This knuckle is designed for vehicle of 1080 kg weight so braking force acting on it produces a moment.
The kerb or curb mass of the TATA indica eV2 model car = 1080 kg. Average weight of 5 members in 72 kg consider = 360 kg
The gross weight of the vehicle = curb weight + mass of the total member
= 1080 + 360 (if the average weight 72kg
multiples of 5 members as considers)
Gross weight = 1440kg
1/4 of the weight of whole vehicle design acting on the steering knuckle
=1440/4
= 360kg
Therefore, the mass acting on the steering knuckle component is,
m = 360kg

Loading Condition (Calculations)

As per the Newton's second law, Force (F) = ma (or) F = mgHere, F = force acting on the knuckle.
m = mass acting the knuckle.
a = acceleration.
g = gravitational acceleration (9.81 m/s as standard value for G force)Loading Condition on Knuckle,
For the calculation of load acting on steering knuckle component, the required loading conditions are considered as mentioned in the table below,

Table 4.3 Loading Conditions

LOADING CONDITIONS	FORCE CONSIDERATION
Braking force	1.5mg
Lateral force	1.5mg
Steering force	40-50 N

Force on the knuckle hub in X -axis	3mg
Force acting on the knuckle hub in Y-axis	3mg
Force acting on the knuckle hub in Z-axis	1mg

Various iteration based on the gravitational acceleration acting on the component are as follows,

Multiaxial Load Cases

The loads act on the steering knuckle connection point during various riding conditions emanating from those generated at the tire-road contact surfaceand varying with vehicle specification. Load cases for the three conditions namely cornering, acceleration, breaking. These loads are to be applied to the finite element model using the global vehicle coordinate system to get the more reliable results.

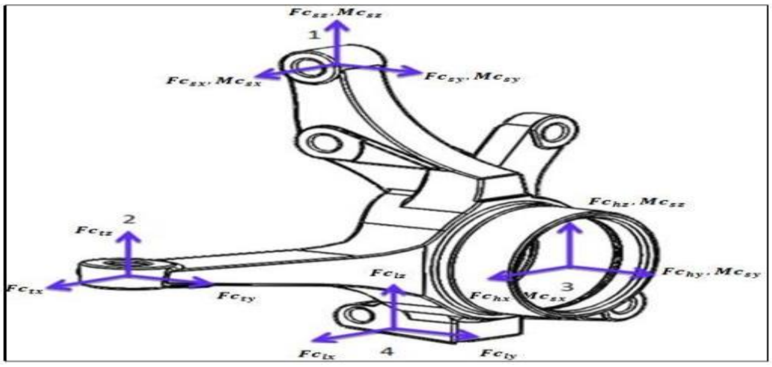


Figure 5.1 multi axial load cases forces at x, y & z directions

The details of load cases are discussed above. The direct of forces acting on the knuckle also vary in x, y &z directions at connections points 1, 2, 3, 4, 5& 6 as shown in figure.

MATERIAL SELECTION

Selection Of Material

Materials are made up of matter. Materials are anything that have weight and occupy some space. The field of engineering is concerned with the optimumuse if materials. The engineering are required to select the most suitable materialsfor their job. There are two types of metals:

Ferrous Metals

- Ferrous metals
- Non-ferrous metal

The metals, which contain iron as their main constituent, are called ferrous metals. Cast iron, wrought iron, pig iron and steel are some ferrous metals.

Non-Ferrous Metals

The metals, which contain a metal other than as their main constituent, are called non ferrous metals. Aluminum, copper, zinc, lead, brass and tin are some non-ferrous metals.

In this project the commonly used cast iron type materials are used. Generally More than 90% of the human beings are used ferrous materials for their general purpose tasks. Ferrous materials can be produced very economically andstrengthened.

5.1 Material Used (Grey Cast Iron)

5.1.1.1 Grey cast iron is the least expensive and the most common type ofcast iron.

5.1.1.2 It is an alloy of carbon and Brass with iron.

6. DESIGN ANALYSIS

6.1 MATERIAL PROPERTIES

There are several materials used for manufacturing of steering knuckle component such as Grey cast iron is used here as a preferred material, it is widelyused. Thus properties of the grey cast iron are illustrated in the following figure.

I.Analyzing Data

Outline of Schematic A2: Engineering Data

	A	B	C	D
1	Contents of Engineering Data		ource	Description
2	Material			
3	Gray Cast Iron			
*	Click here to add a new material			

Properties of Outline Row 3: Gray Cast Iron

	A	B	C	D	E
1	Property	Value	Unit		
2	Density	7200	kg m^-3		
3	Isotropic Secant Coefficient of Thermal Expansion				
4	Coefficient of Thermal Expansion	1.1E-05	C^-1		
5	Reference Temperature	22	C		
6	Isotropic Elasticity				
7	Derive from	Young's Modulus and Poisson's Ratio			
8	Young's Modulus	1.1E+11	Pa		
9	Poisson's Ratio	0.29			
10	Bulk Modulus	8.7302E+10	Pa		
11	Shear Modulus	4.2636E+10	Pa		
12	Tensile Yield Strength	2.4E+08	Pa		
13	Compressive Yield Strength	9.7E+08	Pa		
14	Tensile Ultimate Strength	3.1E+08	Pa		
15	Compressive Ultimate Strength	8.2E+08	Pa		

II. Materials

Component	Material	Safety Factor
Body3	Steel	Yield Strength
Body2	Steel	Yield Strength
Body1	Steel	Yield Strength
Body4	Brass	Yield Strength
Body5	Steel	Yield Strength

III. Brass

Density	8.490E-06 kg / mm^3
Young's Modulus	97000.00 MPa
Poisson's Ratio	0.31
Yield Strength	124.00 MPa
Ultimate Tensile Strength	338.00 MPa
Thermal Conductivity	0.115 W / (mm C)
Thermal Expansion Coefficient	2.050E-05 / C
Specific Heat	380.00 J / (kg C)

IV. Steel

Density	7.850E-06 kg / mm^3
Young's Modulus	210000.00 MPa
Poisson's Ratio	0.30
Yield Strength	207.00 MPa
Ultimate Tensile Strength	345.00 MPa
Thermal Conductivity	0.056 W / (mm C)
Thermal Expansion Coefficient	1.200E-05 / C
Specific Heat	480.00 J / (kg C)

B. Study Report by Autodesk Fusion 360

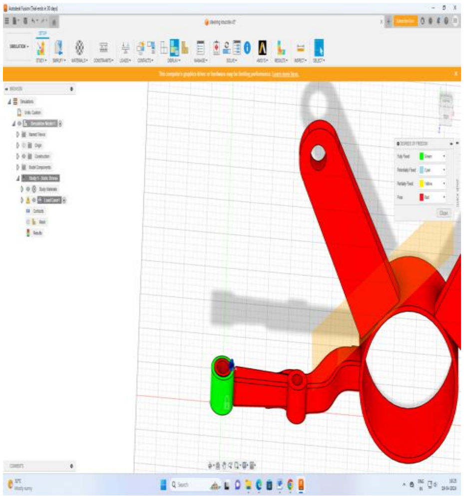


Fig6.1 Stress Analysis of knuckle Bush

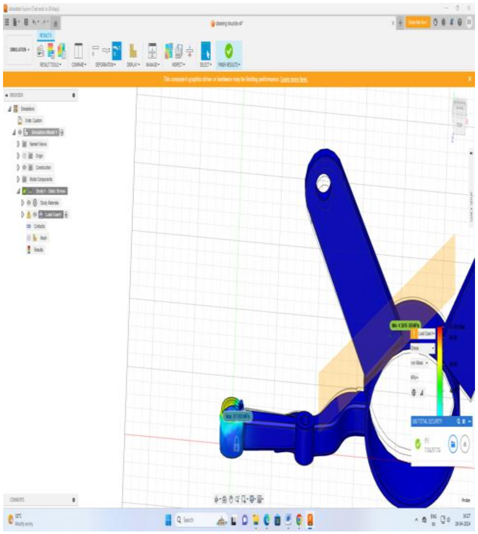


Fig6.2 Equivalent Stress

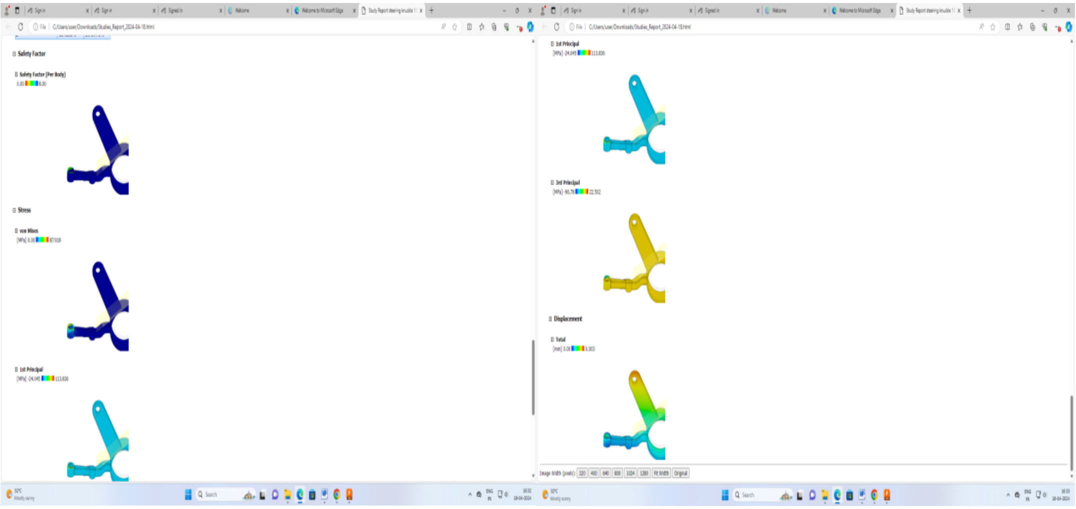


Fig6.3 Model displacement

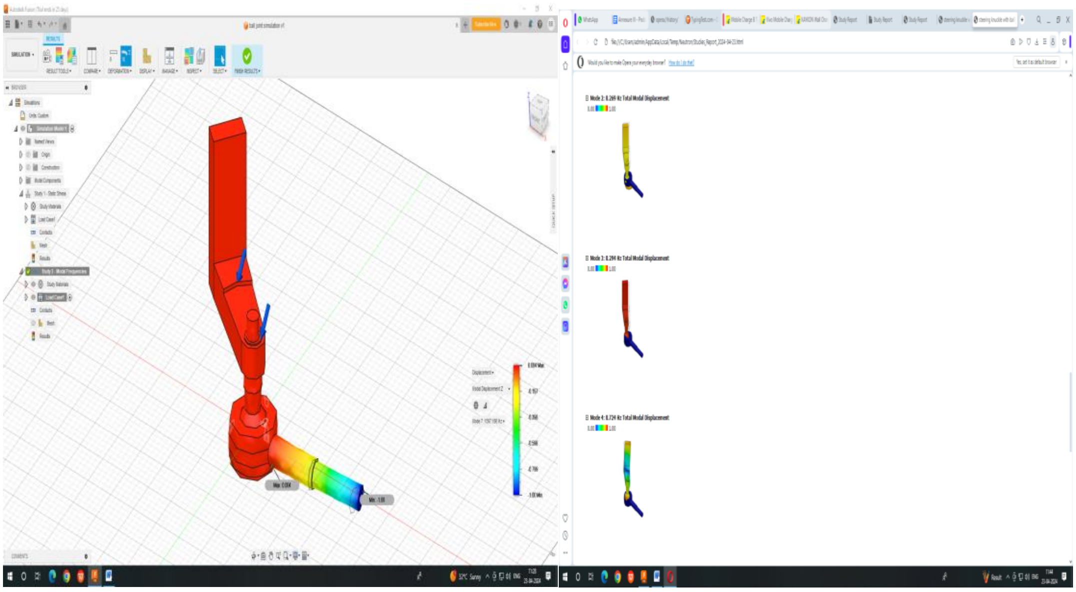


Fig6.4 Stress Analysis of ball joint with Bush

RESULT

The previous model of steering knuckle with ball joint component which is made up of EN 8 have max yield strength 647MPa and mass is about 3.62kg.

- The maximum and minimum total deformation obtained in the Analysis 22456mm and 6101mm.
- The maximum and minimum Equivalent Stress obtained in the Analysis are 5.9216Mpa and 2.2451e - 003MPa.
- The maximum and minimum Shear Stress obtained in the Analysis 2.3462MPa and -2.0968MPa.

CONCLUSION

They Design shape optimization is used in this study for reducing the vibration in a knuckle by 15.23%.It contains Maximum stress and deformation are with a limits. Due to shape optimization gives minimum modifying in the deformation. It has that changing the volume and a shape doesn't any influences significantly to load analysis of the structure. Then finally, the overall weight of the vehicle can be reduced to achieve savings in costs and materials to avoid the oval shape of connected ball joint end, as well as, improve efficiency, to better handling and reduce the vibration in steering wheel and knuckle with ball joint.

REFERENCESs

1. Tagade, P. P., Sahu, A. R., &Kutar mare, H. C. (2015). Optimization and Finite Element Analysis of Steering Knuckle. In *International Journal Of Computer Applications International Conference On Quality Up-Gradation In Engineering ,Science And Technology(ICQUEST2015),(09758887)*
2. AmeyaBhusari,AdityaChavan,SushrutKarmarkar,"FEM&OptimisationofsteeringknuckleofATV",IRFInternationalConference,11thOctober2015,Pune,India,ISBN:978-93-85832-16-1.
3. Kamlesh LalasahebChavan, Sanjay Deodas, SwapnilShashikantKulkarni, "Mass Reduction For Steering Knuckle Armin a Suspension System Through Topology Optimization in CAE", International Journal of Advanced Engineering Research and Studies E-ISSN2249–8974/IV/I/Oct.-Dec,2014/34-35.

4. sMahendra L. Shelar and H. P. Khairnar, “Design Analysis and Optimization of Steering Knuckle Using NumericalMethods and Design of Experiments”,International Journal of Engineering Development and Research, 2014 IJEDR |Volume 2,Issue3|ISSN:2321-9939.
5. Patil, M. A., Chavan, D., &Ghorpade, M. K. U. S. (2013). FEA of Tie Rod of Steering System of Car. InternationalJournalofApplicationorInnovationEngineeringandManagement,2(5),222-227.
6. Chandrakar, S. K., Soni, D. L., &Gardia, S. (2013). FEA of A Steering Knuckle for Life Prediction. InternationalJournalofEngineering,6(5),681-688.
7. Shinde,D., &Kalita, K.(2006).FEANALYSISOFKNUCKLE JOINT PINUSEDINTRACTORTRAILER.
8. SS,M.A.V.K.P.,&Harugade,P.P.M.DevelopmentalDesignandOptimizationofSteeringKnucklewithIntegrated Spindle.
9. Teja,G.P.,Chandu,K.V.P.P.,Krishna,C.R.,&Sreeram,K.Y.(2016).weight optimization of steering knuckle joint using fea.