

Design of Suspension Ball Joint Using FEA and Experimental Method

Jitendra Shinde¹, Sunil Kadam²

¹P.G. Student, Department of Mechanical Engineering, Bharati Vidyapeeth's College of Engg, Kolhapur, India

²Associate Professor, Head of Mechanical Department, Bharati Vidyapeeth's College of Engg, Kolhapur, India

Abstract - This study describes the design and analysis of a suspension system ball joint using finite element analysis and experimental method. The axis of the ball joint element showed a complete fracture which occurred midway between the top and bottom section changes of the element. So there should be something else supports car when the ball-joint fails. This research has a modification of this ball joint for this purpose. Design of ball joint is modified and stress raisers are removed from the previous design of ball joint. Also validate that software results by experimental method. Design of the present ball joint has been improved and from obtained results we can say that the design of ball joint is modified and within safety limit.

Key Words- Ball joint, Modelling, Stress and Fatigue analysis, Design modification, experimental method

1. INTRODUCTION

Ball joint is an important part of car suspension system. Ball joints act as the pivot point between two parts: the suspension and car's tires. Ball joints help support car's weight. The ball joint is one moveable part of a control arm assembly. It is steel bearing stud and socket enclosed in a steel casing. The socket enclosed in steel casing is connected to the control arm. The bearing stud is tapered and threaded so that it fits into a tapered hole in the steering knuckle and the latter connects the tire. This connection needs to be able to rotate horizontally for steering and vertically for shock absorption, hence the use of ball joints that can move in all directions. This projected study is an overall revive of existing ball joint failure and to minimize this failures by improving some failure parameters. So that it will helps to maximize life of ball joint and resulting minimizing road accidents. [1]

1.1 Problem Definition

The failure of ball joint is due to the fracture initiated at the contact point between the ball joint element and its cage where stress concentration creates. Further the reduction of the cross section on the ball joint element leads to stress concentration which further reduces the life of the element. In order to reduce the contact stresses on the ball element

change of the geometric design of the element is necessary. To minimize the failure of suspension ball joint by improving geometrical design of suspension ball joint to achieve maximum performance. [3]

1.2 Methodology

To solve the problem mention above we have first import the Para solid model to the ANSYS which is analysis software by using which we have to solve the problem. In the next step we have to do the meshing on ball joint. Next step we have to apply loads i.e. force and torque which are mentioned in the load case then after we have to apply the boundary conditions by seeing the actual model i.e. where it is fixed. Then we have to solve the problem by using software. After completion of the solution by using software we have to do the physical testing of suspension ball joint.

1.3 Design tool

Various tools were used in the development of our design. The CATIA software is used for tasks such as the three dimensional solid modeling of suspension ball joint. The use of solid modeling techniques allows us the computerization of many difficult engineering calculations that are carried out as a part of the design process. Simulation, planning, and verification of processes of the components of the suspension are some of the tasks that will be performed in CATIA. Also ANSYS software will be used to perform an engineering analysis of the stresses acting on component to reveal the state of stress and failure on the device.

2. MODELING AND ANALYSIS OF BALL JOINT

The measurements of main parts of ball joint have been made in order to create a three dimensional geometry by CATIA software. Then, the model (geometry) has been imported to ANSYS workbench software.

Initially the 3D drawing of ball joint is done by using CATIA according to dimensions specified in the Table.

Table No.1: Dimensions of Suspension Ball Joint

| Diameter | Dimension in mm |
|----------|-----------------|
| D1 | 30 |
| D2 | 20 |
| D3 | 15 |

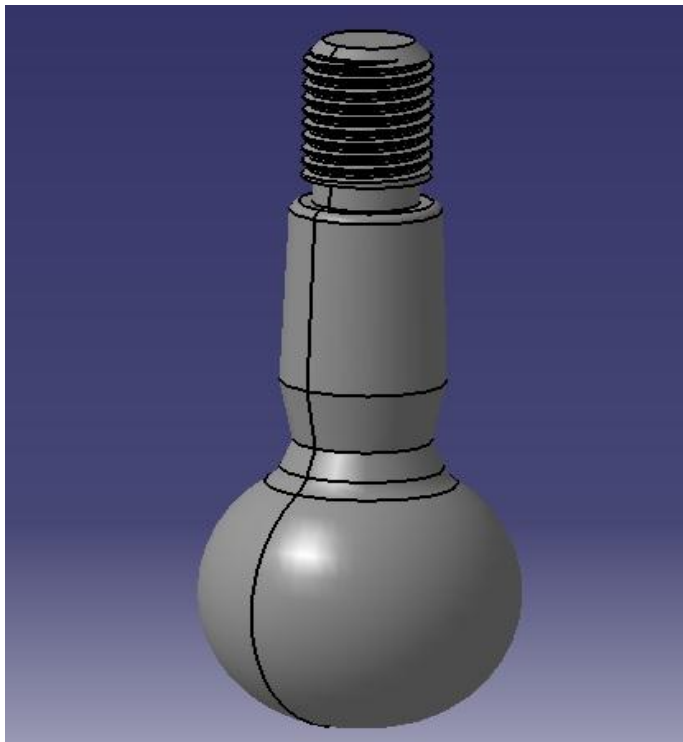


Fig -1: 3D-Model

2.1 Material

According to Spectro test results the ball joint was manufactured using an EN 18D steel alloy.

EN 18D Steel Alloy properties:

- Yield stress=565 MPa
- Ultimate tensile strength=887 MPa
- Density =7.85g/cm³
- Poisson's ratio = 0.29

2.2 Meshing Generation

Meshing of a given model will be done depending on geometry of the model, it is better to have more degrees of freedom hence more number of elements so that results obtained will be closure to analytical results. In two bay panel analyses, crack region is meshed with more number of elements when compared with other parts of fuselage, for obtaining a converged solution which in turn a better solution. [1] The fundamental premise of FEM is that as number of elements (mesh density) is increased, the solution

gets closer and closure, however solution time and compute resources required also increases dramatically as we increases the number of elements to the true solution

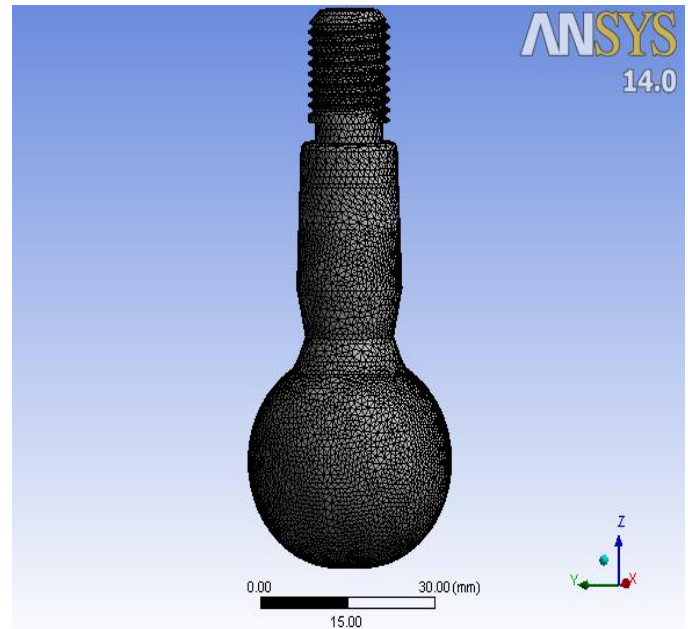


Fig - 2: Meshing of ball joint

2.3 Loading and Boundary Condition

Each tire of this vehicle has a share of the car's weight, gross weight and weights of persons riding the car. The centroid of this car under its full static load is located at the a point of intersection of its diagonal axes, consequently, each tire has a share of quarter of that full static load, If the car has a gross weight of 2300 kg and the weight of its passengers and luggage is 620 kg as well as each passenger is assumed to weight of 100 kg, the total static load will be 2300 kg .So each tire will have a share of 575 kg. when the a car has an impact within the motion, ball joint subjects to longitudinal load equals nearly to double of its tire share static load as well as horizontal load equals to that longitudinal one but it directs a horizontally along the length of car.

A load of (2X575X9.81)N is applied downward to stud at the thread place in its longitudinal direction while other load of(2X575X9.81)N is applied horizontally at a place where it is in contact with knuckle, also a general joint is added to this region to keep only linear motion. A fixed boundary condition is applied to the socket at a place where it's bonded to the lower control arm.

The fatigue stress and its safety factor have been created in these two forms of numerical analysis, the load on ball-joint is fluctuating between its maximum (where the ball-joint faces an impact) and the static load which only includes the weights of car, freight and persons.

Calculations for weight distribution on each wheel:

Kerb Weight: 1680 Kgs

Seating Capacity: 7

Gross Weight = Kerb Weight + Passenger weight + Luggage weight

WG = 2300 Kgs

From standard

The ratio of weight distribution is F/R: 49/51

Weight acting on each front wheel

$$W = (0.50 \times WG) / 2$$

$$= (0.50 \times 2300) / 2$$

$$W = 575 \text{ Kg}$$

So, the reaction force,

$$P = WXg$$

$$= 575 \times 9.81$$

$$P = 5640.75 \text{ N}$$

$$P = 5700 \text{ N Appx.}$$

Assumed condition is of bump, in which the extremity of load is present.

The load considerations are Double Gravity (2g acceleration)

$$P = 11400 \text{ N (taken for Analysis)}$$

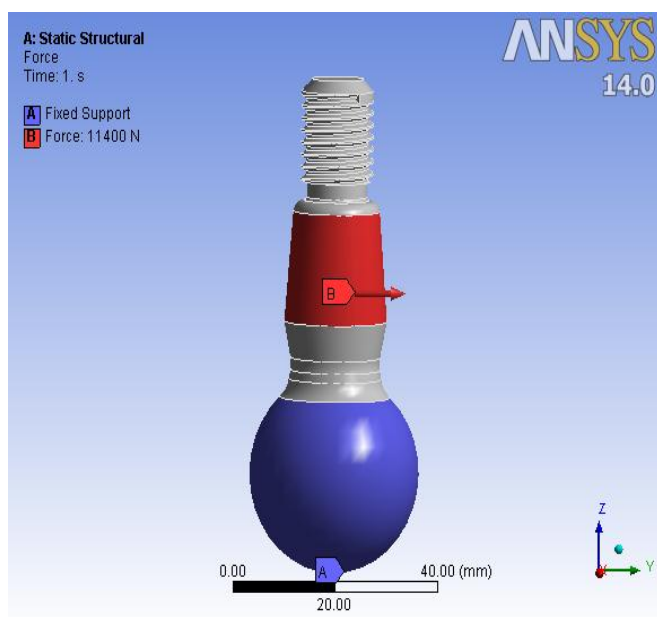


Fig- 3: Loading and Boundary Condition

3. RESULTS FOR ORIGINAL GEOMETRY

From the above loading and boundary condition following results are obtained.

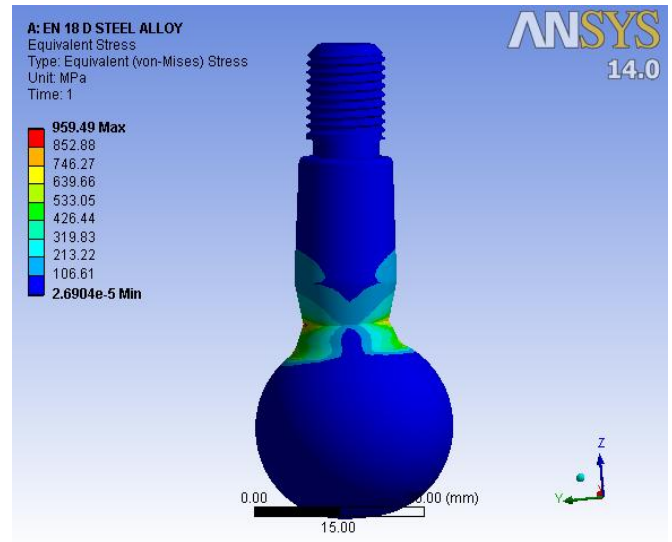


Fig.4- : Equivalent Von-Mises stress

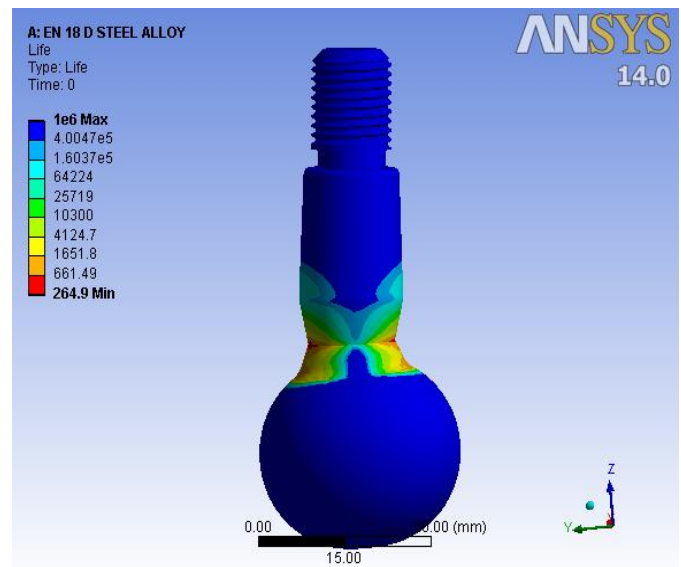


Fig - 5 : Fatigue life

It is since observed that from the results, We came to know that, equivalent von-mises stress goes beyond ultimate tensile strength of the material and hence there is crack initiation in the ball joint at the specified loading conditions and that crack leads towards the fatigue failure of the ball joint as the ball joint is under cyclic loading as per the description given in the problem definition. After the knowing results, it is need to increase diameter of ball joint at failure section, from 15 mm to 17.5 mm and revise the design to get the desired results.

4. RESULTS FOR MODIFIED BALL JOINT

After design modification improved results for ball joint shown below.

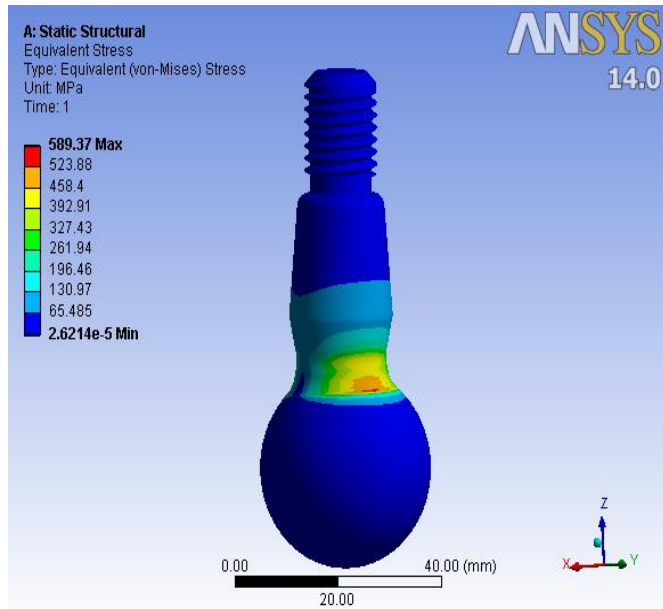


Fig- 6 : Equivalent Von-Mises stress

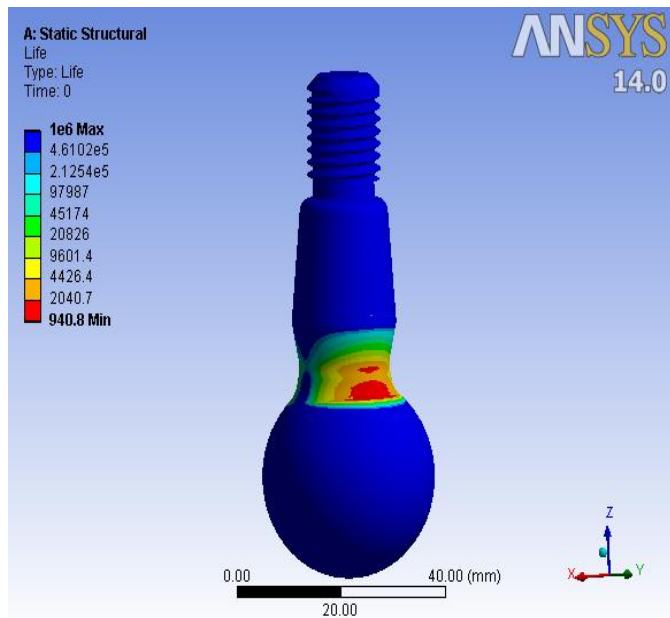


Fig - 7 : Fatigue life

Result of stress and fatigue analysis of modified ball joint shows that stress value has been brought to the safety limit. Hence design of ball joint is modified and stress raisers are removed from the previous design of ball joint.

4. COMPARISON OF ANALYSIS RESULTS

Table No.1: Comparison of Results

| SR. NO. | COMPONENTS | BEFORE MODIFICATION | AFTER MODIFICATION |
|---------|--------------------------------|---------------------|--------------------|
| 1 | Total Deformation | 2.1 MM | 0.2 MM |
| 2 | Equivalent Stress Plot | 959.49 MPa | 589.37 MPa |
| 3 | Major Principal Stress Plot | 916.64 MPa | 493.75 MPa |
| 4 | Equivalent Elastic Strain Plot | 0.0048 | 0.0111 |
| 5 | Max Shear Stress | 524.38 MPa | 339.65 MPa |
| 6 | Factor of Safety | 0.9289 | 1.505 |
| 7 | Fatigue life | 264 (min) cycle/s | 941 (min) cycle/s |

5. EXPERIMENTAL RESULTS

5.1 Spectro Test

After doing spectro test following are chemical composition are obtained and material of the ball joint was found.

Table No 3. : Chemical Composition

| Material | C | Mn | S | P | Cr | Ni |
|--------------------|------|------|-------|-------|-------|-------|
| EN-18D steel alloy | 0.41 | 0.73 | 0.015 | 0.018 | 0.886 | 0.005 |

5.2 Tensile test on Universal testing machine

Objective of tensile test for determine the stress-strain behavior of a material sample and to analyze the results of the tensile test to find the mechanical properties.



Fig.4- : Test Specimen



Fig.4- : Tensile Test Set-Up on Universal Testing Machine

After completing tensile test following result are obtained for EN 18D Steel Alloy.

EN 18D Steel Alloy mechanical properties:

- Yield stress=565 MPa
- Ultimate tensile strength=887 MPa
- Density =7.85g/cm³
- Poisson's ratio = 0.29
- Elongation = 15.33 %

5.2 Shear test on Universal testing machine

During shear test we have first place the test attachment on lower table of universal testing machine and ball joint inserted in shear test attachment, then apply load and got the shear strength of ball joint. Figure shows the test set-up on universal testing machine.



Fig.4- : Shear Test Set-Up on Universal Testing Machine



Fig.4- : Specimen Set-Up on Universal Testing Machine

After completing shear test the shear strength of ball joint was 529.31MPa.

3. CONCLUSIONS

The results for stress and fatigue analysis of ball joint with boundary condition for the previous ball joint tests shows that stress limit was crossed and the stress value is much higher than the ultimate tensile strength of the material, Also because of this there is no safe fatigue life to the ball joint .To modify the design of ball joint to make it safe, reliable and durable. We worked on the stress raiser in the previous design of ball joint and required design modification are done and results of stress and fatigue analysis of modified ball joint shows that stress value has been brought to the safety limit, the component is now have considerably good fatigue life. Hence design of ball joint is modified and stress raisers are removed from the previous design of ball joint. Design of the present ball joint has been improved and from obtained results we can say that the design of ball joint is modified and within safety limit. Also experimental validation of result through finite element analysis was done.

BIOGRAPHIES



Mr. J.G.SHINDE
PG Student
ME(CAD/CAM/CAE)
Bharati vidyapeeth's college
engineering ,Kolhapur

REFERENCES

- [1] Jitendra Shinde,Sunil Kadam, amol patil and samuvel pandit ,“Design Modification And Analysis Of Suspension Ball Joint Using Finite Element Analysis” journal of innovative reaserch in science, engineering and technology, vol. 5, july 2016
- [2] Chen A., “Investigating the behavior of ball joint sealing boots using a 3D Finite Element Model”, 2nd Worldwide Automotive Conference, 2010
- [3] E.A. Ossa, C.C. Palacio and M.A. Paniagua, “Failure analysis of a car suspension system ball joint”Materials Engineering Research Group, Colombia, 2011
- [4] Fischer, I. S., “Numerical analysis of displacement in spatial mechanisms with ball joints”, Mechanism and Machine Theory,35 (2000) p. 1623-1640.
- [5] Ryu YI, Kang DO, Heo SJ, Yim HJ, Jeon JI., “Development of analytical process to reduce side load in strut-type suspension”. J MechSci Technol2010; 24:351–6.
- [6] Suresh, S., “Fatigue of Materials”, 2nd ed., Cambridge University Press, Cambridge, 1998.
- [7] David Gorsich , “Vehicle Systems Engineering and Technology”, Symposium, 2009.
- [8] S. A. Kesulai , “European Journal of Scientific Research” , , Fatigue Life Prediction ISSN 1450-216X Vol.30 No.3 (2009).