DESIGN ANDFAILURE ANALYSIS OF SUSPENSION BALL JOINT USING FINITE ELEMENT ANALYSIS

Jitendra Shinde

P.G. Student, Department of Mechanical Engineering, BV Engineering College, R. K. Nagar, Kolhapur India

Nikhil Jamadade

P.G. Student, Department of Mechanical Engineering, TKIET, Warnanagar, Kolhapur, India

Abstract

This study describes the analysis and investigation of the causes of the sudden failure of a Mac Pherson strut suspension system ball joint. The axis of the ball joint element showed a complete fracture which occurred midway between the top and bottom section changes of the element. It is seen the ball joint of this car fails suddenly without any sign of consumption and this case is a dangerous as well as disturbing factor for traffic and driver. 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. The modification has depended on the results obtained from the numerical analysis of the loaded ball-joint in two situations. Measurements of parts of ball joint of Nissan pickup car have been made and these parts have been drawn in two forms (situations) by using CATIA software imported to ANSYS WORKBENCH software where a three- dimensional model was created and maximum load is applied to ball-joint.

Keyword: Ball joint, Suspension system, Fatigue, Contact stress, analysis

Introduction

Ball joints are used on the front end of almost every car, truck and minivan. 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 and, as is the case with some vehicles, ball joints may be used to help set the alignment. 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. Ball joints are a critical part of any car's suspension and steering. They attach the wheel hub, which the wheel and tire are mounted to the rest of the suspension. 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. While ball joints last for a long time, they do wear. The polished metal ball rides in a polished metal cage. Space between the two is filled with grease to reduce wear. However, if the grease leaks out of the ball joint or any dirt and impurities get into the grease, the ball joint may become worn or damaged. There is a rubber boot over the joint to help keep dirt out, but that does not mean that there is no way in which dirt can enter. In an automobile, ball joints are spherical bearings that connect the control arms to the steering knuckles. They are used on virtually every automobile made and work similarly to the ball-and-socket design of the human hip joint. A ball joint consists of a bearing stud and socket enclosed in a casing; all these parts are made of steel. The bearing stud is tapered and threaded, and fits into a tapered hole in the steering knuckle. A protective encasing prevents dirt from getting into the joint assembly. Usually, this is a rubber-like boot that allows movement and expansion of lubricant. Motion-control ball joints tend to be retained with an internal spring, which helps to prevent vibration problems in the linkage. A ball joint is used for allowing free movement in two planes at the same time including rotating in those planes. Combining two such joints with control arms enables motion in all three planes, allowing the front end of an automobile to be steered and a spring and shock (damper) suspension to make the ride comfortable. The front suspension system ball joint fails due to many of the parameters so it is necessary to study the exact reasons for the failure of ball joint in this it intended to observe these failures with the different tests. 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.

Related Work

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. Before going on with description the model, there is a need to know two terms:-

- Travel: - Travel is the measure of distance from the bottom of the suspension stroke (such as when the vehicle is on a jack and the wheel hangs freely) to the top of the suspension stroke (such as when the vehicle's wheel can no longer travel in an upward direction toward the vehicle).
- Angle of ball joint:- is defined as the angle between the longitudinal axis of the stud and the longitudinal axis of socket.

The angle of ball joint changes with the travel. When the car hits a bump, the travel will proportionally increase with the amount of impact [S. A. Kesulai, 2009]. So, this angle must be taken in consideration during the analysis of stresses. Consequently, the stud and socket of ball joint have been drawn by CATIA software with consideration of this angle. So, the main parts of ball-joint must be drawn in two situations (forms):-

1. In the first form, the main parts of ball-joint are drawn normally by CATIA software regardless of joint's angle, then the model (geometry) has been imported to ANSYS workbench software.
2. In the second form, the main parts of ball-joint are drawn by CATIA software under consideration to joint's angle, then the model (geometry) has been imported to ANSYS workbench software.

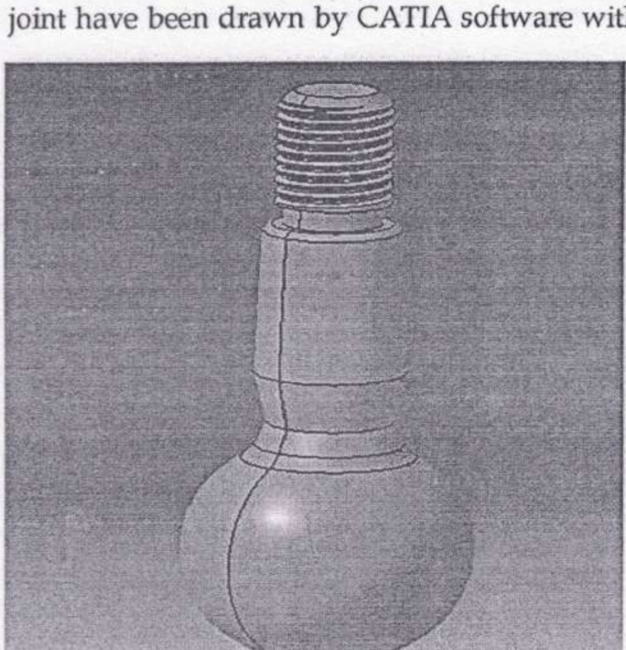

Figure A. 3D-Model

Material

According to chemical composition analysis by OES, the ball joint was manufactured using an EN 18D steel alloy.

A microstructure formed mainly by tempered martensite with acicular grains of ferrite on the grain boundaries. The presence of tempered martensite indicates that the material suffered a heat treatment of quenching and tempering. Despite the beneficial effect of increasing material toughness of acicular ferrite in low carbon steels.

Meshing Generation

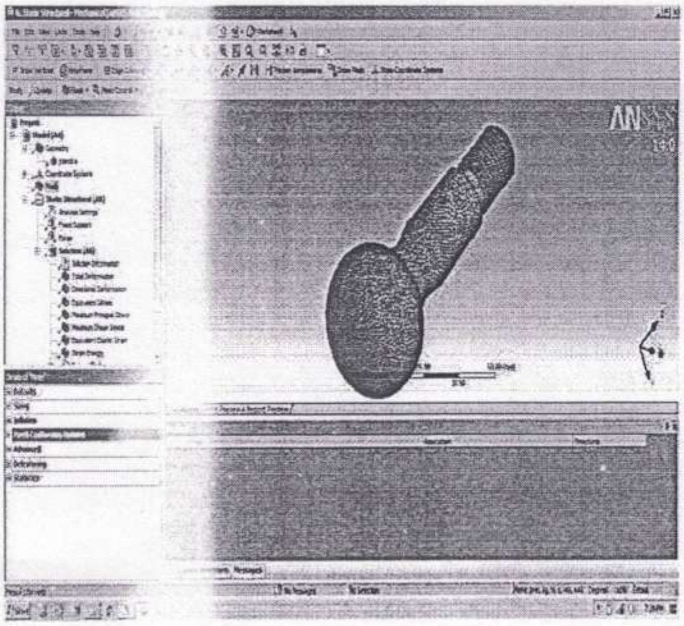

Figure B. Meshing of Ball Joint

The mesh must be homogenous. The elements must have equal sizes as far as possible and their numbers must be enough to get accurate results. Also the elements must be dense at a sensitive region such as the contacts, margins...etc. So, different options for meshing process have been used in each part of the joint to get the desired accuracy. The mesh of each part of the ball-joint is shown in the Fig.

Finite Elements Analysis of the Ball Joint

Finite Elements Analysis (FEA) it is possible to find the locations of high stresses suffered by the analyzed element and their values. In this way it is feasible to draw specific conclusions on the causes and possible solutions to avoid the recurrence of these kind of failures. In this analysis the ball joint was geometrically modeled as shown in Fig C. The loading, boundary and contact conditions of the model are given as:

1. A fixed contact in zone A (ball), shown in blue3 color in.
2. A lateral load, assumed constant in this analysis to simplify the modeling, with a value of 11400N and applied at the end of the element and midway. This load value was used according to the experimental study of Ryu et al. [13], who found a value of load of approximately 11400N for a suspension system similar to the one studied here, so it is considered that this value of load represents a realistic estimate of the load applied to the element. The analysis and calculations of maximum stresses were performed with the support, in order tohighlight the places of higher stresses in the element. The contact support between the axis of the element and its cage was modeled as frictionless because the failed element surfaces did not show any indications of fretting or wear suggesting friction between them. The elements used in the FE mesh were hexagonal with a size of 1 mm. The mesh had a total of 3375 elements. The size of the elements and mesh were selected after performing series of analysis with the same loads and using different mesh sizes. These analyses showed that for elements smaller than 1 mm, the

values of the stresses in the element varied considerably, reaching high values related with singularities caused by small elements. For elements bigger than 1mm the results were not representative as the size of the elements were higher than the contact region.

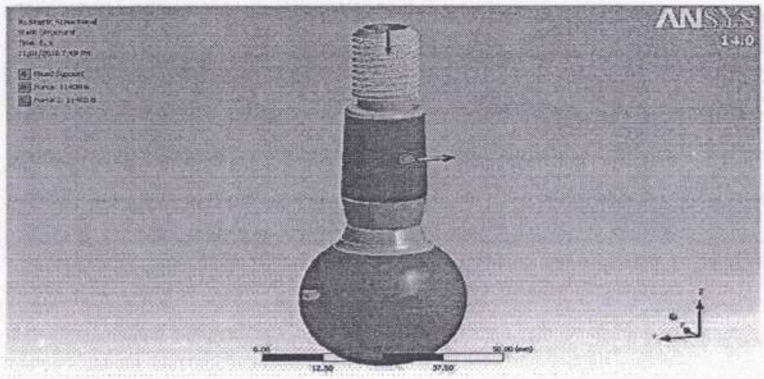

Figure C. Ball joint boundary and loading conditions used in the numerical analysis

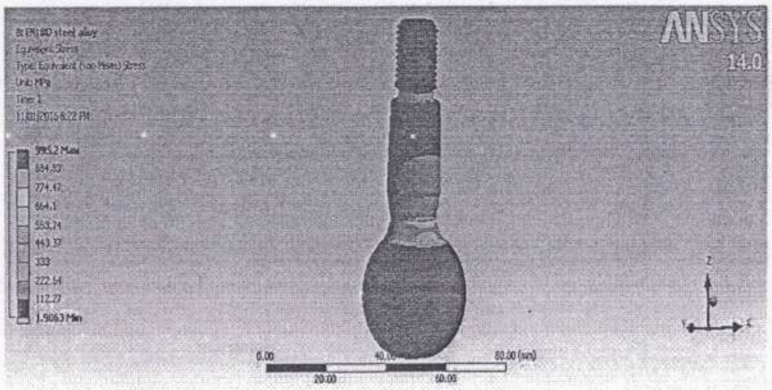

Figure D. Equivalent Von Mises stress distribution of the ball joint with contact support

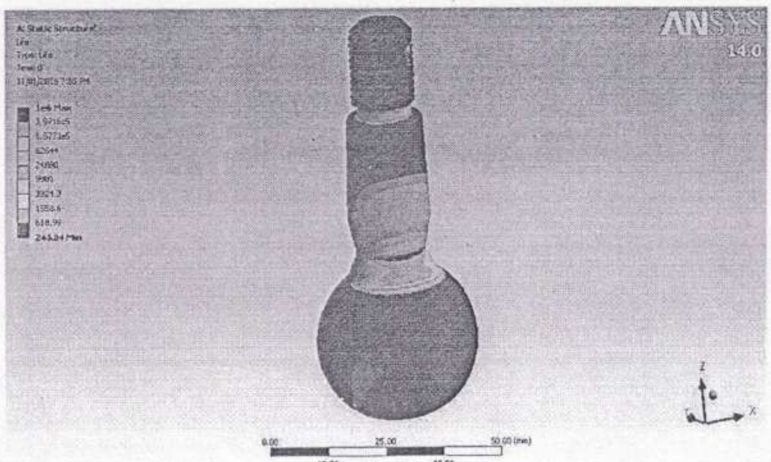

Figure E. Fatigue life of the ball joint

Conclusion

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 .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. Therefore, the causes of the failure of the ball joint are (i) Defective heat treating process; and (ii) Defective geometric design of the element cross section. In order to reduce the contact stresses on the ball joint element, a change on the geometric design of the element was proposed. In the proposed design.(5) It is then suggested to modify the geometry of the ball joint to increase the loading section and reduce the contact stresses with the cage

References

1. Agah Uguz and S. Hakan Oka, Bursa, "Finite Element modeling of ball joints under dynamic loading" Material prüfungJahrg. 46 (2004) 10.
2. Bong-Su Sin,"Process Design of a Ball Joint, Considering Caulking and Pull-Out Strength", Department of Mechanical Engineering, Dong-A University, Busan 604-714, Republic of Korea, 2014.
3. Byung-Hyun Jang and Kwon-Hee Lee, "Analysis and design of a ball joint, considering manufacturing process", J Mechanical Engineering Science, Vol 228, 2014.
4. Chen A., "Investigating the behavior of ball joint sealing boots using a 3D Finite Element Model", 2nd Worldwide Automotive Conference, 2010.
5. 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.
6. Fischer, I. S., "Numerical analysis of displacement in spatial mechanisms with ball joints", Mechanism and Machine Theory, 35 (2000) p. 1623-1640.
7. Fischer, I. S., "Velocity analysis of mechanisms with ball joints", Mechanics Research Communications, 30 (2003)69-78.
8. J. Park, S. Choi, K. Na, and Y. Kim, "Application of FE analysis for optimal design of caulking process", Journal of Materials Processing Technology, vol. 198, no. 1-3, pp. 471-477, 2008.
9. Jaeyoung Kang," Theoretical model of ball joint squeak", Division of Mechanical and Automotive Engineering, College of Engineering, Kongju National University, Cheonan-Si, Republic of Korea, 2011.