Indian Journal of Science and Technology, *Vol 10(31), DOI: 10.17485/ijst/2017/v10i31/113918, August 2017*
ISSN (Print) : 0974-6846
ISSN (Online) : 0974-5645

Finite Element Methods based Structural Analysis of Universal Joint using Structural Steel Material

Gagandeep Singh*, **Sandeep Singh and Rajbir Singh**

Department of Mechanical Engineering, Bhai Gurdas Institute of Engineering and Technology, Sangrur - 148001, Punjab, India; Gagandeepkhokar07@gmail.com, mann.sdeep@gmail.com, ghuman008@gmail.com

Abstract

The present work deals with the optimization of universal joint using structural steel in order to reduce the mass and the stress value using FEM. The universal joint model was prepared in CAD software SOLIDWORKS and the static structural analysis was done in CAE tool ANSYS to determine the equivalent stress (von-mises). First of all, the failure analysis was done which shows that existing universal joint of structural steel was safe below the Torque moment 3850 N-m. Then the structural analysis of the universal joint was done for torque value 3800 N-m. The sensitivity analysis was run to determine sensitive area of universal joint where stresses were high and the sensitive dimensions were changed to find the minimum stress value and mass. For the optimized design, the Equivalent stress (von-mises) reduction in case of structural steel is 15.724% for the torque value of 3800 N-m. The mass of the universal joint reduces by 0.451%. The mass reduction from the cross alone is 4.473%. Final result shows that the stress reduction percentage and total mass reduction percentage comes out to be 15.7 and 0.451 % respectively.

Keywords: Finite Element Methods, Optimization, Universal Joint

1. Introduction

A universal joint is a joint or coupling in a rigid rod that permits the rod to bend in any direction and is normally utilized as a part of shafts that transmits turning movement. It comprises of a couple of yokes found near one another, arranged at 90° to each other, associated by a cross shaft. Universal joints are fit for transmitting torque and rotational movement starting with one shaft then onto the next when their axis are slanted to each other by some angle, which may continually differ under working conditions. Universal joints are consolidated in the vehicle's transmission system to perform three fundamental functions:

- Propeller shaft end joints between longitudinally front mounted gearbox and back conclusive drive hub.
- Rear hub drive shaft end joints between the sprung last drive and the unsprung back wheel stub pivot.
- Front pivot drive shaft end joints between the sprung front mounted last drive and the unsprung front wheel guided stub hub.

Figure 1.1. Universal joint.

Universal joint, generally called a u-joint, is found in numerous car applications and also in different mechanics. For instance, a u-joint is utilized as a part of vehicles between the drive train bar and the transmission or pivot where the bars meet at a right edge. There are three fundamental plans of u-joints, each intended for a particular application.

- Hooke.
- Ring and Trunnion.
- Bendix-Weiss.

* *Author for correspondence*

Ordinary applications of widespread joints incorporate air ship, apparatuses, control systems, electronics, Instrumentation, medicinal and optical gadgets, arms, radio, sewing machines, material machinery and device drives.

1.1 Materials used in Universal Joint

Universal joints differ in view of their material composition, type of hub and the applications for which they are designed. Steel is the most widely recognized material utilized, either in stainless form or alloyed with different metals to handle more noteworthy torque and temperature. Plastics and thermoplastics are frequently utilized as a part of developing universal joints, as this loans more prominent rust and consumption resistance and in addition electrical and attractive protection in applications where this is required. In the present work, structural steel is considered for the universal joint.

1.2 Literature Review

The literature audit of the past work done by different specialists in the field of structure analysis of universal joint utilizing FEM is displayed underneath. We have exhibited a survey of the important study keeping in mind the end goal to gather more data with respect to our work. Following is the writing audit of a portion of the pertinent work done before:

In[1] did a re-design of a universal coupling which will permit power transmission between two misalign hub. A methodology is made for new universal coupling designs which could permit the engineer to transmit the movement of the driver shaft to drive shaft at misalign axis (more than one axis). In their work, 3-D model of the universal coupling is set up by utilizing CAD programming for modal and stress analysis. A finite element based enhancement is utilized to optimize the configuration of Universal coupling. The forces are applied to the displayed universal coupling in CAD software for analysis. These powers and minutes are utilized as the boundary conditions for finite element model of the universal coupling. In[2] watched that because of the headways in PC helper planning and investigation innovations, it is conceivable to outline better couplings with highly enhanced failure rate. In this paper, they direct static basic analysis on a universal coupling utilizing CAE software and study the different stresses and strains created in the joint. In[3] took a summed up case for study while seeking after exposition deal with their subject and tossed light on the causes, area and extent of stresses on the parts in the sub-gathering. Ordinarily, the yokes and the web were demonstrated utilizing Finite Element Techniques. Numerical treatment was offered to estimate reactions to the significant parameters being observed. The benchmark variation was put to test for experimentation for assessing results about towards acceptance. Design alternatives could deliberate on variations for material and/or geometry. Change is looked for in stress distribution without radial change in the current design. In[4] considered TATA 1210 model for examination. It is seen in their work that the failure of yoke happened because of the different stresses initiated which are contemplated for analysis of failed yoke. Numerical analysis of universal joint is done using Sotherberg theory. For FE analysis, CAD model of universal joint is modelled utilizing Pro-E and this model is imported in ANSYS where stress analysis is performed by FEM. FEM have been performed by different parameters. From the yield of this analysis, it is found that results got are in close concurrence with each other and max stress is concentrated at shaft and external surface of burden. In[5] portray the stress distribution of the universal joint utilizing Finite Element Analysis. The Finite element analysis is performed utilizing Computer Aided designing (CAE) tool. The principle target of this thesis was to research and investigate the stress distribution of universal yoke at the real engine condition amid power transmit and decrease of weight by altering the dimensions. The yoke was actualized in the Automobile named Tata 407. Regardless of all the stress in the yoke is not harmed because of high tensile strength but rather it might come up short under fatigue. Accordingly, it is imperative to decide the basic range of concentrated stress for proper alteration. The model is outlined in CATIA programming and the Finite element analysis is performed utilizing ANSYS Workbench. In[6] in their work analyzed the stress generation, deformation and strain in universal coupling. Circumferential stress is applied at the yoke slot and furthermore on the hub and simulated separately. The simulation is completed with the assistance of SolidWorks 2010. To demonstrate the impact of temperature ascend because of friction at the yoke slot, thermal load is gradually expanded at the slot. It has been demonstrated that friction between yoke slot and hub can expand the thermal stress, which can in the long run lead to failure of yoke or hub material. It is additionally found that the hub encounters a bigger anxiety contrasted with the burden when stacked under same weight. Accordingly, the hub has the higher likelihood to fall flat than the yoke. Toward the end of the paper, a few suggestions with respect to universal coupling building material and lessening of friction have been made. Last results acquired are very precise and fit in with the physical and stacking conditions. In[7] played out an analysis on the universal joint and the propeller shaft. In the universal joint yoke, certain alterations are made in the current geometry and analyzed for the indistinguishable loading and boundary conditions as in the reference paper from which the subject has been taken. If there should be an occurrence of propeller shaft a near study has been made between two shafts differing in their material, keeping in perspective the conceivable mass reduction that can be acquired without influencing the function ability of the shaft. Both the parts are analyzed in ANSYS and the outcomes are compared. In[8] in his study completed fracture analysis of universal joint yoke and a drive shaft of a vehicle power transmission framework. Spectroscopic examinations, metallographic investigations and hardness estimations are likewise done for every part. For the assurance of stress conditions at the fizzled segment, stress analysis is additionally completed by the limited component strategy. Some normal explanations behind the failures might be the manufacturing and design faults deficiencies, upkeep issues, raw material issues and also the client began shortcomings. FEM examination of widespread coupling with the assistance of ANSYS for various torque or load condition is exhibited in this paper and it is confirmed by manual estimation. In[9] presented another system which is intended for the transmission of power between two crossing shafts. They worked out that the system comprises of one drive body and one driven body; six aide arms and three associating arms. The converging edge between the info body and the yield body that are coupled to information and yield shafts, can be changed up to 100 degrees while the speed proportion between the two shafts stays steady. Their concentrate additionally incorporates a kinematic analysis and a recreation utilizing Visual NASTRAN, Autodesk Inventor Dynamic and COSMOS Motion. The softwares demonstrated that this instrument can transmit consistent speed proportions at all points between two shafts. By contrasting the graphs of explanatory investigation and simulation analysis, legitimacy of conditions was demonstrated. At long last, by creation and assessment of the component it was demonstrated that this system can transmit steady speed for all intents and purposes. In[10] concentrated on that force transmission arrangement of vehicles comprise a few segments which here and there experience failures. Some basic explanations behind the disappointments might be manufacturing and design shortcomings, upkeep flaws, crude material deficiencies, material processing faults and in addition, the client started issues. In their study, they did the fracture analysis of a universal joint yoke and a drive shaft of a vehicle power transmission system. Spectroscopic analysis, metallographic examinations and hardness estimations are additionally completed for every part. For the determination of stress conditions at the fizzled segment, stress analyses are additionally completed by the Finite element methods. In[11] present the shape enhancement of an automotive universal joint, by all the while considering fabricating cost, extreme drivable joint angle and part volume. In their research, universal joint designs are broke down and analyzed utilizing a weighted entirety of three target capacities: Minimization of machining cost, augmentation of bordering shaft joint point and minimization of aggregate part volume. Part demonstrating and analysis is directed utilizing the Finite Element Analysis bundle ANSYS and optimization is executed utilizing MATLAB. The outcomes show Pareto boondocks for both the flange and weld yoke, developed utilizing the Adaptive Weighted Sum method. These frontiers unmistakably delineate the exchange off between machining cost and joint point; that is, to build the joint angle, a relating increment in the expense of the part is required. It has been demonstrated that boost of drivable joint angle requires a concurrent increment in machining expense of 4.4% and 2.7% for the flange and weld yoke, separately.

2. Objectives of the Present Work

A structural analysis of the universal joint under the loading conditions similar to practical loading conditions is helps to determine the critical region where maximum stresses and deformations occur. This analysis also helps to bring in the modifications in the design of the universal joint in order to reduce the load impact and the reduction of the weight if possible. The objectives of present work are:

- Study the structural behavior the universal joint under the loading conditions of torsional moment.
- Identification of the maximum torque value by failure analysis of structural steel universal joint in CAE tool ANSYS at which it fails.
- Identification of the sensitive part of the universal joint by performing structural analysis applying the maximum torque at which the subject is safe. The sensitive part refers to the region with maximum equivalent stress.
- Modifications in the sensitive and non-sensitive parameters of the universal joint in order to optimize the universal joint design as an approach to obtain maximum stress and mass reduction.
- Identification of new high torque value up to which the optimized structural steel universal joint is safe. This procedure will verify the optimization of the universal joint

3. CAD Modeling

The CAD models of the universal joint and its assembly are made in SOLIDWORKS. The structural steel was the material used for the universal joint. The mass of the each part of the universal joint assembly is given below:

Yoke = 8.5679 X 2 = 17.1358 Kg.

Cross = 1.9159 Kg.

Total weight = 19.052 Kg.

The drawings of the universal joint cross and yoke are given below:

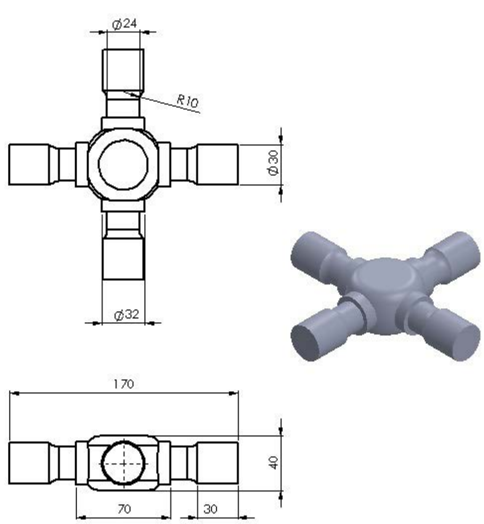


Figure 4.1. Orthogonal and isometric view of cross of universal joint.

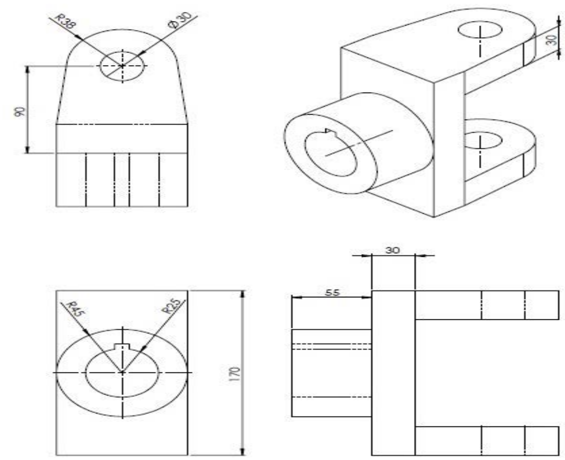


Figure 4.2. Dimensions of the yoke.

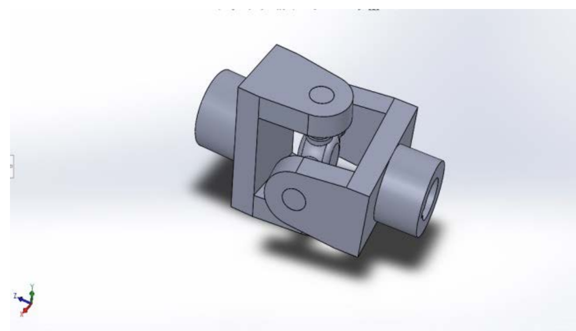


Figure 4.3. Universal joint assembly.

4. Static Structural Analysis

The structure examination of universal joint was performed in the ANSYS 14.5. Mechanical properties of structural steel and boundary conditions are given below;

Table 5.1. Properties of structural steel

Property	Value
Density	7850 kg/m^3
Young's modulus	2 E+ 05 MPa
Poisson's ratio	0.3
Tensile yield strength	250 MPa
Ultimate tensile strength	460 MPa

The assembly of the universal joint was imported into ANSYS first and material was assigned and meshing was done. The number of nodes and elements were 130441 and 79110 respectively.

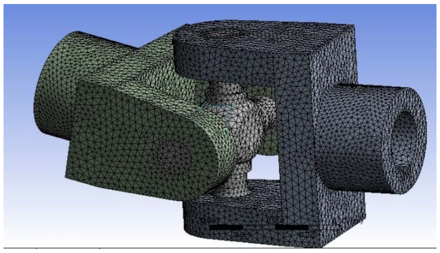


Figure 5.1. Meshing.

Boundary conditions

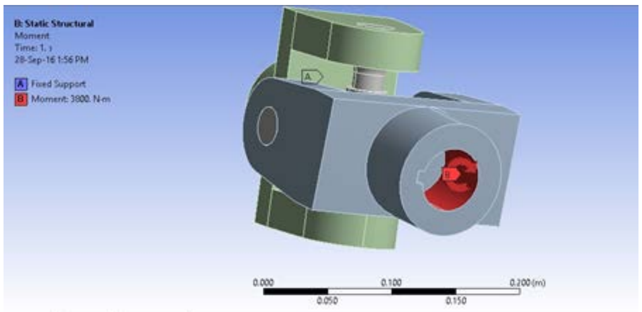
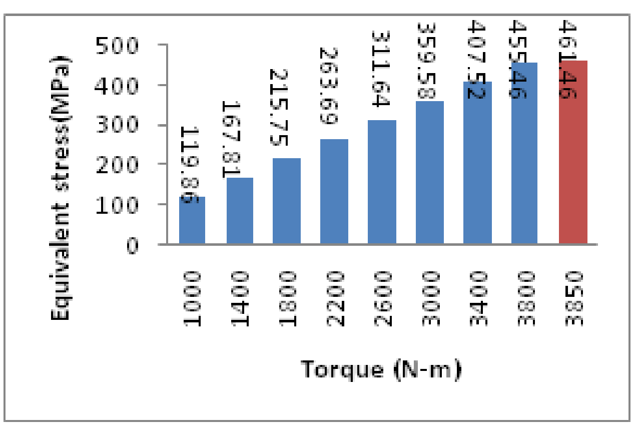


Figure 5.2. Torque applied at hub.

The hub of one of the yokes was fixed and the torque was applied at the other hub. The cross was kept constrained with the reference to both the yokes. The value of the torque was determined from the failure analysis.

4.1 Failure Analysis

The failure analysis of the universal joint was run to determine the torque at which the universal joint structure would exceed the ultimate tensile strength of the material used for universal joint. The different values of the torque were assigned to the subject under the same boundary conditions. The graph 1 shows the equivalent stress values vs the torque assigned to the universal joint.



Graph 5.1. Torque vs. equivalent stress (von-mises).

It is found that the universal joint structure tends to fail just at and above the torque value equal to 3850 N-m. So further analysis was done at maximum torque at which the subject was safe i.e. 3800 N-m

4.2 Structural Analysis of Universal Joint at 3800 N-m

The structural analysis of the universal hub was performed at 3800 N-m and the materials assigned were structural steel. The equivalent stress (von- mises) and deformation occurred are given in the following table:

Structural steel

Table 5.1. Stress and deformation for SS

Equivalent stress (von-mises)	Deformation
455.47 MPa	0.85451 mm

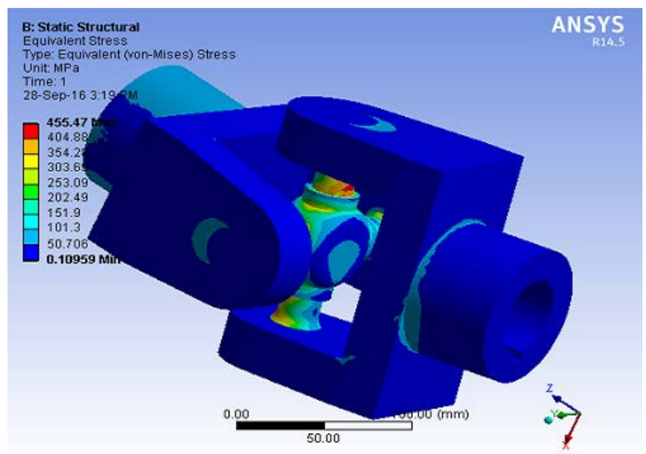


Figure 5.3. Structural analysis of structural steel.

5. Optimization of Universal Joint

From the structural analysis of the universal joint at 3800 N-m torque, it was found that the maximum stress occurred at the universal joint cross. So the sensitivity analysis was run in which the sensitive and non-sensitive parameters were changed and results were collected for each change. The sensitivity analysis was run in order to reduce the mass and stress value. The figures 3.4 and 3.5 show the changed parameters of original and optimized design of universal joint. The stress values and the mass of the optimized universal joint cross are lower than the original design.

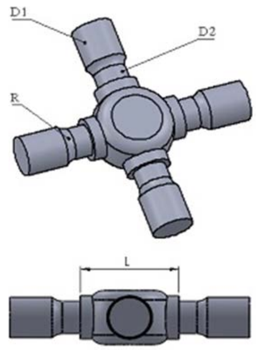


Figure 6.1. Original universal joint.

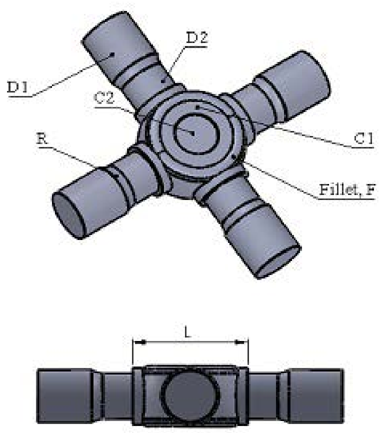


Figure 6.2. Optimized universal joint.

The following two tables show the comparison of dimensions and stress and mass values for both the original and optimized design of the universal joint cross.

Table 6.2. Comparison of parameters

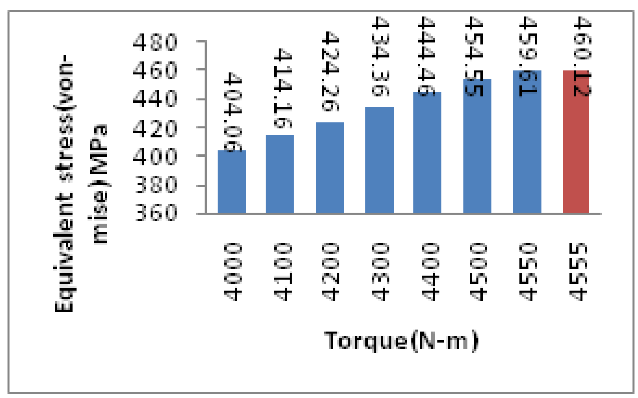
Calculated Entities	Original Design	Optimized Design
Equivalent stress (von-mises)	455.47 MPa	383.85 MPa
Deformation	0.85451 mm	0.77279 mm
Mass of Universal joint	19.052 kg	18.966 kg
Mass of cross	1.9159 kg	1.8302 kg
Percentage stress reduction	_	15.724%
Percentage mass reduction in Cross	_	4.473%

Table 6.1. Comparison of equivalent stress (von-mises) and mass values

Parameters	Original Design	Optimized Design
D_1	30 mm	30 mm
D_2	24 mm	26 mm
R	10 mm	10 mm
L	70 mm	64 mm
C_1	_	ϕ 42 mm, depth 3.5mm
C_2	_	ϕ 25 mm, depth 1.5 mm
Fillet, F	_	mm

5.1 Failure Analysis of Optimized Universal Joint

The failure analysis of the optimized design was conducted in order to find the maximum limit of torque value up to which the design runs safely. The material taken was structural steel.



Graph 6.1. Equivalent stress in the universal joint at respective torque values.

It is clear that the existing design could be run for torque value up to 3850 N-m but the optimized design can be run for up to 4550 N-m at it just reaches the UTS at 4555 N-m. Therefore the optimized design is safe up to the torque 4555 N-m.

6. Conclusion and Future Scope

6.1 Conclusion

The present work is about the optimization of a universal joint using structural steel in order to reduce the mass and the stress values its sensitive part using FEM. The results and conclusion drawn from the present work in which the structural analysis of universal joint was performed is briefed below:

- The failure analysis shows that existing universal joint of structural steel was safe below the Torque moment 3850 N-m. It just fails at this torque value.
- The most sensitive area at which the high stresses are generated is universal joint cross. So the optimization of the cross was done in order to reduce stress and mass from it. The structural analysis of the universal joint was done for torque value 3800 N-m. It is the maximum torque value at which the subject is safe.
- The equivalent stress (von-mises) reduction in case of structural steel is 15.724 for the torque value equal to 3800 N-m.
- The mass of the optimized design reduces by 0.451%. The mass reduction from the cross alone, which is the most sensitive part is 4.473%.
- The original universal joint was safe below the torque value of 3850 N-m, but the optimized universal joint came out to be safe for the torque value equal to 4550 N-m and fails just at 4550 N-m. Hence the optimized design could run at even higher torque values than before.

6.2 Future Scope

- The optimization of the universal joint can be done considering the other lighter materials also. Hence a new material can be proposed for the universal joint.
- Further mass reduction can be done from the non-sensitive area of the universal joint that is yokes.

7. References

1. Shah P, Patil A. Modeling and FEA of universal coupling of an automobile truck. International Journal for Innovative Research in Science and Technology. 2016; 3(2):131–6.
2. Mandal A, Sharma U, Pant H. Static structural analysis of universal joint to study the various stresses and strains developed in power transmission systems. International Journal of Engineering Research and Technology. 2016; 5(3):71–6.
3. Kolekar DS, Kalje AM, Kulkarni SS. Structural analysis of universal joint using finite element methodology. International Journal of Scientific Research and Management Studies. 2015; 2(3):136–42.
4. Mahajan SS, Khamankar SD. Finite element analysis of universal joint. International Journal of Emerging Technologies and Innovative Research. 2014; 1(7):754–61.
5. Chaudhry S, Krishan G. Finite element analysis and weight reduction of universal joint using CAE tools. International Journal of Engineering Research and Technology. 2014; 3(10):81–7.
6. Rahman MZ, Adhikary D, Mumu TR. Failure analysis of a universal coupling using finite element method. Global Journal of Researches in Engineering Mechanical and Mechanics Engineering. 2013; 13(9):11–5.
7. Giridhar NS, Hetawal S, Baskar P. Finite element analysis of universal joint and propeller shaft assembly. International Journal of Engineering Trends and Technology. 2013; 5(5):321–7.
8. Sheikh SM. Analysis of universal coupling under different torque condition. International Journal of Engineering Science and Advance Technology. 2012; 2(3):690–4.
9. Yaghoubi M, Jafary A, Mohtasebi SS. Design, simulation and evaluation of a new universal joint with intersecting angle up to 100 degrees for farm machineries. International Journal of Engineering Science and Advance Technology. 2010; 1(4):149–52.
10. Bayrakceken H, Tasgetiren S, Yavuz I. Two cases of failure in the power transmission system on vehicles: A universal joint yoke and a drive shaft. Engineering Failure Analysis. 2007; 14:716–24. Crossref
11. Cristello N, Kim Y. Design optimization of an automotive universal joint considering manufacturing cost. 17th IASTED International Conference Modeling and Simulation. 2006; 4(1):499–504.