Review of Some Recent Innovations on Structural Performance Analysis of Beryllium Copper Alloy Material on Universal Joint

Zulkar Nain Khan[1], Dr. Manish Gangil[2]
[1]*M.Tech Student, Department of Mechanical Engineering, Sri Satya Sai College of Engineering, RKDF University, Bhopal*
Department of Mechanical Engineering, Sri Satya Sai College of Engineering, RKDF University, Bhopal

***Abstract*- The mechanical characteristics of alloy material beryllium copper will be analyzed by a comparative study using conventional material of universal joint i.e. structural steel and beryllium copper by FEM testing on Ansys 14.0. After studying various research articles of different authors we have found out these parameters and factors which can be further optimized for improving the structural strength of universal joint. material of construction optimization methods, Stress concentration, geometrical dimensions and mechanical construction, Finite element analysis, meshing methods, Weight reduction and life cycle. These are the parameters which can be optimized for getting better results of structural performance enhancement.**

INTRODUCTION

A universal joint is a positive, mechanical connection between rotating shafts, which are usually not parallel, but intersecting. They are used to transmit motion, power, or both. The simplest and most common type is called the Cardan joint or Hooke joint. It is shown in Figure 1. It consists of two yokes, one on each shaft, connected by a cross-shaped intermediate member called the spider. The angle between the two shafts is called the operating angle. It Is generally, but n01 necessarily, constant during operation. Good design practice calls for low operating angles, often less than 25°, depending on the application. Independent of this guideline, mechanical interference in the construction of Cardan joints limits the operating angle to a maximum (often about 37½°), depending on its proportions. The two fork ends are assembled co-axially with respect to the centre block. The pins are assembled into the holes provided in the fork end. They are held in position by means of a collar and a collar pin.

Factors Affecting Performance of Universal joint

After studying various research articles of different authors we have found out these parameters and factors which can be further optimized for improving the structural strength of universal joint.

2.1.1 Material of Construction

Ms. Nilesha Patil, Mrs. Sayli M.Sable and Mr. Kashinath Munde [A], Studied and calculated the stresses in Universal joint using analytical method. In this study, modeling and analysis of a universal joint was performed by using Finite Element Method. The commercial finite element package ANSYS version 17 was used for the solution of the problem. The modeling of the universal joint was done using 3D software. Here CATIA V5 has been used for modeling. The simulation part was carried out using the Analysis software, ANSYS. With the Boundary constrains and the tensile load applied, the universal joint is analyzed and the values are tabulated. They used structural steel as the new material instead of white cast iron and grey cast iron. They found this replacement effective in terms of von mises stresses.

Prof. Swati Datey, et.al. [B], Studied in this project for stresses calculation on the universal joint and to improve the performance of universal joint to a certain extent with CATIA V5 and FEM. After study of universal joint used in tractor-trailer and analysis on universal joint pin they concluded that material plays a very important role in stress reduction acting on joint especially on pin. They changed material like grey cast iron (ASTM grade 20 (EN-JL 1020), ASTM grade 35 (EN-JL1040), ASTM grade 60 (EN-JL 1070)), Stainless steel and Titanium alloy and found that deviations in Equivalent stress (von mises), shear stress and total deformation occurs at

same load and diameter in which it has maximum stresses. They concluded that increase in pin diameter can lead to protect bending of joint.

Nipun Kumar, Dr. Gian Bhushan and Dr. Pankaj Chandna [C], Studied for universal joint made of stainless steel, gray cast iron magnesium, aluminum, stainless steel, structural steel and gray cast iron. They analyzed for stress and deformation under different loading conditions. The CAD model of universal joint was made in CATIA V5 R20 and analyzed in ANSYS 15. It was observed that stresses developed for universal joint made of magnesium were least and the universal joint made of aluminum can sustain maximum tensile load without failure.

Kodali. Vikas and Kandula. Deepthi [D], analyzed the E Glass and S2 Glass epoxy composite pinned joints with two serial holes by varying distance from the free edge of the plate to the diameter of the first hole and width of the specimen to the diameter of the holes and also and the distance between center of two holes-to-hole diameter. Structural and Fatigue analysis were done using Cosmos. By observing the structural analysis results, the stress and displacement values were less than their respective strength values. They concluded that using composite materials is safe for serial pinned joints. Damage factor was very less for both materials and life was about 106 cycles.

Shaik.John Bhasha and Hari Sankar Vanka [E], Studied on design and analysis of a universal joint which is used in power transmission. Universal joint was design for 50KN axial load by theoretical calculation. Final dimensions from theoretical calculation, model of Universal joint was made in CATIA V5 and model was taken to ANSYS and simulated with various material and check for best material which suit for given design load. it was concluded that Teflon was best for design and it was close to stress got for stainless steel and cast iron.

Geun-Yeon Kim, Seung-Ho Han and Kwon-Hee Lee [F], Studied and changed the existing material made of GCD45 to Al6082M and recommended the lightweight design of the universal joint as the optimal design technique to be installed in small cars. Six shape design variables were selected for the optimization of the universal joint and the criteria relevant to stiffness and durability were considered as the design requirements during the optimization process. The Meta model-based optimization method that uses the kriging interpolation method as the optimization technique was applied. The result shows that all constraints for stiffness and durability are satisfied using A16082M, while reducing the weight of the universal joint by 60% compared to that of the existing GCD450.

2.1.2 Optimization Methods

Ms. Nilesha U. Patil, Mrs. Rupali S. Sewane and Mr. Kashinath H. Munde [G], Studied and calculated the stresses in Universal joint and optimized the model of Universal joint for its weight reduction. Modeling of the universal joint was done using 3D software. Here CATIA V5 had been used for modeling. The simulation part was carried out using the Analysis software, ANSYS. With the Boundary constrains and the twisting moment applied, the universal joint was analyzed. Then using Topology optimization material was removed. Again, analysis was done on an optimized model for stresses and deformation and optimized values. The maximum stress and deformation values were in the acceptable limits.

Mahesh P. Sharma, et.al. [H], Have done static analysis of steering universal joint. They designed a universal joint which accommodates dual caliper mountings for increasing braking efficiency & reducing a stopping distance of a vehicle. CAD modal of universal joint was prepared in CREO2.0. Static analysis was done in ANSYS WORKBENCH by constraining the universal joint, applying loads of braking torque on caliper mounting, longitudinal reaction due to traction, vertical reaction due to vehicle weight and steering reaction. They have also done shape optimization of same universal joint and saved material resource. Shape optimization of universal joint was done using ANSYS WORKBENCH making objective function as reducing weight. Shape optimization method used in this study reduced the mass of universal joint by 19.35%. Also factory of safety is between 3 to 4. Maximum stress and displacement were within control. They concluded that the overall weight of the vehicle can be reduced to achieve savings in costs and materials, as well as, improve fuel efficiency and reduce carbon emissions.

Ms.Nilesha U. Patil, et. al. [I], Studied and calculated the stresses and deformation in Universal joint and optimized the model of same Universal joint. They used CATIA V5 for modeling. They aimed to use FEA and Taguchi method to improve the quality of manufactured goods and engineering development of

design for studying variation. Taguchi recommends the use of the S/N ratio to measure the quality characteristics deviating from the desired values analysis. Regardless of the category of the quality characteristic, a greater S/N ratio corresponds to better quality characteristics. It was predicted that Taguchi method is a good method for optimization of various machining parameters as it reduces the number of experiments. Stress, strain and deformation were within acceptable limits.

Pankaj Dulani and S. A. K. Jilani [J], Studied the problem of the failure of the universal joint pin in any mechanism for general due to crushing, tearing and shearing. The aim of the present paper was to study calculate the stresses in Universal joint using analytical method. The study focused on the optimization of design parameters kept in mind for the universal joint pin. The Neural Network Tool, a nontraditional global optimization technique had been used as the solution methodology for its inherent advantages. Optimal results so obtained were compared with remodeled universal joint pin with stress minimizing effect considered as a key factor. After remodeling of the universal joint using the predicted optimized parameters obtained by neural network the model was used to generate the value of stress which was compared with the neural network result in order to prove that optimized model is better as compared to the previously selected four models.

2.1.3 Stress Concentration

Miss. Yogini .V. Deore, et. al. [K], Studied the Universal joint The FEA Analysis of Universal joint was done and various shear and tensile stresses results were plotted. The Analytical solution of universal joint was found out using standard calculations. The force applied universal joint was 50 KN .The diameter of pin was proposed to be around 30 mm. FEA software results were correct as per theoretical calculations. it was also concluded that certain high stresses were generated near universal joints and this result were helpful for further analysis of Headstock for reducing the stresses, increasing life and reliability of headstock.

Dinesh Shinde And Kanak Kalita [L], Studied stresses on tractor trailer during acceleration (tensile) and during deceleration (compressive). Forces acting over the joint were calculated by considering Newton's Second Law of motion. Pin was considered separately for the analysis and finite element analysis is done on it. They concluded that Numerical value of tangential force and von mises stresses acting on the universal joint were maximum in case of deceleration.

Abhishek Mandal & Utkarsh Sharma [M], conducted static structural analysis on a universal coupling using advanced computer aided engineering software and study the various stresses and strains developed in the joint. Results concluded that the fork pin experiences the maximum compressive stresses and strains as referenced earlier. Also stated that region where the fork and the fork pin makes contact experiences generally higher compressive stress and bending stresses. Also analyzed that stress concentration in the collar and pin due to the presence of notch that leads to frequent wearing out of the pin which causes the shaft to wobble unnecessarily which reduces the mechanical efficiency of the transmission system. This leads to failure of the transmission system.

2.1.4 Geometrical dimensions and mechanical construction

Suraj Yadav, et.al. [N], Have done modeling and analysis of universal joint under a certain conditions. Modeling and analysis of a universal joint was performed by using 3D software CATIA & Finite Element Analysis (FEA) respectively. The commercial finite element package ANSYS version 15 was used for the solution of problem. They concluded that 30C8 material having maximum permissible stress is 400MPa and Maximum stresses developed in universal joint are 201MPa. So design is safe. They also concluded that pin of 25 mm diameter can sustain load of 50 KN without a failure.

Shankar Majhi & Shaheen Beg Mughal [O], analyzed universal joint pin stresses during its operation. Force acting on the fork and pin were calculated by the theoretical study and analytical method. Subjected to high stresses in pin were studied by using CATIA V5 and finite element method. According to their theoretical study, calculation and F.E.A results were similar on 50 mm diameter at 60 KN. They concluded that when stress on pin increases, bending increases but when we increase the pin diameter it will wear maximum stress on that force.

Ravindra S. Dharpure and Prof D. M. Mate [P], Reviewed the problem of the failure of the universal joint pin in a railway coupling due to shearing as per the defined conditions and analysis the present steel material can be substituted with a proper elastic material. The presently for the problem of shear failure of the pin alternatively plastic universal joint pin that will accept bending fatigue, thereby reducing pin failure can be used. The pin made of a plastic material having a flexibility that will allow it to bend and return to its original shape and to also be self-lubricating. Further, the pin eliminates rust and corrosion and produces a low coefficient of friction between the pin and the coupler body and universal joint, thus enhancing opening and closing of the universal joint by reducing rotational resistance, thereby promoting safety. It has been known that steel pins, either at the time of installation or after service, can cause a "lazy universal joint", i.e., a universal joint that will not open all the way on decoupling.

Sourav Das, Vishvendra Bartaria & Prashant Pandey [Q], studied for calculating the stresses in Universal joint using analytical method. Material of the universal joint is considered as mild steel grade 30C8, ANSYS software was run and the stress contour, displacement contour, strain energy contour were obtained. It was proposed that instead of mild steel pin we can also use high strength high modulus steel pin that can further enhance the capacity to withstand higher loads. The shape of the universal joint can be changed for improved properties. Further study in this direction can made by using various directions of the pin and the capacity to withstand load.

2.1.5 Finite element analysis meshing method

Sangamesh B. Herakal, Ranganath Avadhani and Dr.S.Chakradhar Goud [R], Studied to calculate the stresses in Universal joint using analytical method. They concentrated on which type of meshing is preferable for components. universal joint was modeled by making use of CATIA, later on that model was imported in HYPERMESH and carried out both mesh those were hexahedral and tetra mesh. The model was solved by using Abacus software. They concluded that fork takes higher stress and eye takes less stress under loading condition. They showed that hex mesh is better than the tetra mesh. They also concluded that further study in this direction can made by using various directions of the pin and the capacity to withstand load.

2.1.6 Weight reduction and life cycle

Dhananjay S Kolekar, abhay M. Kalje and Swapnil S Kulkarni [S], Have done finite element analysis of the universal joint to find the stress and displacement. For modeling of the component PRO-E software was used. Pre-processing work like meshing and analysis work was carried out in HYPERWORKS software. The geometry was modified using topology and free size optimization which enabled to reduce stress level marginally well below the yield limit. They got a percent mass reduction of about 7%. The developed stresses for this model were within the acceptable limits which showed the safety of model.

Pilla. Anitha and V. Hari Shankar [T], focused on optimization of steering universal joint targeting reducing weight as objective function with required strength, frequency and stiffness. They used optimization which refers to different cases in the shape optimization and also the topology optimization. The modeling of this project was done in CREO Parametric 2.0 and the analysis is carried out in ANSYS 15.0. The optimization of results was achieved i.e. less stress value and also less weight. The model was analyzed with Cast Iron, Aluminum alloy and S- Glass Epoxy composite. There was a significant amount of weight reduction when they used S-Glass Epoxy material.

Nishant Vibhav Saxena And Dr. Rohit Rajvaidya [U], proposed the modification of one of the material that changed cast iron by a composite polymer material. The proposed system had many advantages over other system such as making the device, simpler and having maximum safety and is ecofriendly. Composite polymers are characterized by a high flexibility material. They used ANSYS 13 used for analysis of universal joint with modified material and varying loads. They concluded that parts made out of composite materials are economical to produce, and facilitate overall systems cost reductions by eliminating secondary operations for parts, such as machining, as well as facilitating reduction in part count when compared with metal parts.

Vivek Shaw, et.al. [V], analyzed advanced materials focusing on a mechanical joint, i.e. the Universal joint. They suggested a modification over the

conventionally used material, such as Aluminum alloy that is widely used for manufacturing the Universal joints. They used CATIA V5R18 for modeling the 3D geometry of Universal joint and ANSYS (Workbench 16.2) was used for finite element analysis of the same with the conventional and composites materials respectively. The results approved that the use of composite material not only decreases the weight of the material but it also improves the life of the component as the composite material shows less deformation in comparison to the conventional one. Due to application of composite material there was a negligible change in stress value but directional deformation and weight of the system got reduced by 73.7 % and 22.02% respectively.

CONCLUSION

There literature survey summarizes that following factors and parameters can be used and analyzed to get the improved structural strength of universal joint material of construction, optimization methods, Stress concentration, geometrical dimensions and mechanical construction, Finite element analysis meshing methods, Weight reduction and life cycle. These are the parameters which can be optimized for getting better results of structural performance enhancement.

REFERENCES

[1] Ms. Nilesha Patil, Mrs. Sayli M.Sable and Mr. Kashinath Munde, "Static Structural Analysis of Universal joint", International Journal of Advanced Technology in Engineering and Science, vol 4, issue 12, december 2016.

[2] Prof. Swati Datey, Amit A. Rangari, Adarsh A. Dongre, Kunal A. Paraskar & Sanket V. Lidbe, "Analysis of Universal joint used in Mahindra 575 DL", IJARIIE-ISSN(O)-2395-4396, Vol-3 Issue-2 2017.

[3] Nipun Kumar, Dr. Gian Bhushan and Dr. Pankaj Chandna, "Analysis of Universal joint of Various Materials using CAE Tools", International Journal of Engineering Technology, Management and Applied Sciences, Volume 5, Issue 1, January 2017.

[4] Kodali. Vikas and Kandula. Deepthi, "Analysis of Serial Pinned Joints in Composite Materials", International Journal of Computational Engineering Research, Vol, 04, Issue, 10, October– 2014.

[5] Shaik.John Bhasha and Hari Sankar Vanka, "Modeling and Analysis of Universal joint", International Journal & Magazine of Engineering, Technology, Management and Research, Volume No: 2 (2015), Issue No: 11 (November).

[6] Geun-Yeon Kim, Seung-Ho Han and Kwon-Hee Lee, "Structural Optimization of a universal joint with Consideration of Stiffness and Durability Requirements", The Scientific World Journal, 2014.

[7] Ms. Nilesha U. Patil, Mrs. Rupali S. Sewane and Mr. Kashinath H. Munde, "Optimization of Universal joint by using FEA", International Conference on Ideas, Impact and Innovation in Mechanical Engineering), Volume: 5 Issue: 6, 2017.

[8] Mahesh P. Sharma, Denish S. Mevawala, Harsh Joshi and Devendra A. Patel, "Static Analysis of Steering universal joint and Its Shape Optimization", IOSR Journal of Mechanical and Civil Engineering, PP 34-38, 2014.

[9] Ms.Nilesha U. Patil, Prof.P.L.Deotale, Prof. S.P.Chaphalkar, A.M.Kamble & Ms.K.M.Dalvi5, " Application Of Taguchi Method For Optimization Of Universal joint" International Journal of Recent Trends in Engineering & Research (IJRTER) ,Volume 03, Issue 11; November - 2017 .

[10] Pankaj Dulani and S. A. K. Jilani, "Diameter and Spiral Thickness Optimization of Universal joint Using Neural Network", International Journal of Science and Research, Volume 5 Issue 2, February 2016.

[11] Miss. Yogini .V. Deore ,Prof. J.R.Mahajan K, Mr. Vinay Patil & Mr. Balasaheb Ugale, " A Static Structural Analysis Of Universal joint", International Journal Of Advanced Research In Engineering & Management (Ijarem) ,Pp. 10-22, Vol. 03, Issue 04, 2017.

[12] Dinesh Shinde And Kanak Kalita, "FE Analysis Of Universal joint Pin Used in Tractor Trailer", ARPN Journal Of Engineering And Applied Sciences, Vol. 10, No. 5, March 2015.

[13] Abhishek Mandal & Utkarsh Sharma, "Static Structural Analysis of Universal Joint to Study the Various Stresses and Strains Developed in

Power Transmission Systems", International Journal of Engineering Research & Technology, Vol. 5 Issue 03, March-2016.

[14] Suraj Yadav, Sanket Benade, Sushil Angchekar, Vaibhav Dhokle and Prof. Rakesh Kolhapure, "Design And Analysis Of Universal joint By Using FEA",Technical Research Organization India,Volume-4, Issue-6, 2017.

[15] Shankar Majhi & Shaheen Beg Mughal, "Modeling and analysis of universal joint used in tractor", International Research Journal of Engineering and Technology, Volume: 04 Issue: 07, July -2017.

[16] Ravindra S. Dharpure and Prof D. M. Mate, "Study and Analysis of Pin of Universal joint in Train", Journal of Emerging Technologies and Innovative Research, Volume 1 Issue 3.

[17] Sourav Das, Vishvendra Bartaria & Prashant Pandey, "Analysis of Universal joint Of 30C8 Steel For Automobile Application", International Journal Of Engineering Research & Technology, Vol. 3 Issue 1, January – 2014.

[18] Sangamesh B. Herakal, Ranganath Avadhani, Dr.S.Chakradhar Goud, "Structural Static Analysis Of Universal joint", International Journal Of Engineering Research And General Science, Volume 4, Issue 2, March-April, 2016.

[19] Dhananjay S Kolekar, abhay M. Kalje, swapnil S Kulkarni, "Design, Development And Structural Analysis Of Universal Joint", International Journal Of Advanced Engineering Research And Studies, Vol 4, issue 4, July-September, 2015/09-12.

[20] Pilla. Anitha and V. Hari Shankar, "Design And Topology Optimization Of A Steering Universal joint Using FEA", International Journal of Scientific and Research Publications, Volume 6, Issue 11, November 2016.

[21] Nishant Vibhav Saxena And Dr. Rohit Rajvaidya, "Study & Analysis Of Universal joint With The Replacement Of Material By Using Teflon", Int. Journal Of Engineering Research And Applications, Vol. 5, Issue 3, (Part -4) March 2015.

[22] Vivek Shaw, Tanuj Srivastava, Rohit Ghosh and Dr. Rabindra Nath Barman," Numerical Simulation Of Universal joint Using Finite Element Method: A New Approach Based On Composite", International E-Journal For Technology And Research, Volume 1, Issue 5, May 2017.

[23] Frederick T. Wallenberger; Paul A. Bingham (October 2009). Fiberglass and Glass Technology: Energy-Friendly Compositions and Applications. Springer. pp. 211–. ISBN 978-1-4419-0735-6. Retrieved 29 April 2011.

[24] Ashokrao Shinde, Omkar Chandrakant Vibhute, "Modeling and Analysis of Universal joint ", International Journal of Engineering Research & Technology, Vol. 6 Issue 07, July – 2017