Modeling and Failure Analysis of Universal Joint using ANSYS

Rahul Arora
Mechanical Department (M.E Student), National Institute of Technical Teacher Trainee and Research, INDIA

Abstract – Vehicle transmission framework comprises a few parts which in some cases experience the ill effects of various burdens (disappointments). An all inclusive joint comprise of two yokes, one on each pole, associated by cross-molded middle of the road part i.e. creepy crawly. Yoke get together are constantly subjected to torsion and shear. Era of stress, removal and strain in a general coupling has been broke down. Yoke get together are turning part and here and there experience the ill effects of weakness by use of variable torque. In this paper limited component investigation of the part is done to discover the stress and deformation of the last item. For demonstrating of the segment Solidworks CAD software is utilized. Pre- handling work like cross section and examination work is done in ANSYS CAE software. Utilizing FEA investigation, we can recognize the nature and qualities of stresses following up on the Yoke and evaluate the impact of the load/mass geometry/boundary limit conditions over the yoke.

Index Terms – FEA: Finite Element Analysis.

1. INTRODUCTION

All inclusive joint otherwise called general coupling, U joint, Cardan joint, Tough Spicer joint, or Hooke's joint is a joint or coupling in an inflexible bar that enables the bar to "bend" toward any path, and is usually utilized as a part of shafts that transmit revolving movement. It comprises of a couple of pivots found near one another, arranged at to each other, associated by a cross shaft. An all inclusive joint comprise of two yokes, one on each pole, associated by cross- formed middle of the road part i.e. creepy crawly. Yokes gathering are constantly subjected to torsion and shear. Working of the mill utilizations of universal coupling incorporate flying machine apparatuses, control systems, gadgets, instrumentation, medicinal and optical gadgets, arms, radio, sewing machines, material hardware and device drives. Fig. 1 demonstrates a monetarily accessible all universal coupling and Fig. 2 indicates computer aided design plan of a coupling.

Fig.1

There are some accessible writing on all universal coupling. The curiosity of the present writing is that it gives accentuation on yoke and center independently. The principle push zone of a coupling is the yoke space center point interface and the center corners.

In the yoke space center point interface, the anxiety demonstrations are circumferential. In the meantime, frictional interface creates warm. This warmth produces warm anxiety. So amid the operation of a coupling, for the most part two sorts of stresses work specifically circumferential anxiety and warm anxiety. Amid our examination, we connected circumferential worry at the yoke opening center point interface to perceive how it influences the stress proliferation in both the yoke and the center point. What's more, we connected warm anxiety and expanded it slowly to show how it impacts era of stresses. To demonstrate the strain rate and the relocation in the yoke, the strain form and the uprooting shape are likewise plotted. The recreation was completed in ANSYS 2015, the legitimacy and worthiness of which is settled. The material chose is structural steel.

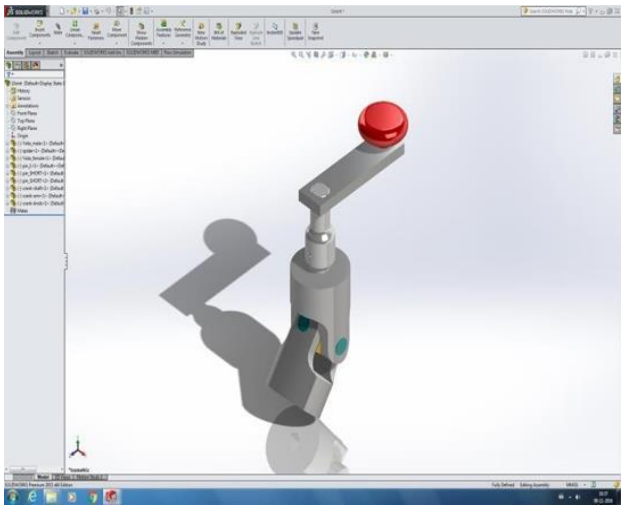

Fig 2. UNIVERSAL JOINT IN SOLIDWORKS

2. BACKGROUND AND REVIEW

The fundamental point of this venture is to outline a model and decide the Von-Misses stresses, component relocation, and enhancement in the current guiding yoke. On the off

chance that the current outline demonstrates the disappointment, then propose the base plan changes in the current guiding yoke. In this venture, just the static FEA of the controlling yoke and the directing shaft has been performed by the utilization of programming. At that point the last derivation will be made the plan is acknowledged or not from the FEA breaks down.

3. FORMULATION OF THE PROBLEM

The equation of motion relating the two angular positions of the two yokes is given by,

tan Y1=cosβ tan Y2 where,Y1=angle of rotation of yoke 1

Y2=angle of rotation of yoke 2

β = the angle of the yokes with respect to each other .

The angles Y1 and Y2 in a rotating joint will be function of time. Differentiating the equation of motion with respect to time and using the equation of motion itself to eliminate a variable yields the relationship between the angular velocities are -: ω1=dY1/dt and ω2=dY2/dt

ω2=ω1cosβ/(1-sin2β cos2Y1)

The angular speeds are not straightly related but instead are occasional with a period twice that of the pivoting shafts. The precise speed connection can again be separated to get the connection between the rakish increasing speed α1 and α2.

4. SOLUTION METHODOLOGY

The computer aided design drawing is completed in SolidWorks 2015. At first male yoke, center point and female yoke are attracted part drawing alternative independently. At that point these three things are collected in get together drawing choice.

An exploratory and limited component examination of universal coupling was done with help of ANSYS for various torque condition. Concentrated some basic explanation behind disappointment of drive shaft and all inclusive joint yoke of car power transmission framework with the end goal that disappointment might produce and configuration shortcomings, material handling issue and client started issues. The disappointment investigation was done for an all inclusive joint burden and a drive shaft of a car control transmission framework.

5. OBJECTIVES OF RESEARCH

A parallel assignment was the appraisal of the all inclusive joint conduct under stacked conditions through some numerical reenactments. Many harmed tests of the Universal joints were likewise gathered. The harmed areas of such joints were painstakingly inspected. The expository and the numerical mimicked results were confirmed by contrasting them and the result of the examinations about some harmed tests. After watchful examinations of the harmed heading and by turning to the systematic and the numerical mimicked comes about a few cures are proposed. It is foreseen that such recommendations can expand the execution and enhance the future of the General joints. The following are the objectives of the study

- Study the application of case in consideration.
- Secure 3D in SOLIDWORKS and Modeling followed by pre-processing (meshing).
- Using Finite Element Methodology, conduct linear static structural analysis over the meshed model.
- Analyze for iterations and comparison.

6. METHODOLOGY

This can be understood by a flow chart given below-:

7. MODELLING

The stacking states of torsional minute and the rotational speed is kept for the outcome. The model of universal joint was investigated in ANSYS viewing the segment as

comprised of basic steel, which is a material in the alloy steel gathering. Basic steel has Ductile Yield quality of 250 MPa and Elastic Extreme quality of 460 MPa. The computer aided design model of all inclusive joint was composed in SOLIDWORKS programming and examined in ANSYS workbench. Be that as it may, there were a few changes made in the geometry of the current model.

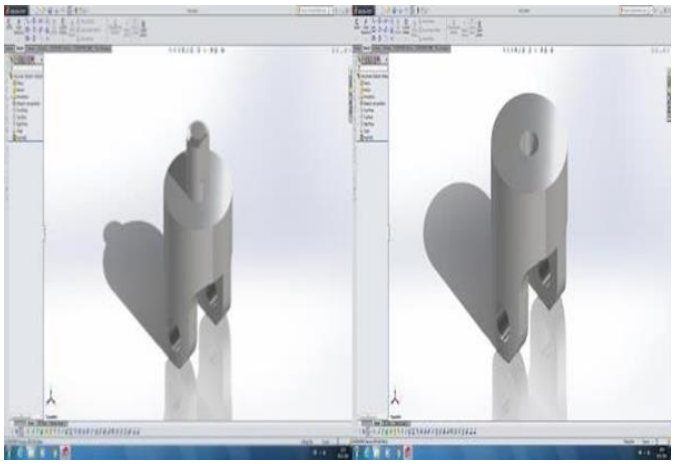

Fig3.MALE YOKE Fig4.FEMALE YOKE

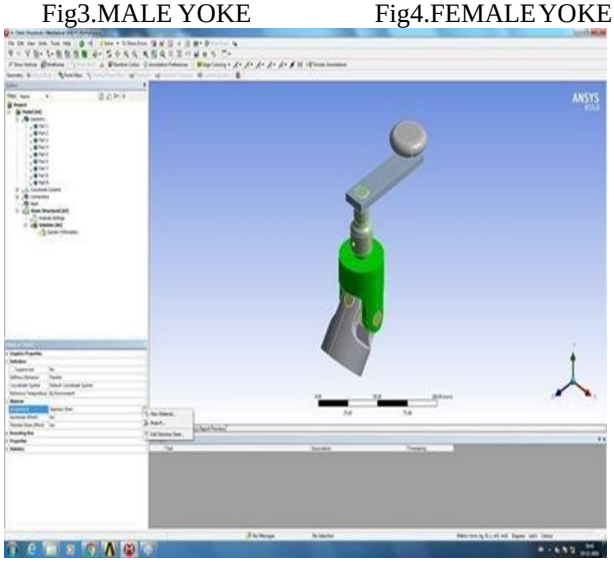

Fig.5 MATERIAL SELECTION AS STRUCTURAL STEEL

In the wake of creating the computer aided design models and get together in SOLIDWORKS, it is spared in .igs configuration and this .igs record is foreign in SolidWorks programming. At that point this foreign record is spared in the parasolid (.xt) configuration to dodge the information misfortune. Lastly the model imported in ANSYS Workbench.

All inclusive JOINT Examination

After culmination of the Limited Component Demonstrate, limit condition and loads are connected. Client can characterize requirements and loads in different ways. This helps the client to monitor stack cases. The limit condition is the accumulation of various powers, backings, limitations and whatever other condition required for finish examination. Applying limit condition is a standout amongst the most commonplace procedures of investigation. An extraordinary care is required while allocating burdens and imperatives to the components. The FEA aftereffects of the all inclusive joint have been dissected and contrasted and the accessible outcomes for approval. The shading appeared in the widespread joint speaks to the burdens and distortion display in the component. Red shading demonstrates the most extreme anxiety and maximum stress, though, blue shading demonstrates the base anxiety and least disfigurement in the figure. Fig. 7 demonstrates the aggregate maximum stress in the all inclusive joint (gathering).

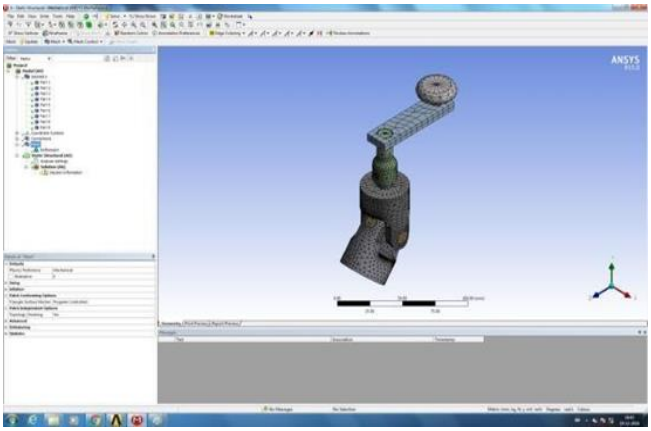

Fig6. MESHING IN ANSYS

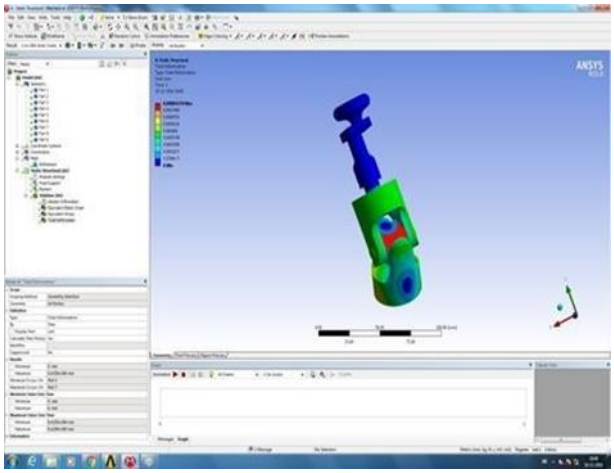

fig7. TOTAL DEFORMATION

The joint yoke is dissected for the most extreme key stress, greatest shear stress, add up to twisting and most extreme key strain for a torsion snapshot of 40 Nm. Fig. 7 demonstrates the greatest important worry in the joint yoke for the torsional snapshot of 40 Nm.

The investigation consequences of joint yoke and pin after advancement of the current model are appeared in table below.Fig8. demonstrates the consequence of the joint.

Sr. No.	Parameters	Existing Results	FEA Results
1.	Maximum Principal Stress (Pa)	11.92 e6 Pa	8.32 e6 Pa
2.	Maximum shear stress (Pa)	7.82 e6 Pa	7.97 e6 Pa
3.	Total deformation (m)	1.82 e-5 m	0.85 e-5 m
4.	Maximum principal strain	8.76 e-5	5.32 e-5

Table-1

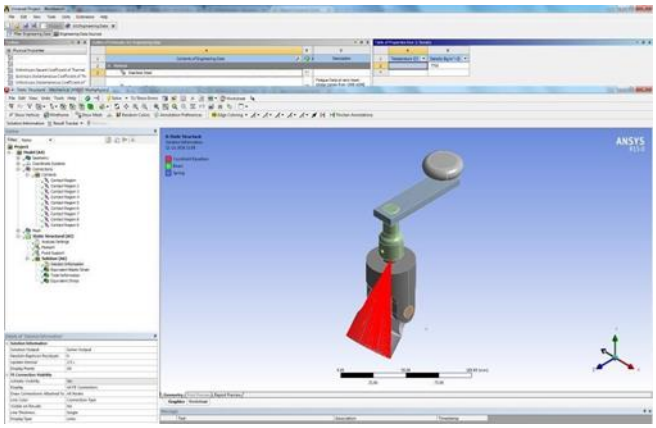

Fig8. RESULT OF UNIVERSAL JOINT

The ANSYS examination demonstrates that the most extreme guideline push esteem got is 8.326 X e6 Pa, which is inside as far as possible as endorsed in the paper. Besides, this esteem is additionally not as much as the most extreme stress esteem in the first segment. In this way we can reason that the new displayed segment is well inside as far as possible.

8. CONCLUSIONS

The outcomes were gotten are very positive which was normal. This outcome center the connection between the assembling expense and joint edge execution measures of a car widespread joint. From the outcomes acquired from FE Analysis, numerous dialogs have been made. The outcomes acquired are well in concurrence with the accessible existing outcomes. The model exhibited here, is well sheltered and under reasonable cutoff of stresses. It can be noticed that disappointment of segment is happen due assembling and configuration blame, crude material shortcomings, looks after issues, material preparing deficiencies, to evade this issues different methods, for example, either, change material or geometry and direct examination on the same lastly actualized in the arrangement or used to discover the best outline of Joint with considering the all the component, for example, weight, cost, Fatigue life, push dissemination, solidness, and so forth.

The mechanical stress is discovered most extreme close to the sharp edges.

REFERENCES

[1] Vishal Rathi and Nitin K. Mandavgade (2009) "Fem analysis of universal joint of Tata 407" Second International Conference on Emerging Trends in Engineering and Technology, ICETET-09, ISBN: 978- 0-7695-3884-6, PP. 98-103.

[2] Nick Cristello, and Yong Kim (2006) "Design optimization of an automotive universal joint Considering manufacturing cost", 17th IASTED International Conference Modeling and Simulation, ISSN: 530-091 , PP. 499–504.

[3] S.K.Chandole, M.D.Shende, M.K. Bhavsar" Structural Analysis Of Steering Yoke Of An Automobile For Withstanding Torsion/ Shear Loads",IJRET: International Journal of Research in Engineering and TechnologyVolume: 03 Issue: 03 | Mar-2014.

[4] H. Bayrakceken, S. Tasgetiren, I. Yavuz (2007) "Two cases of failure in the power transmission system on vehicles: A universal joint yoke and a drive shaft", Engineering Failure Analysis, Vol. 14, ISSN: 1350-6307, PP. 716- 724.

[5] Majid Yaghoubi, Ali Jafary, Seyed Saeid Mohtasebi (2010) "Design, simulation and evaluation of a new universal joint with intersecting angle up to 100 degrees for farm machineries", AJAE 1(4), ISSN:1836-9448, PP: 149-152.

[6] Muhammad Ziaur Rahman, Debasish Adhikary & Tazmin Rashid Mumu (2013) "Failure Analysis of a Universal Coupling using Finite Element Method

[7] P.G. Tathe, Prof. D.S. Bajaj and Swapnil S. Kulkarni, "Failure Analysis And Optimization In Yoke Assembly Subjected By Torsion And Shear", International Journal of Advanced Engineering Research and Studies E-ISSN2249–8974. JulySept,2014.

[8] Naik Shashank Giridhar, Sneh Hetawal and Baskar P.,"Finite Element Analysis of Universal Joint and Propeller Shaft Assembly", International Journal of Engineering Trends and Technology (IJETT) - Nov 2013 – Volume Number 5.