Finite Element Analysis and Optimization of Universal Joint Yoke using CAE Tools

Ravikant
Department of Mechanical Engineering
Amity University Haryana, Haryana, India
ravisharma.nitk@gmail.com

Abstract — The objective of the universal joint yoke is to form connection between two shafts i.e. transmission shaft and drive shafts, whose axis intersect and the rotation of one shaft about its own axis results in the rotation of the other shaft about its axis. The work described here forms part of an optimization project carrying out the design optimization of universal joint yoke and drive shaft using the ANSYS software. The model of universal joint yoke has been developed in Solid works then imported in ANSYS workbench and the model of drive shaft has been generated in ANSYS itself. In this work finite element analysis of a universal joint yoke and drive shaft has been taken as a case study. In the present work the static analysis of universal joint yoke and a drive shaft has been carried out. The results for equivalent von-mises stress, strain and deformation has been plotted and compared with the existing results.

Keywords — Universal joint yoke, Solid works, Finite Element Analysis, Optimization, ANSYS workbench

I. INTRODUCTION

Universal joint is a form of connection between two shafts, whose axis intersect, the rotation of one shaft about its own axis results in the rotation of the other shaft about its axis. As yoke joint is the motion transmission component. So load acting on the yoke is two torsional moments acting at spider mounting locations of the yoke in the opposite directions. The maximum stress produced in the yoke can be minimized by improving the existing design of the existing yoke. The cardon joint suffers from one major problem. Even when the input drives shaft axle rotates at a constant speed, the output drive shaft axle rotates at a variable speed, thus causing vibration and wear. The variation in the speed of the drive shaft depends on the configuration of the joint.

A. M. Heyes [1] carried out the automotive component failure. The failure of vehicle components is an area which is likely to affect all of us at one stage or another. In this paper the distribution of component failures is discussed, as well as the causes thereof. Four case studies are presented to give insight in the methodology of failure analysis of automotive components, and the valuable information which can be gained thereby.

S. R. Hummel, C. Chassapis [2] presented the configuration design and optimization of universal joints. Universal joints are used to connect misaligned shafts that are intersecting. They transmit rotational motion from one shaft to another. The joint consists of input and output yokes and a cross trunnion. The cross stunning consists of a block and two pins. The large pin goes through the block and the small pin goes through the block and the large pin. In this investigation a systematic approach to the design and optimization of the ideal universal joint has been developed. The relationships to design universal joints with the minimum diameter required to handle a given input torque for a given joint angle have been derived. Universal joints which are designed using the approach presented here will be ensured not to have interference between the various parts of the mechanism when in operation.

Scott Randall Hummel, Constantin Chassapis [3] presented the Configuration design and optimization of universal joints with manufacturing tolerances. This paper describes an approach to the design and optimization of Cardan joints with manufacturing tolerances. Relationships have been developed to ensure that binding does not occur between the various components of the mechanism. The optimization methodology presented here minimizes the diameter of the yokes of the joint while ensuring that the strength is adequate to handle a given input torque.

H. Bayrakceken, S. Tasgetiren, I. Yavuz [4] analysed the two cases of failure in the power transmission system of vehicles: A Universal joint yoke and a drive shaft. Power transmission system of vehicles consist several components which sometimes encounter unfortunate failures. Some common reasons for the failures may be manufacturing and design faults, maintenance faults raw material faults, material processing faults as well as the user originated faults. In this study, fracture analysis of a universal joint yoke of an automobile power transmission system is carried out. Spectroscopic analyses, metallographic analyses and hardness measurements are carried out for each part. For the determination of stress conditions at the failed section, stress analyses are also carried out by the finite element method.

N. K. Mandavgade, Vishal Rathi [5] have done the FEM analysis of universal joint of TATA- 407 traditional design has been done by simple calculation but with increase in product performance and reliability it is difficult to follow the traditional iterative design procedure. To satisfy the market needs it is necessary to provide a computational capacity along with the creativity of the human being. A widely used numerical method for solving structural problems in both industry and academia is FINITE ELEMENT METHOD.
With increase in product performance and reliability it is difficult to follow the traditional iterative design procedure. To satisfy the market needs it is necessary to provide a computational capacity along with the creativity of the human being. As the various scientist and engineers have performed different analysis on joint yoke and drive shaft and try to optimize the design so that the failure of the yoke and drive shaft can be reduced. A widely used numerical method for solving structural problems in both industry and academia is finite element method. The finite element method is a simple, robust and efficient method of obtaining a numerical approximate solution for a given mathematical model of a structure. As product performance becomes more important and as designs become more complex the simple method has becomes inadequate. A detailed analysis and significant research effort have been devoted to the investigation of structured analysis of the power transmission. An attempt to redesign the yoke and drive shaft has been done to minimize the stresses and strain of the universal joint yoke and drive shaft so that the better performance of the joint yoke and drive shaft has been achieved.

As yoke joint is a motion transmission component, so load acting on the yoke is two torsional moments acting at spider mounting locations of the yoke in the opposite directions and in drive shaft torsional, bending and normal forces occur during the working of the shaft. The main objective of this work is to perform the Finite element analysis of universal joint yoke and drive shaft using CAE Tools, so as to determine the total deformation and stress distribution in the yoke and shaft. The deformation and stress contours have been plotted and patterns are studied. The results are compared and verified with available experimental and standard results. The optimization of yoke and drive shaft also achieve the reducing the weight of the assembly thus reducing weight and cost. Typical design variables for the yoke are the prongs and the thickness of the base plate. And the distance between the two prongs has to be same to maintain the other dimensions. The distance between the prongs was not changed where as the other dimension will affect the stresses and deformation in the yoke.

II. OBJECTIVES
The main objective of the work is to determine the equivalent Von-Mises stress, shear stress, strain and total deformation. In this paper, only the static FEA of the joint yoke was performed. To give the optimum design parameter a number of deformation, random designs were generated by varying the values of the design variables within the specified limits till optimized design had been reached. The results were determined under the same weight and loading condition as for the existing joint yoke [4].

<table>
<thead>
<tr>
<th>Material selected</th>
<th>Structural steel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s Modulus,(E)</td>
<td>2.0* 10^5 MPa</td>
</tr>
</tbody>
</table>

MODEL OF JOINT YOKE
The first step in pre processing is to prepare a CAD Model of joint yoke in Solid work. CAD modelling of the complete yoke is generated using Solid work software. The CAD model of joint yoke has been created in Solid work as shown in fig 1.

FIG.1 CAD model of universal joint yoke

MESH GENERATION
Finite element mesh is generated using parabolic tetrahedral elements (45335 elements). The von Mises stress is checked for convergence. An automatic method is used to generate the mesh in the present work.

FIG.2 Meshed model of universal joint yoke
FEA RESULTS OF UNIVERSAL JOINT YOKE

After completion of the finite element model, boundary condition and loads are applied. Two torsional moments of 200 Nm is applied at the spider mounting location in the opposite direction to each other on the joint yoke and cylindrical support is given at the centre hole of the base plate. And after that comparisons were made for optimization purpose. The static results are shown in figures given below.

The red region in the following figures shows the region of maximum stress and deformation. The fig. 2 shows the equivalent (von-mises) stress, fig. 3 shows shear stress, fig. 4 shows the equivalent elastic strain and fig. 5 shows total deformation.

Table: FEA Results Comparison Table

For Moment of 200 Nm

<table>
<thead>
<tr>
<th>Name</th>
<th>Existing results</th>
<th>FEA results</th>
<th>variation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Equivalent Von Mises stress</td>
<td>356 MPa</td>
<td>290.26 MPa</td>
<td>18.46 %</td>
</tr>
<tr>
<td>Shear stress</td>
<td>-</td>
<td>79.028 MPa</td>
<td>-</td>
</tr>
<tr>
<td>34Equivalent elastic strain</td>
<td>-</td>
<td>0.0014513</td>
<td>-</td>
</tr>
</tbody>
</table>
The values of the stresses considered here only the maximum values of the equivalent Von Mises stress, equivalent elastic strain and total deformation. Weight reduction is also considered in this optimization.

### Weight optimization

<table>
<thead>
<tr>
<th>Name</th>
<th>Original</th>
<th>Optimized</th>
<th>Percentage Variation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Weight</td>
<td>0.66246kg</td>
<td>0.64315 kg</td>
<td>2.91 %</td>
</tr>
</tbody>
</table>

### III. CONCLUSION

1. On the basis of the current work, it is concluded that the design parameter of the yoke with modification gives sufficient improvement in the existing results.

2. The weight of the yoke is also reduced by 0.01931 kg. Thereby, reduces the cost of the material.

3. The stress is found maximum near the cylindrical support. This can be reduced by increasing the material at the fork near the prongs.

### REFERENCES


