A Survey on Torsional Analysis of Hooke’s Joint

Prafulla R. Patil1, Dr. S. I. Kolhe2

1 PG Student, J. T. M., COET, Faizpur, Jalgaon (MS), India.
2 Assistant Professor, J. T. M., COET, Faizpur, Jalgaon (MS), India.

Abstract- A universal joint, (universal coupling, U-joint, Cardan joint, Hardy-Spicer joint, or Hooke's joint) is a joint or coupling in a rigid rod that allows the rod to 'bend' in any direction, and is commonly used in shafts that transmit rotary motion. It consists of a pair of hinges located close together, oriented at 90° to each other, connected by a cross shaft. The universal joint is not a constant velocity joint. An automotive drive train is an assembly of one or more driveshaft, hook's joint, and slip joint that forms the connection between the transmission and the drive axle. The function of drive train is that it allows the driver to control the power flow, speed and multiple the engine's torque.

The present study includes a study on hook joint. The literatures have been discussed in this paper and several sample calculations are taken for variation of torque. The model is created in CATIA and analyze in ANSYS 15.

Keywords-- hook's joint, constant velocity joint, CATIA, ANSYS 15, etc.

I. INTRODUCTION

In day-to-day life every aspect is influenced by the work of engineer. The equipment we use, the food we eat, and the vehicles we travel in and many more all are developed with the assistance of design engineering. 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 procedures. As product performance and serviceability becomes more important and as designs becomes more complex the simple method have becomes more inadequate. To understand the growth and its implication for design, it is necessary to look at how design solutions are implemented. To satisfy the market needs it is necessary to provide a computational capacity along with the creativity of the human being.

An all-inclusive joint (U-joint) is a joint in an unbending bar that allows the pole to climb and down while turning with a specific end goal to transmit power by changing the point between the transmission yield shaft and the driveshaft as appeared in Figure1. The most widely recognized sorts of U-joint utilized as a part of car industry is Hooke or Cardan joint (Birch and Rockwood, 2005). An essential U-joint comprises of driving burden, driven burden, creepy crawly and trunnions.

Every association part of the bug and trunnion are amassed in needle bearing together with the two burdens. The driving burden compels the insect to pivot the other two trunnions. The past activity causes the determined burden to turn.

Figure 1 Typical Hooke’sJoint

II. LITERATURE REVIEW

A] History-

The fundamental idea of the widespread joint depends on the outline of gimbals, which have been being used since artifact. One foresight of the widespread joint was its utilization by the old Greeks on ballistae. In Europe the widespread joint is frequently called the Cardano joint or Cardan shaft, after the Italian mathematician Gerolamo Cardano; However, in his compositions, he specified just gimbal mountings, not all inclusive joints. The system was later depicted in Technica curiosa sive mirabilia artis (1664) by Gaspar Schott, who erroneously guaranteed that it was a steady speed joint. In no time a while later, somewhere around 1667 and 1675, Robert Hooke investigated the joint and found that its pace of pivot was non uniform, yet that this property could be utilized to track the movement of the shadow on the face of a sundial. Actually, the segment of the condition of time which represents the tilt of the tropical plane with respect to the ecliptic is completely undifferentiated from the scientific depiction of the all-inclusive joint. The initially recorded utilization of the term general joint for this gadget was by Hooke in 1676, in his book Helioscopes. He distributed a portrayal in 1678, bringing about the utilization of the term Hooke’s joint in the English talking world.
In 1683, Hooke proposed an answer to the non-uniform rotating rate of the general joint: a couple of Hooke's joints 90° out of stage at either end of a middle of the road shaft, a plan that is presently known as a sort of consistent speed joint. Christopher Polhem of Sweden later reevaluated the widespread joint, giving ascent to the name Polhemsknut in Swedish.

In 1841, the English researcher Robert Willis dissected the movement of the widespread joint. By 1845, the French architect and mathematician Jean Victor Poncelet had dissected the development of the all-inclusive joint utilizing circular trigonometry. The term all inclusive joint was utilized as a part of the eighteenth century and was in like manner use in the nineteenth century.

B) Review from Authors

H. Bayrakceken, in his paper infers that two instances of weariness disappointments of segments in the force transmission arrangement of traveler autos are considered in this study. A few test and numerical examinations are completed. As a conclusion, the taking after focuses can be drawn:

1. Both disappointments are happened as a consequence of weariness procedure.
2. The split starting area of the joint burden compares to most astounding emphasize focuses. A few change on the configuration of the joint might be considered for avoidance of later disappointments.
3. The drive shaft disappointment is by all accounts started from disgraceful warmth treatment conditions, however the fizzled segment and the break starting areas additionally correspond with the exceedingly focused on districts. The gentle anxiety focuses additionally accelerate the disappointment. [1]

Dhwani V. Sanghani1, in his work design & finite element analysis of hooke’s coupling is carried out. The failure of component is occur due to manufacturing and design fault, shear failure, improper assembly, raw material faults, maintenance faults, material processing faults, drivable joint angle, cyclic load, wear, noise etc. The main objective of this work is to reduce shear failure. The modeling of proposed design is done by using CREO software & static and dynamic analysis is done in ANSYS software. In existing design von mises stress and shear stress are 704.71 MPa & 351.3 MPa respectively. After the modification in pin’s design von mises stress and shear stress are reduced to 241.46 MPa & 120.04 MPa respectively. By the comparison of both the result it is found that the von mises stress is reduced from 704.71MPa to 241.46MPa & shear stress is reduced from 351.3MPa to 120.04MPa. So shear failure is automatically reduced. [2]

Dhananjay S Kolekar, obtained the result are quite favorable which was expected. Finite element method is effectively utilized for addressing the conceptualization and formulation for the design stages. The stresses derived during analysis phase normally indicate the potential solution. The iterations are carried out in the analysis phase which yields the suitable values for design parameter. To improve performance, geometry has been modified using topology and free size optimization which enables to reduce stress level marginally well below the yield limit. [3]

Ritesh P. Neve, found that the use of composite material reduces the weight of joint significantly as composite having lower density. Hooke’s joint is a joint in a rigid rod that allows the rod to bend in any direction, and is commonly used in shafts that transmit rotary motion. It generally consists of two hinges located close together, oriented at 90 degree to each other, connected by a cross shaft. It is widely used in industrial applications and vehicle drivelines to connect misaligned shaft. Automobile industries are exploring composite material in order to obtained reduction of weight without significant decrease in vehicle quantity and reliability. This is due to the fact that the reduction of weight of vehicle directly proportional to fuel consumption. Hooke’s joint yoke have certain modification are made in existing geometry and analyzing for the identical boundary condition. Initial torque required to give rotation to the transmission system is large as the weight reduces this surplus torque is utilizes. The reduction in weight gives further advantages in increase in fuel economy of vehicle. In this study analysis is being perform on hooke’s joint .in this joint certain modification are made in the existing geometry and analyzes for the identical loading and boundary condition. Universal joint will be analyzing in the ANASYS and result will be compared. [4]

Siraj Mohammad Ali Sheikh, shows the power produced from an engine of automobile can be transferred to the drive wheel by power transmission system. Each automobile has different power transmission system constructive features depend on the vehicle’s driveline concept. (H.Bayrakceken et al., 2006) To transmit the driving torque from the engine or gear unit to the wheels, most of passenger car and light vehicle driven by combustion engine has at least two driveshaft as a basic requirement (Amborn, P. 1995). During operation, torsional stress and bending stress was experienced by driveshaft due to the weight of the car or misalignment of journal bearing (Asi, 2006). In order to meet the requirements of one of the most highly stressed components in automotive assembly, a failure investigation must be conducted.
Finite element method was used as stress analysis to determine the stress conditions at the failed section. [5]

III. PROPOSED METHODOLOGY

Traditional approach to design analysis involves the application of classical or analytical techniques. This approach has the following limitations:

i. Stresses and strains are obtained only at macro level. This may result in inappropriate deployment of materials. Micro level information is necessary to optimally allocate material to heavily stressed parts.

ii. Adequate information will not be available on critically stressed parts of the components.

iii. It may be necessary to make several simplifications and assumptions to design complex components and systems, if design analysis is carried out in the conventional manner.

iv. Manual design is time consuming and prone to errors.

v. Design optimization is tedious and time consuming.

FEA is a convenient tool to analyze simple as well as complex structures. The use of finite element analysis is not restricted to mechanical engineering systems alone. FEA finds extensive application in electrical engineering, electronics engineering, micro electro mechanical systems, biomedical engineering etc. In manufacturing, FEA is used in simulation and optimization of manufacturing processes like casting, machining, plastic molding, forging, metal forming, heat treatment, welding etc. Structural, dynamic, thermal, magnetic potential and fluid flow problems can be handled with ease and accuracy using FEA.

FEA was initially developed in 1943 by R. Courant to obtain approximate solution to vibration problems. Turner et al published in 1956 a paper on “Stiffness and Deflection of Complex Structures”. This paper established a broader definition of numerical analysis as a basis of FEA. Initially, finite element analysis programs were mainly written for main frame and mini computers. With the advent of powerful PC’s, the finite element analysis could be carried out with the help of several FEA software packages. Finite element method can be applied to a variety of design problems concerning automobiles, airplanes, missiles, ships, railway coaches and countless other engineering and consumer products.

AI Structural Analysis of Hooke’s Joint

Finite element analysis has been carried out by ANSYS 15 software. ANSYS is a general-purpose finite-element modeling package for numerically solving a wide variety of mechanical problems. These problems include static/dynamic, structural analysis (both linear and nonlinear), heat transfer, and fluid problems, as well as acoustic and electromagnetic problems.

Here the hook joint is discretized into approximately 8671 elements having 15700 nodes by default meshing of tetrahedral and mesh size is 2 mm. The solution is obtained by using ANSYS Work Bench modeler. The maximum shear stress are found out and from the resulted data, the comparison has made among the all types of torque. The results are tabulated in Table 1.

Following steps show the guidelines for carrying out Structural Analysis.

Define Materials
1. Set preferences. (Structural steel)
2. Define constant material properties.
   (Density = 7850 Kg/m$^3$, Young’s modulus = 207GPa, Poisson’s ratio =0.3)
Model the Geometry
3. Follow bottom up modelling and create/import the geometry.
Generate Mesh
4. Define element type. (Default mesh of element size 2 mm)
5. Mesh the area.
Apply Boundary Conditions
6. Apply constraints to the model. (Fixed support and Moment)
Obtain Solution
7. Specify analysis types and options. (Structural ANSYS for maximum Shear Stress and total deformation)
8. Solve

The ANSYS 15 finite element program was used for Structural analysis of hooke’s joint. For this purpose, the model is created in CATIA software and imported in ANSYS Workbench modeler for the structural analysis. The type of model depends on boundary conditions. Hence 7 models are selected depending on the torque transmitting capacity of the hooke’s joint.

The figure 2 to figure 8 represents the maximum shear stress analysis calculation of all types of hooke’s joint structure. The table 1 shows the tabulated data of finite element formulation for all types of models.
Figure 2 Maximum Shear Stress for 100 Nm

Figure 3 Maximum Shear Stress for 150Nm

Figure 4 Maximum Shear Stress for 200Nm

Figure 5 Maximum Shear Stress for 250Nm

Figure 6 Maximum Shear Stress for 300 Nm

Figure 7 Maximum Shear Stress for 350Nm
Figure 8 Maximum Shear Stress for 400Nm

Table 1
Tabulated Data for Various Models by ANSYS

<table>
<thead>
<tr>
<th>Sr. No.</th>
<th>Model</th>
<th>Torque Rating (Nm)</th>
<th>Max. Shear Stress (N/mm²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Model 1</td>
<td>100</td>
<td>146.11</td>
</tr>
<tr>
<td>2</td>
<td>Model 2</td>
<td>150</td>
<td>219.17</td>
</tr>
<tr>
<td>3</td>
<td>Model 3</td>
<td>200</td>
<td>292.22</td>
</tr>
<tr>
<td>4</td>
<td>Model 4</td>
<td>250</td>
<td>365.28</td>
</tr>
<tr>
<td>5</td>
<td>Model 5</td>
<td>300</td>
<td>438.33</td>
</tr>
<tr>
<td>6</td>
<td>Model 6</td>
<td>350</td>
<td>511.39</td>
</tr>
<tr>
<td>7</td>
<td>Model 7</td>
<td>400</td>
<td>573.34</td>
</tr>
</tbody>
</table>

IV. CONCLUSION

The presented work includes torsional analysis of a hooke’sjoint. The hooke’s joint can be loaded under range of torque. The comparison for the models are tabulated in table 1. The following conclusions can be drawn from this work.

- The work shows the importance of design, analysis and experimentation for designing any machine component.
- The hooke’s joint can be loaded to measure maximum shear stress in order to avoid yielding.
- As the torque on the hooke’s joint increases, simultaneously shear stress is increases.
- FEM reduces the effective time interval for the analytical as well as experimental method in order to design the joint.
- Finite element technique is more convenient to use because of implementation of computer processor to perform the task.
- It is better to design the model in solid modeling software and then make analysis in any one of the ANSYS Package to analyze the stresses than moving for analytical or experimental work which are time consuming.

REFERENCES