Modal analysis of drive shaft using FEA

Ravikant¹, Gopal Krishan², Mukesh Didwania³
¹,²,³Department of MAE, Amity University Haryana

ABSTRACT

The objective of the drive shaft is to connect with the transmission shaft with the help of universal joint whose axis intersects and the rotation of one shaft about its own axis results in rotation of other shaft about its axis. The model of drive shaft has been generated in Solidworks and then imported in ANSYS workbench. In this work finite element analysis of a drive shaft has been taken as a case study. In the present work the modal analysis of a drive shaft has been carried out to obtain the inherent frequencies and vibration mode shapes with their respective deformation. The maximum stress point and dangerous areas are found by the deformation analysis of drive shaft. The relationship between the frequency and vibration modal is explained by the modal analysis of drive shaft.

Keywords — Drive shaft, Finite Element Analysis, ANSYS workbench.

I. INTRODUCTION

As this work is based on finite element analysis, so it is required that a component on which analysis is to be done should have practical application. The component chosen for this purpose is a drive shaft which finds widespread applications in all vehicles. The drive shaft was generated in Solidworks and imported in ANSYS workbench. Universal joint is a form of connection between two shafts, whose axes intersect. The rotation of one shaft about its own axis results in the rotation of the other shaft about its axis. The main objective of this work is to perform the Finite Element Analysis of drive shaft using CAE Tools, so as to determine the total deformation and stress distribution in the shaft. The deformation and stress contours have been plotted and patterns are studied. The results are compared and verified with available existing results. The optimization of drive shaft also achieve the reduction in the weight of the assembly of universal joint and thus reduction in cost. There is a vast amount of literature related to Finite Element Analysis is present. The literature review presented here considers the major development in implementation of FEA. In the study of structural dynamics, experimental modal analysis has emerged as an extremely useful procedure. Performed under controlled conditions, it encompasses excitation of a structure, or component; acquisition of data; and the subsequent analysis of the response. The uses of modal analysis are varied and range from determination of natural frequencies and damping factor to full development of a mass-spring-damper model of a particular system.

II. METHODOLOGY

FINITE ELEMENT METHOD (FEM)

Finite Element method (FEM) simulates a physical part or assembly’s behaviour by dividing the geometry of the part into a number of elements of standard shapes, applying loads and constraints, then calculating variables of interest – deflection, stresses, temperature, pressures etc. The behaviour of an individual element is usually described by a relatively simple set of equations. Just as the set of elements would be joined together to build the whole structure, the equation describing the behaviour of the individual elements are joined into a set of equations that describe the behaviour of the whole structure. FEM is

- A numerical method
- Mathematical representation of actual problem
- Approximate method

Definition of FEM is hidden in the world itself. Basic theme is to make calculation at only limited number of points and then interpolate the result for entire domain (surface & volume).

Finite- any continuous object has finite degree of freedom & it’s just not possible to solve in this format. Finite Element Method reduces degree of freedom from infinite to finite with the help of discretization i.e. (nodes & elements).

Element- all the calculations are made at limited number of points known as nodes. Entity joining nodes and forming a specific shape such as quadrilateral or triangular etc. is known as Element. To get value of variable (say displacement) at where between the calculation points, interpolation function (as per the shape of element) is used.

Method- There are three methods to solve any engineering problem. Finite Element Analysis belongs to numerical method category. A Finite Element program takes the elements you have defined, lists the equations for each unknown value, puts them together as a matrix equation, and then solves all these for the values of the unknown parameters.
The equilibrium equation is of the form:

\[ [K] \times [X] = [F] \]

Since it's analogous to the equations of spring deflection, \( K \) is often called stiffness matrix, \( X \) is called the deformation matrix, and \( F \) is called the load matrix. \( K \) is a square matrix, with one row and column for each unknown variable in the problem definition. If there are 100 nodes in a model, and each node has 6 unknowns, then stiffness matrix would be 600×600. \( X \) and \( F \) are each column-matrix which has 1 column and 600 rows.

G. M. E. Cooke [1] explained an introduction to the Mechanical Properties of Structural Steel at Elevated Temperatures. Cooke presents data on the elevated temperature mechanical properties of hot rolled structural steel used in buildings and explains their physical meaning. The properties include Poisson's ratio, thermal expansion and phase transformation, stress-strain relationships, and elastic modulus. Some room temperature data are given as benchmarks. It also attempts to explain some anomalies in elastic modulus measurements and the difficulties in idealizing stress-strain data.

R. Paoluzzi, G. Rigamonti and L. G. Zarotti [2] explained the simulation studies of vehicle-transmission interactions. Within the frame of dynamic simulation techniques, the paper describes a vehicle locomotion model, inclusive of the internal power train and the soil-wheel interface. Two application areas are given as examples. The first refers to hydrostatic transmission sizing. The second treats the system response to fast transients due to external inputs or loads. Dynamic simulation is gaining popularity as an additional tool available to designers to improve the characteristics of physical systems. The recognized benefits of the approach are financial (by saving one or more prototypes and by cheaper planning of experimental tests), and related to performance (better understanding of the system behaviour, study of special working conditions or faults and analysis of the system sensitivity to the design parameters). An additional benefit is less recognized (i.e. the ability of simulation to integrate contributions from different specialized fields). In actual fact, this is the case with vehicle locomotion, a complex problem which involves two main fields (i.e. soil-vehicle interaction and power transmission). It is easy to understand that they interact and influence each other, but experience shows that their developments are almost independent. Consequently, the simulation environment seems to offer a promising opportunity of synthesis and synergy.

A. M. Heyes [3] 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 [4] 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 stubbing 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.

Goksenli, I.B. Eryurek [5] described the Failure analysis of an elevator drive shaft. In this study failure analysis of an elevator drive shaft is analyzed. By stress analysis, minimum and maximum normal shear stress values occurring at the fracture surface during operation is investigated. Due to too long length of shaft only keyway and fracture region is modelled and analysed. At first forces and torque acting on the shaft are determined. To examine stress distribution at the keyway and fracture surface, finite element method was applied. After visual investigation of the fracture surface it is concluded that fracture occurred due to torsional-bending fatigue. Fatigue crack initiated at the keyway edge. Considering elevator and driving systems, forces and torques acting on the shaft are determined; stresses occurring at the failure surface are calculated. Stress analysis is also carried out by using finite element method (FEM) and the results are compared with the calculated values. Endurance limit and fatigue safety factor is calculated, fatigue cycle analysis of the shaft is estimated. Reason for failure is investigated and concluded that fracture occurred due to faulty design or manufacturing of the keyway (low radius of curvature at keyway corner, causing high notch effect).

### III. MATERIAL PROPERTIES

<table>
<thead>
<tr>
<th>Material selected</th>
<th>Structural steel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s Modulus,( (E) )</td>
<td>( 2.0 \times 10^5 ) MPa</td>
</tr>
<tr>
<td>Poisson’s Ratio</td>
<td>0.30</td>
</tr>
<tr>
<td>Tensile Ultimate strength</td>
<td>460MPa</td>
</tr>
<tr>
<td>Tensile Yield strength</td>
<td>250 MPa</td>
</tr>
<tr>
<td>Compressive yield strength</td>
<td>250MPa</td>
</tr>
<tr>
<td>Density</td>
<td>7850kg/m3</td>
</tr>
<tr>
<td>Behavior</td>
<td>isotropic</td>
</tr>
</tbody>
</table>

### IV. CAD MODEL

The cad model of the drive shaft was generated in Solidworks and imported in Ansys workbench in parasolid format.
V. MESHED MODEL

The meshed model of drive shaft is shown in fig. 2. The number of elements is equal to 29484 and number of nodes equal to 11069.

VI. RESULTS

The vibration characteristic of drive shaft was obtained with modal analysis. Six mode shapes of drive shaft were found along with their natural frequencies and the deformation distribution. The simulation results show that the frequency goes on increasing from first mode shape to last mode shape. The minimum frequency was found in first mode shape i.e. 44.051 Hz and maximum deformation occurs in fifth mode shape at an frequency of 714.54 Hz.
VII. CONCLUSION

The maximum stress point and dangerous areas are found by the deformation analysis of drive shaft. The relationship between the frequency and the vibration modal is explained by the modal analysis of drive shaft. The simulation results show that the frequency goes on increasing from first mode shape to last mode shape. The minimum frequency was found in first mode shape i.e. 44.051 Hz and maximum deformation occurs in fifth mode shape at an frequency of 714.54 Hz. From the simulation results it is clear that maximum deformation occurs at the free end. Base on the results, we can forecast the possibility of mutual interference between the drive shaft and other parts. The resonance vibration of system can be avoided effectively by appropriate structure design. The results provide a theoretical basis to optimize the design and fatigue life calculation.

REFERENCES

[6] ANSYS 12 Workbench