Vol.2, No.1 ISSN Number (online): 2454-9614

Proceedings of International Conference on Recent Trends in Mechanical Engineering-2K15(NECICRTME-2K15), 20th – 21st November,2015

STRESS ANALYSIS ON DRIVE SHAFT

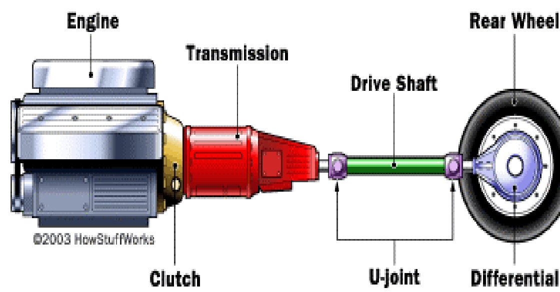
Sk.Farook Jilani [1], K.Kiran Chand [2],

[1]M.Tech Student, Department of Mechanical Engineering, Narasaraopeta Engineering College, Narasaraopeta
[2]Asst.Professor, Department of Mechanical Engineering,
Narasaraopeta Engineering College, Narasaraopeta , AP,India
[1]farookjilanishaik@gmail.com,kkchand415@gmail.com.[2]

***ABSTRACT* - In current market, drive shaft is the most important component to any power transmission application; automotive drive Shaft is one of this. A drive shaft, also known as a propeller shaft or Cardan shaft, it is a mechanical part that transmits the torque generated by a vehicle's engine into usable motive force to propel the vehicle. Physically, it is tubular in design, with an outside and inside diameter, which spins at a frequency governed by engine output. Drive shaft must operate in high and low power transmission of the fluctuating load. Due this fluctuating load it becomes fail and tends to stop power transmission. Thus it is important to make and design this shaft as per load requirement to avoid failure. Now a day's two pieces steel shaft are mostly used as a drive shaft. The two-piece steel drive shaft consists of three universal joints, a center supporting bearing and a bracket, which increases the total weight of an automotive vehicle and decreases fuel efficiency. However, in this project work an attempt is made to evaluate the suitability of composite material for the purpose of automotive drive shaft application. A Static and Dynamic analysis, composite shaft is analyzed using Finite Element Analysis Software for composites with the objective of minimizing the weight of the shaft, which is subjected to the constraints such as torque transmission, critical buckling torque capacity and also we are modifying the geometric shape to improve efficiency**

1. Introduction:

A driveshaft is the connection between the transmission and the rear axle of the car. As shown in Figure 1, power generated by the engine is transferred to the transmission via a clutch assembly. The transmission is linked to the driveshaft by a yoke and universal joint, or u-joint, assembly. The driveshaft transmits the power to the rear end through another yoke and u-joint assembly. The power is then transferred by the rig and pinion or rear differential to the rear wheels.



(1)

The entire driveline of the car is composed of several components, each with rotating mass. The rule of thumb is that 17-22% of the power generated by the engine is lost to rotating mass of the drive train. The power is lost because it takes more energy to spin heavier parts. This energy loss can be reduced by decreasing the amount of rotating mass. Light weight flywheels and transmission gears, aluminum and carbon-fiber drive shafts, riffle-drilled axels, and aluminum hubs are all examples of replacement or modified parts used to reduce the amount of rotating mass.

Power transmission can be improved through the reduction of inertial mass and light weight. Substituting composite structures for conventional metallic structures has many advantages because of higher specific stiffness and higher specific strength of composite materials. Composite materials can be tailored to efficiently meet the design requirements of spacecraft's, airplanes, automobiles, boats, sports' equipment's, bridges and buildings.

2. Design considerations:

The primary load carried by the drive shaft is torsion. The shaft must be designed to have enough tensional strength to carry the torque without failure. In addition, the possibility of tensional buckling must be considered for a thin-walled tube. The third major design requirement is that the drive shaft has a bending natural frequency which is sufficiently high. An optimum design of the drive shaft is desirable, which is cheapest and lightest but meets all of the above load requirements. Based on some reliable collected data the above three load-carrying requirements are summarized in Table 1.

3. LOAD REQUIREMENTS FOR DRIVE SHAFT DESIGN

REGULAR	VALUES	SAFETY FACTOR
Maximum torque	2020	3
Minimum buckling torque	>2020N-m	
Minimum flexural frequency	93.3	

4. MATHEMATICAL FORMULAE

The shearing stress (τ_{xy}) is defined and given by Equation

τ_{xy}==……………….(1)

Where T is the applied torque and R is the outer radius and r is the inner radius of the shaft, J is polar moment of inertia of the shaft.

The shear stress and yield stress is related using equation 2

τ_{xy}= ……………..(2)

Where σy is yield stress, and FOS is factor of safety

The critical torsional buckling torque, Tb is given by Equation (3)

$T_b=(2r^2{}_m t)(0.272)(E_x\times E_y{}^3)^{0.25}()^{1.5}$ …………………..(3)

Where t is the overall wall thickness, rm is the mean radius, and Ex and Ey are the average in-plane elastic moduli in the axial and transverse directions respectively. The drive shaft is idealized as a pinned-pinned beam. The lowest natural frequency is calculated using the Equation (4).

F_n= = ……………………(4)

Where, f_n is the lowest natural frequency in hertz. W/g = m is the mass per unit length, I, is the moment of inertia and L is the length of the drive shaft. The critical speed of the shaft (N) and natural frequency (f_n) are related by using equation (5)

N=60 f_n……………………(5)

5.Composites: Composite materials (or composites for short) are engineered materials made from two or more constituent materials with significantly different physical or

Chemical properties which remain separate and distinct on a macroscopic level within the finished structure.

The most primitive composite materials were straw and mud combined to form bricks for building construction. The most advanced examples perform routinely on spacecraft in demanding environments. Those composites closest to our personal hygiene form our shower stalls and bath tubs made of fiberglass. Solid surface, imitation granite and cultured marble sinks and counter tops are widely used to enhance our living experiences.

Composites are materials created by combining dissimilar materials with a view to improve the properties or to create materials with desired properties

S.No	Material	Specific Gravity (gm/cc)	Tensile strength (Gpa)	Elastic modulus (Gpa)	Specific strength
1	Aluminum alloy	2.8	0.46	72	0.16
2	Titanium	4.5	0.93	110	0.21
3	Steel alloy	7.8	0.99	207	0.13
4	E-glass/epoxy	0.45	2.54	140	0.43

Comparison of the properties of metals and FRC

Importance of the composite materials:

A material is any substance employed in making some useful thing or artifact. The metals and ceramics are materials used in industries as good conductors and refractory materials respectively. Insulation was based on natural products such as wood, paper, shellac and gutter percha. Since early 1900's one of the most fruitful development in materials has been the discovery and exploitation of synthetic of high polymers.

The polymers are substances, which consists of long chains or networks, built up by the repeated linkage of small reactive molecules. Each of the simple unit is called a 'mer' hence a polymer is simply the result of joining together of many identical units. Depending on the detailed chemistry and the spatial arrangement of the chains and their cross-links, the following can be produced.

Fiber reinforced polymers

Fiber-reinforced polymers or FRPs include wood (comprising cellulose fibers in a lignin and hemicelluloses matrix), carbon-fiber reinforced plastic or CFRP, and glass-reinforced plastic or GRP. If classified by matrix then there are thermoplastic composites, short fiber thermoplastics, long fiber thermoplastics or long fiber-reinforced thermoplastics. There single layer/ply/lamina

6.INTRODUCTION TO CAD & CATIA-V5

INTRODUCTION

Computer Aided Design (CAD) is a technique in which man and machine are blended in to problem solving team, intimately coupling the best characteristics of each. The result of this combination works better than either man or machine would work alone , and by using a multi discipline approach, it offers the advantages of integrated team work.

GEOMETRIC MODELING:

A geometric modeling should be an unambiguous representation of its corresponding object. The model should be unique and complete to all engineering functions from documentation to engineering analysis to manufacturing .

.

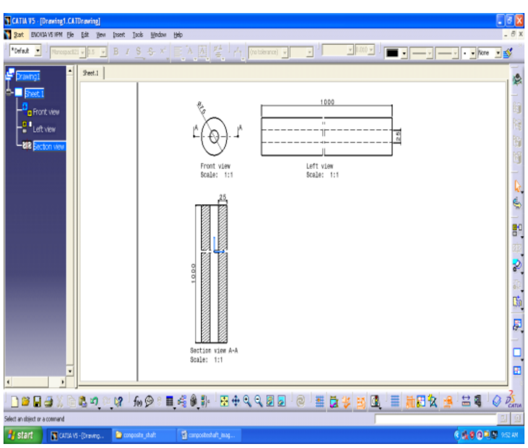
SOLID MODELING:

A solid model of an object is a completed representation of the object. This model is capable of complex geometry data representation that is the art completely defined ,solid modeling techniques based on information ally complete, valid and unambiguous of object solid modelers store more information (geometry and topology) than wire frame modelers of surface (geometry only). Both wire frame and surface modelers are incapable of handling special address ability as well as verifying that the model is well framed or not. Solid models can be quickly created without having to define individual locations as with wire frames. Solid modeling produces accurate designs, provides complete three-dimensional improves the quality of the design, improves and has potential for functional automation and integration.

BOUNDARY REPRESENTATION:

Boundary representation is one of the most popular and widely used schemes to create solid of physical objects. Boundary representation of object in term of its surface boundaries like vertices, edges and faces . B-reps model is based on the topological notation that a physical object is bounded by a set of faces . These faces region (or) subsets of closed and oriental surface .Each face is bounded by edges and each edge has 2 vertices. Thus topologically a boundary model of any object is comprised of faces , edge and vertices of an objects linked together in such a way as to ensure the topological consistency of the model. The database of bounded model contains both is topology and geometry performing Euler operations creates topology and performing Euclidean calculations creates geometry.

DRAWING OF SHAFT:



Model of shaft with 70 diameter:

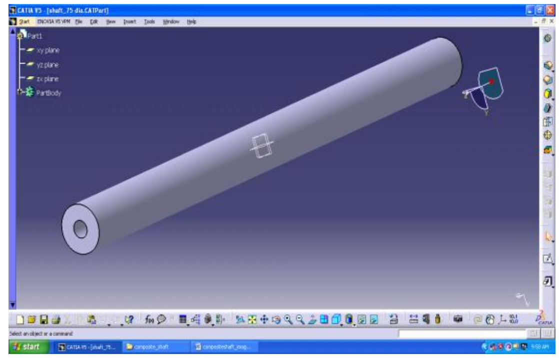


Fig 1.2

7. INTRODUCTION TO FEA

Finite Element Analysis (FEA) was first developed in 1943 by R. Courant, who utilized the Ritz method of numerical analysis and minimization of variation calculus to obtain approximate solutions to vibration systems. Shortly thereafter, a paper published in 1956 by M. J. Turner, R. W. Clough, H. C. Martin, and L. J. Topp established a broader definition of numerical analysis. The paper centered on the "stiffness and deflection of complex structures".

By the early 70's, FEA was limited to expensive mainframe computers generally owned by the aeronautics, automotive, defense, and nuclear industries. Since the rapid decline in the cost of computers and the phenomenal increase in computing power, FEA has been developed to an incredible precision. Present day supercomputers are now able to produce accurate results for all kinds of parameters.

There are generally two types of analysis that are used in industry: 2-D modeling, and 3-D modeling. While 2-D modeling conserves simplicity and allows the analysis to be run on a relatively normal computer, it tends to yield less accurate results. 3-D modeling, however, produces more accurate results while sacrificing the ability to run on all but the fastest computers effectively. Within each of these modeling schemes, the programmer can insert numerous algorithms (functions) which may make the system behave linearly or non-linearly. Linear systems are far less complex and generally do not take into account plastic deformation. Non-linear systems do account for plastic deformation, and many also are capable of testing a material all the way to fracture.

Types of Engineering Analysis

Structural analysis consists of linear and non-linear models. Linear models use simple parameters and assume that the material is not plastically deformed. Non-linear models consist of stressing the material past its elastic capabilities. The stresses in the material then vary with the amount of deformation as in.

Vibrational analysis is used to test a material against random vibrations, shock, and impact. Each of these incidences may act on the natural vibrational frequency of the material which, in turn, may cause resonance and subsequent failure.

Fatigue analysis helps designers to predict the life of a material or structure by showing the effects of cyclic loading on the specimen. Such analysis can show the areas where crack propagation is most likely to occur. Failure due to fatigue may also show the damage tolerance of the material.

Heat Transfer analysis models the conductivity or thermal fluid dynamics of the material or structure. This may consist of a steady-state or transient transfer. Steady-state transfer refers to constant thermo properties in the material that yield linear heat diffusion.

Results of Finite Element Analysis

FEA has become a solution to the task of predicting failure due to unknown stresses by showing problem areas in a material and allowing designers to see all of the theoretical stresses within. This method of product design and testing is far superior to the manufacturing costs which would accrue if each sample was actually built and tested.

In practice, a finite element analysis usually consists of three principal steps:

- **Preprocessing**: The user constructs a model of the part to be analyzed in which the geometry is divided into a number of discrete sub regions, or elements," connected at discrete points called nodes." Certain of these nodes will have fixed displacements, and others will have prescribed loads. These models can be extremely time consuming to prepare, and commercial codes vie with one another to have the most user-friendly graphical "preprocessor" to assist in this rather tedious chore. Some of these preprocessors can overlay a mesh on a preexisting CAD file, so that finite element analysis can be done conveniently as part of the computerized drafting-and-design process.
- **Analysis**: The dataset prepared by the preprocessor is used as input to the finite element code itself, which constructs and solves a system of linear or nonlinear algebraic equations

$$K_{ij}u_j = f_i$$

8. INTRODUCTION TO ANSYS

INTRODUCTION

ANSYS is general-purpose finite element analysis (FEA) software package. Finite Element Analysis is a numerical method of deconstructing a complex system into very small pieces (of user-designated size) called elements. The software implements equations that govern the behaviour of these elements and solves them all; creating a comprehensive explanation of how the system acts as a whole. These results then can be presented in tabulated, or graphical forms. This type of analysis is typically used for the design and optimization of a system far too complex to analyze by hand. Systems that may fit into this category are too complex due to their geometry, scale, or governing equations.

ANSYS is the standard FEA teaching tool within the Mechanical Engineering Department at many colleges. ANSYS is also used in Civil and Electrical Engineering, as well as the Physics and Chemistry departments.

ANSYS provides a cost-effective way to explore the performance of products or processes in a virtual environment. This type of product development is termed virtual prototyping. With virtual prototyping

Types of Structural Analysis

Static Analysis--Used to determine displacements, stresses, etc. under static loading conditions. Both linear and nonlinear static analyses. Nonlinearities can include plasticity, stress stiffening, large deflection, large strain, hyper elasticity, contact surfaces, and creep.

Modal Analysis--Used to calculate the natural frequencies and mode shapes of a structure. Different mode extraction methods are available.

Harmonic Analysis--Used to determine the response of a structure to harmonically time-varying loads.

Transient Dynamic Analysis--Used to determine the response of a structure to arbitrarily time-varying loads. All nonlinearities mentioned under Static Analysis above are allowed.

Spectrum Analysis--An extension of the modal analysis, used to calculate stresses and strains due to a response spectrum or a PSD input (random vibrations).

Buckling Analysis--Used to calculate the buckling loads and determine the buckling mode shape. Both linear (eigenvalue) buckling and nonlinear buckling analyses are possible.

Explicit Dynamic Analysis--This type of structural analysis is only available in the ANSYS LS-DYNA program. ANSYS LS-DYNA provides an interface to the LS-DYNA explicit finite element program. Explicit dynamic analysis is used to calculate fast solutions for large deformation dynamics and complex contact problems.

Proceedings of International Conference on Recent Trends in Mechanical Engineering-2K15(NECICRTME-2K15), 20th – 21st November,2015

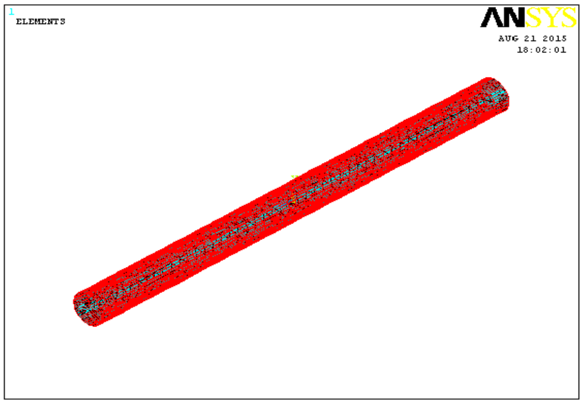


Fig 1.5
The above image is showing the loads applied on a shaft

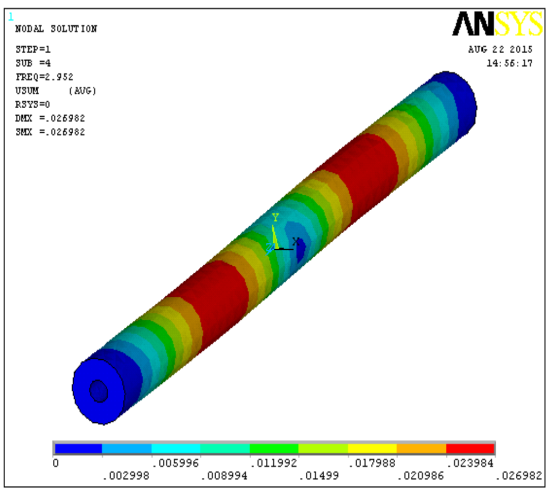


Fig 1.6

The above images are showing the mode shapes due to natural frequency of the object the values are as follows.

For Mode1=1.116
For Mode2=1.116
For Mode3=2.952
For Mode4=2.952

Harmonic analysis:

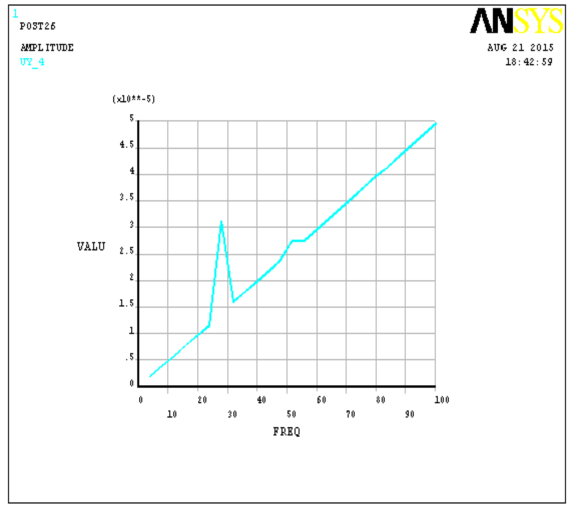


Fig 1.7

The above images are showing the graphs of displacement due to loads, natural frequency and external frequencies acting on composite shaft.

Model analysis:

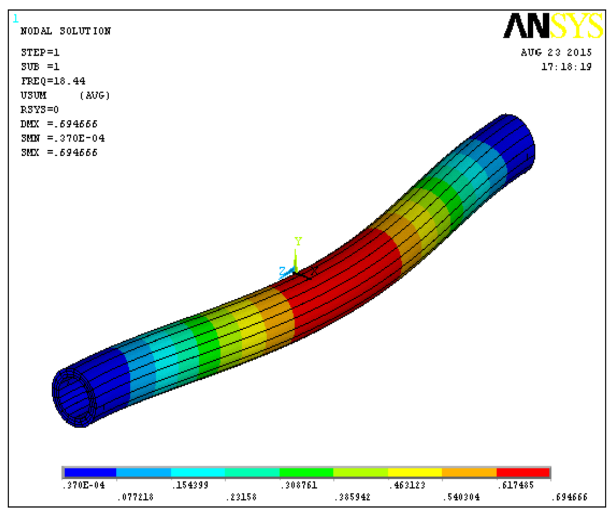


Fig 1.8

The above images are showing the mode shapes due to natural frequency of the object the values are as follows.

For Mode1=18.44

Results and Discussion

8.Results Table:

	75 diameter			
	displacement	**Stress**	**Frequency**	**Harmonic**
Steel	0.748	19.386	1.116	$0.9e^{-3}$
Carbon epoxy	$0.1e^{-3}$	0.77	17.408	$0.1e^{-3}$
E glass	$0.178e^{-3}$	0.785	11.387	$1.7e^{-4}$
ECE	$0.33e^{-7}$	-	17.53	-
CEC	$0.261e^{-7}$	-	20.543	-
	70 diameter			
	displacement	**Stress**	**Frequency**	**Harmonic**
Steel	$0.514e^{-4}$	0.779	10.525	$0.5e^{-6}$
Carbon	$0.106e^{-3}$	0.780	16.416	$0.1e^{-5}$
E glass	$0.173e^{-3}$	0.7971	10.735	$0.1125e^{-5}$
ECE	$0.285e^{-7}$	-	16.629	-
CEC	$0.227e^{-7}$	-	19.538	-
	75 diameter			
CEC	$0.127e^{-6}$	-	18.44	-

DISCUSSION

In this project we observe different materials which are suitable for composite shaft.

The 75 diameter shaft is suitable for shaft due to its structural stability and the composites having very less stress due to its construction of material structure and the material matrix.

The multiple layers with the variation of angles splitting the load.

CONCLUSION

In this paper we have done the analysis on composite drive shaft to increase the efficiency as well as to decrease the cost and weight.

In the first step we have analyzed existing models with existing materials steel, FRP (fiber reinforced polymer) and CRF (carbon reinforced fiber) as per previous papers.

In the next step we have chosen the materials which are suitable for shaft with the combination of FRP and CRF.

In the next step we have done the analysis on combination of FRP and CRF,

Carbon fiber as inner and outer, fiber polymer as middle core and fiber polymer as inner and outer, carbon fiber as middle core.

In the next step we have reduced the thickness of the shaft and analyzed with carbon fiber as inner and outer material using layers method with reinforcement angles.

As per the above results we can conclude that 75 dia with 12 mm wall thickness shaft with carbon fiber as inner and outer and fiber as middle core is suitable for drive shaft due to low stress, less weight and less manufacturing cost.

Using this type of shaft we can increase the mechanical efficiency by reducing the weight and this type of shafts are easy to manufacture and cost effective.

REFERENCES

1. Gay, D.; V. Hoa, S.; W. Tsai, S. (2004). Composite materials: design and application, CRC press.
2. Pollard, A.; (1999): Polymer matrix composite in drive line applications. GKN technology, Wolverhampton.
3. Rangaswamy, T.; Vijayrangan, S. (2005). Optimal sizing and stacking sequence of composite drive shafts. Materials science, Vol. 11
No 2., India.
4. Rastogi, N. (2004). Design of composite drive shafts for automotive applications. Visteon Corporation, SAE technical paper series.
5. Lee, D. G.; Kim, H. S.; Kim, J. W.; Kim, J. K. (2004). Design and manufacture of an automotive hybrid aluminum/composite driveshaft, composite structures, (63): 87-99.
6. Pappada, S.; Rametto R. (2002). Study of a composite to metal tubular joint. Department of Materials and Structures Engineering ,Technologies and Process, CETMA , Italy.