CHAPTER 5

FINITE ELEMENT ANALYSIS

5.1 INTRODUCTION

Finite element analysis was carried out for static, torsional and dynamic conditions for conventional steel shaft and composite material shafts using ANSYS. The analysis results were compared with analytical results obtained in Chapter 3 and 4. The primary unknowns in this structural analysis are displacements and the other quantities such as stresses, strains, and reaction forces, are then derived from the nodal displacements.

5.1.1 Defining Elements, Real Constants, and Materials

There are several steps that must be taken in order to properly define the composition of a composite structure within ANSYS. First is the selection of the element type. ANSYS offers five element types which are listed in Table 5.1, which can be used to define layered composites.

Due to the simple geometry of the model and the assumption of linear response in this work, the shell 99 element is selected. Shell 99 linear layered structural shell and FEA model of the composite drive shaft tube are shown in Figures 5.1 and 5.2. Once the element type has been selected, the material properties, layer orientation, and layer thickness must be defined within each element. In ANSYS these properties are set using real constants. Real constants are user-defined element characteristics, which represent the
configuration of the element. The first step is to define the materials that will be used in the model. Possible materials are accessed through the material models section of ANSYS. In this dialog window, the orthotropic mechanical properties can be set for any number of materials. In the real constant dialog window, the preliminary option is to define the number of layers in the model. Once the number of layers is specified, the composition of each individual layer is defined. Each layer can be represented by any one of the material models that have been defined. The layer orientation is defined as the direction of the layer coordinate system relative to the global coordinate system and the orientation is defined by entering the angle between the X-axes of each coordinate system. Finally, the thickness of each individual layer can be defined to meet the specifications of the composite. There may be as many real constants as are necessary to accurately represent the structure being modeled. The values of young’s modulus $E_2 = E_3$, rigidity modulus $G_{12} = G_{31}$, and Poisson’s ratio $\nu_{23} = \nu_{12}$ are assumed suitably.

**Table 5.1  List of layered element types in ANSYS**

<table>
<thead>
<tr>
<th>Type Name</th>
<th>Description Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shell 99</td>
<td>Linear Layered Structural Shell</td>
<td>8-node, 3-D shell, 6-DOF per node, up to 250 layers</td>
</tr>
<tr>
<td>Shell 91</td>
<td>Nonlinear Layered Structural Shell</td>
<td>8-node, 3-D shell, 6-DOF per node, up to 100 layers</td>
</tr>
<tr>
<td>Shell 181</td>
<td>Finite Strain Shell</td>
<td>4-node, 3-D shell, 6-DOF per node, up to 255 layers with large strain</td>
</tr>
<tr>
<td>Solid 46</td>
<td>3-D Layered Structural Solid</td>
<td>8-node, 3-D solid, 3-DOF per node, up to 250 layers</td>
</tr>
<tr>
<td>Solid 191</td>
<td>Layered Structural Solid</td>
<td>20-node, 3-D solid, 3-DOF per node, up to 100 layers</td>
</tr>
</tbody>
</table>
Figure 5.1 Shell 99 linear layered structural shell

Figure 5.2 Finite Element Model of composite shaft tube
5.1.2 Boundary Conditions

The boundary condition for static, torsional buckling, and harmonic analysis are fixed at one end and in the other end it is radially fixed and torque (tangential force) given to the other end nodes.

5.2 STATIC ANALYSIS

5.2.1 Introduction

Static analysis deals with the conditions of equilibrium of the bodies acted upon by forces. A static analysis can be either linear or non-linear. All types of non-linearities such as large deformations, plasticity, creep, stress stiffening, contact elements etc. are allowed. This chapter focuses on static analysis. A static analysis calculates the effects of steady loading conditions on a shaft structure, while ignoring inertia and damping effects such as those carried by time varying loads. A static analysis is used to determine the displacements, stresses, strains and forces in structures or components caused by loads that do not induce significant inertia and damping effects.

In static analysis, loading and response conditions are assumed, that is the loads and the structure responses are assumed to vary slowly with respect to time. The kinds of loading that can be applied in static analysis include

1. Externally applied forces, moments and pressures
2. Steady state inertial forces such as gravity and spinning
3. Imposed non-zero displacements
A static analysis result of structural displacements, stresses and strains and forces in structures for components caused by loads will give a clear idea about whether the structure or components will withstand for the applied maximum forces. If the stress values obtained in this analysis crosses the allowable values it will result in the failure of the structure in the static condition itself. To avoid such a failure, this analysis is necessary. In ferrous materials hardening, redesigning and replacing with stronger material is possible whereas in non-ferrous and composite materials, hardening is not possible. There is only one way i.e. redesigning and replacing the material with a stronger material.

This chapter deals with an analysis of stresses responsible for the failure of the composite shaft. The failure of the drive shaft occurs mainly due to shear stresses, as it will be mainly subjected to twisting. So static analysis must be carried out to find out whether it will withstand the static load. The static analysis aims at finding out shear stresses, deflections at different points. The maximum shear stress occurs at the point where it is fixed to another shaft that is connected to gearbox. The results of the study on composite materials are compared with those of steel drive shaft to assess the suitability of the composite materials for the drive shaft.

First for the aforementioned boundary conditions, analysis is performed with the prestress ‘ON’ condition for the Eigen buckling analysis. The results obtained are Vonmises stress and stresses in all the three planes. The comparison results of shear strength and design stresses of various material shafts are shown in Table 5.2 and Figure 5.4 presents the comparison results of stresses.
5.2.2 Static Analysis Results

Figure 5.3 Static analyses (Boron/ Epoxy shaft tube)

Table 5.2 Comparison of stresses

<table>
<thead>
<tr>
<th></th>
<th>Steel</th>
<th>E-Glass / Epoxy*</th>
<th>HM-Carbon/ Epoxy*</th>
<th>Boron/ Epoxy#</th>
<th>E-Glass/ Vinylester#</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear strength (MPa)</td>
<td>175</td>
<td>191</td>
<td>396</td>
<td>312</td>
<td>155</td>
</tr>
<tr>
<td>Design shear stress (MPa)</td>
<td>134</td>
<td>137</td>
<td>274</td>
<td>287</td>
<td>134</td>
</tr>
<tr>
<td>Factor of safety</td>
<td>1.30</td>
<td>1.39</td>
<td>1.44</td>
<td>1.08</td>
<td>1.15</td>
</tr>
</tbody>
</table>


5.3 TORSIONAL BUCKLING ANALYSIS

5.3.1 Introduction

Advanced composite materials seem ideally suited for long, power drive shaft applications. Their elastic properties can be tailored to increase the torque they can carry as well as the rotational speed at which they operate. For thin walled shafts, the failure mode under an applied torque is torsional buckling rather than material failure. On the other hand the rotational speed is limited by lateral stability considerations. Most designs are sub critical, i.e. rotational speed must be lower than the first natural bending frequency of the shaft. This frequency is proportional to $(E/\rho)^{1/2}$ where $E$ is the longitudinal stiffness modulus of the shaft and $\rho$ the material density. For lay-ups contain a
significant proportion of fibers running along the shaft’s axis, this ratio can be made larger than for metal shafts resulting in higher natural frequencies.

For a realistic drive shaft system, improved lateral stability characteristics must be achieved together with improved torque carrying capabilities. The dominant failure mode, torsional buckling is strongly dependent on fiber orientation angles and ply stacking sequence.

5.3.2 Torsional Buckling Analysis Results

After performing static analysis with pre-stress ON condition, next is to perform Eigen buckling analysis to get the critical torsional buckling load. From this, we are able get the torsional buckling strength of the shaft along with the buckling mode shapes. The comparison results of torsional buckling strengths of various material shafts are shown in Table 5.3. Figures 5.5 to 5.9 show the first torsional buckling mode shapes for steel, E-Glass/Epoxy, HM Carbon/Epoxy, Boron/Epoxy and E-Glass/Vinylester, and shaft tubes.

<table>
<thead>
<tr>
<th></th>
<th>Steel</th>
<th>E-Glass/ Epoxy</th>
<th>E-Glass/ Vinylester</th>
<th>HM Carbon/ Epoxy</th>
<th>Boron/ Epoxy</th>
</tr>
</thead>
<tbody>
<tr>
<td>Analytical</td>
<td>29856</td>
<td>4056</td>
<td>3910</td>
<td>5213</td>
<td>5825</td>
</tr>
<tr>
<td>FEA</td>
<td>27900</td>
<td>3812</td>
<td>3790</td>
<td>5065</td>
<td>5745</td>
</tr>
</tbody>
</table>
Figure 5.5  Torsional buckling mode of steel shaft tube

Figure 5.6  Torsional buckling mode of E-Glass/Epoxy shaft tube

Figure 5.7  Torsional buckling mode of HM-Carbon/Epoxy shaft tube

Figure 5.8  Torsional buckling mode of Boron/Epoxy shaft tube

Figure 5.9  Torsional buckling mode of E-Glass/Vinylester shaft tube
Variation of Torsional Buckling strength

In Figure 5.10, it is observed that the steel shaft is having more buckling strength and followed by Boron/Epoxy, HM Carbon/Epoxy, E-Glass/Epoxy and E-Glass/Vinylester material shafts. The buckling strength of the composite shafts is always less compared to steel shaft of the same geometry because these properties depend on the stiffness of the material.

5.3.3 Effect of Boundary Conditions on Torsional Strength

Variations of torsional buckling strength for different boundary conditions of composite drive shaft tubes are discussed below.
Various boundary conditions considered:

1. S-S without axial constraint
2. S-S with constraint
3. Clamped at both ends
4. One end clamped and other S-S without axial constraint

Figure 5.11 Effect of boundary conditions on torsional buckling strength of the shaft
In Figure 5.11, it is observed that boundary conditions have little influence on torsional buckling strength. For fully fixed boundary conditions the value of the torsional buckling strength is more compared to other boundary conditions. For fully fixed boundary conditions the value of the torsional buckling strength is more compared to other boundary conditions.

5.4 DYNAMIC ANALYSIS

In the dynamic analysis of the shaft, the modal analysis is performed along with the harmonic analysis. Here the critical speed of the shaft can be predicted for the given boundary conditions along with the amplitude of the vibration at any selected node.

5.4.1 Modal Analysis

5.4.1.1 Introduction

Any structure, submitted to an external excitation will deform and vibrate in a characteristic manner that should be known in advance. In fact, vibration is a combination of different modes of vibration defined by the frequency from which the vibration is generated, its shape, the fact that this mode is damped or not, the properties of rigidity, and the inertia characteristic of this mode. All these values are known as the modal parameters of the considered vibration mode and are the result of the original mechanical design.

Modes when properly damped are not considered dangerous, as oscillations disappear with time. However, should a structure be excited by a force whose frequency corresponds to one of its modes, the vibrations, instead of being damped down, will increase until the destruction of the structure. It is therefore essential to know the different modes of vibration, especially those,
which, because of their proximity, influence each other. So it is necessary to know the natural frequency of structure and its harmonics. With the approximate methods like FEA we can get only approximate value of natural frequency, which includes some errors also. The only way to determine the natural frequencies is experimentation using external excitations.

5.4.1.2 Free Vibration Analysis

When an elastic system free from external forces is disturbed from its equilibrium position it vibrates under the influence of inherent forces and is said to be in the state of free vibration. It will vibrate at its natural frequency and its amplitude will gradually become smaller with time due to energy being dissipated by motion. The main parameters of interest in free vibration are natural frequency and amplitude. The natural frequencies and the mode shapes are important parameters in the design of a structure for dynamic loading conditions.

Modal analysis is used to determine the vibration characteristics such as natural frequencies and mode shapes of a structure or a machine component while it is being designed. It can also be a starting point for another more detailed analysis such as a transient dynamic analysis, a harmonic response analysis or a spectrum analysis. Modal analysis can be used for a pre-stressed structure such as a spinning turbine blade. Another useful feature is modal cyclic symmetry, which allows reviewing the mode shapes of a cyclic symmetric structure by modeling just a sector of it.

Modal analysis in the ANSYS family of products is a linear analysis. Any non-linearities such as plasticity are ignored even if they are defined. There are several mode extraction methods in the modal analysis in ANSYS.
5.4.1.3 Different Types of Mode Extraction Methods in ANSYS

Different types of mode extraction methods in ANSYS are

- Subspace method - The subspace method is used for large symmetric eigen value problems. Several controls are available to control the subspace iteration process.

- Block Lanczos method - The Block Lanczos method is used for large symmetric eigen value problems. This method can be used for the same type of problems for which the subspace method can be used. But this method is used to achieve a faster convergence rate than subspace method. The Block Lanczos method uses the sparse matrix solver, overriding any solver specified via the ESOLV command.

- Power Dynamics method - The power dynamics method is used for very large models and is especially useful to obtain a solution for the first several modes to learn how the model will behave.

- Reduced method - The reduced method is faster than the subspace method because it uses reduced system matrices to calculate the solution. However, it is less accurate because the reduced mass matrix is approximate.

- Unsymmetric method– The unsymmetric method is used for problems with unsymmetrical matrices such as fluid-structure interaction problems.

- Damped method - The damped method is used for problems where damping cannot be ignored, such as bearing problems. Modal
analysis should be carried out on the drive shaft tube in order to see
that whether the natural frequency of the drive shafting system lies
within the specified range that is 0 to 90Hz. As explained above, the
natural frequency of the driver shafting system is one of the
important characteristics which determine the operating frequency of
the transmission system. So a free vibration analysis is carried on the
carbon/epoxy and glass/epoxy shafts to find out the natural
frequencies and their mode shapes.

5.4.1.4 Boundary Conditions

- To find the critical speed of the shaft, the boundary condition
  considered as pinned- pinned condition.

- At one end the central nodes (for steel two nodes and for all
  composite materials four nodes) are pinned. At the other end, the
  bottom most node is free in axial direction.

- The frequencies obtained are then multiplied by 60 to obtain critical
  speeds as material natural frequencies. The mode shapes for all
  material combinations are obtained to their corresponding critical
  speeds.

Figures 5.12 to 5.21 shown are first and second bending frequency
modes for steel and composite drive shafts.
5.4.1.5 Modal Analysis Results

Steel Drive shaft tube

Figure 5.12 First bending frequency mode of Steel drive shaft tube

Figure 5.13 Second bending frequency mode of Steel drive shaft tube

E-Glass/Epoxy drive shaft tube

Figure 5.14 First bending frequency mode of E-Glass/Epoxy drive shaft tube

Figure 5.15 Second bending frequency mode of E-Glass/Epoxy drive shaft tube
**Figure 5.16** First bending frequency mode of HM Carbon/Epoxy drive shaft tube

**Figure 5.17** Second bending frequency mode of HM Carbon/Epoxy drive shaft tube

**Figure 5.18** First bending frequency mode of Boron/Epoxy drive shaft tube

**Figure 5.19** Second bending frequency mode of Boron/Epoxy drive shaft tube
E-Glass/Vinylester drive shaft tube

Figure 5.20 First bending frequency mode of E-Glass/ Vinylester drive shaft tube

Figure 5.21 Second bending frequency mode of E-Glass/ Vinylester drive shaft tube

Table 5.4 Variation of critical speed (rpm)

<table>
<thead>
<tr>
<th></th>
<th>Steel</th>
<th>E-Glass/ Epoxy</th>
<th>HM Carbon/ Epoxy</th>
<th>Boron/ Epoxy</th>
<th>E-Glass/ Vinylester</th>
</tr>
</thead>
<tbody>
<tr>
<td>Analytical</td>
<td>9323</td>
<td>6515</td>
<td>9270</td>
<td>9522</td>
<td>5890</td>
</tr>
<tr>
<td>FEA</td>
<td>8650</td>
<td>5553</td>
<td>8580</td>
<td>8555</td>
<td>5688</td>
</tr>
</tbody>
</table>
In Figure 5.22, it is observed that E-Glass/Epoxy and E-Glass/Vinylester shafts have minimum amount of the critical speed compared to the other material shaft. Boron/Epoxy shaft has the highest critical speed. The bending natural frequency of the composite shafts are always less compared to steel shaft of the same geometry because these properties depend on the stiffness of the material and for steel, the stiffness is very high compared to composites.

5.4.2 Harmonic analysis

5.4.2.1 Introduction

Any sustained cyclic load will produce a sustained cyclic response (a harmonic response) in a structural system. Harmonic response analysis gives the ability to predict the sustained dynamic behavior of structures, thus enabling
to verify whether the designs will successfully overcome resonance, fatigue, and other harmful effects of forced vibrations or not. Harmonic response analysis is a technique used to determine the steady-state response of a linear structure to loads that vary sinusoidally (harmonically) with time. The idea is to calculate the structure's response at several frequencies and obtain a graph of some response quantity (usually displacements) versus frequency. "Peak" responses are then identified on the graph.

5.4.2.2 Three Solution Methods

Three harmonic response analysis methods are available: full, reduced, and mode superposition. The ANSYS/Professional program allows only the mode superposition method. Before we study the details of how to implement each of these methods, let us explore the advantages and disadvantages of each method.

The full method is the easiest of the three methods. It uses the full system matrices to calculate the harmonic response (no matrix reduction). The matrices may be symmetric or asymmetric. The advantages of the full method are:

- It is easy to use, because no need for choosing master degrees of freedom or mode shapes.
- It uses full matrices, so no mass matrix approximation is involved.
- It allows unsymmetrical matrices, which are typical of such applications as acoustics and bearing problems.
• It calculates all displacements and stresses in a single pass.

• It accepts all types of loads: nodal forces, imposed (non-zero) displacements, and element loads (pressures and temperatures).

• It allows effective use of solid-model loads.

5.4.2.3 Dynamics Options

• Forcing Frequency Range - The forcing frequency range must be defined (in cycles/time) for a harmonic analysis. Within this range, the number of solutions to be calculated must be specified.

• Damping - Damping in some form should be specified; otherwise, the response will be infinity at the resonant frequencies.

5.4.2.4 Specifications and Results

For harmonic analysis, the boundary condition is considered as same as the Static analysis condition that is the shaft is fully fixed at one end and radially fixed at the other end. The cyclic load of 100 N and the frequency range set as 0-2000 Hz in steps of 200Hz each is applied at each nodes and the harmonic response have been taken from harmonic analysis. The responses have been compared for different shaft materials. The response from harmonic analysis is obtained in terms of frequency (in Hz) vs. rotational deflection (in radians). The frequencies are multiplied with 60 to obtain the speed (in rpm). The rotational deflections are multiplied with 57.23 (i.e. 180/π) to obtain the deflection in degrees. The deflection in degrees vs. speed in rpm is shown in Figures 5.23 to 5.27.
The harmonic analysis results for steel shaft show that the maximum deflection at resonance conditions occur in the range of 0 to 2000 Hz frequencies and is maximum at 600 Hz (36000rpm) with amplitude of 5.08e-3 radians.

Figure 5.23 Harmonic response for steel shaft

Figure 5.24 Harmonic response for E-Glass/Epoxy shaft
The harmonic analysis results for E-glass/epoxy shaft show that the maximum deflection at resonance conditions occur in the range of 0 to 2000 Hz frequencies and is maximum at 600 Hz (36000rpm) with amplitude of 0.05539 radians.

![Response of HM-Carbon/Epoxy shaft](image)

**Figure 5.25 Harmonic response for HM Carbon/Epoxy shaft**

The harmonic analysis results for HM carbon/Epoxy shaft show that the maximum deflection at resonance conditions occur in the range of 0 to 2000 Hz frequencies and is maximum at 1000 Hz (60000rpm) with amplitude of 0.238058E-01 radians.
The harmonic analysis results for Boron/Epoxy shaft show that the maximum deflection at resonance conditions occur in the range of 0 to 2000 Hz frequencies and is maximum at 800 Hz (48000rpm) with amplitude of 2.49e-2 radians.
The harmonic analysis results for E-Glass/vinylester shaft show that the maximum deflection at resonance conditions occur in the range of 0 to 2000 Hz frequencies and is maximum at 1600 Hz (96000 rpm) with amplitude of 0.06219 radians.

**Table 5.5 Comparison of response (angular) at resonant Frequencies**

<table>
<thead>
<tr>
<th></th>
<th>Steel</th>
<th>E-Glass/ Epoxy</th>
<th>E-Glass/ Vinylester</th>
<th>HM Carbon/ Epoxy</th>
<th>Boron/ Epoxy</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frequency (Hz)</td>
<td>600</td>
<td>600</td>
<td>1600</td>
<td>1000</td>
<td>800</td>
</tr>
<tr>
<td>Critical speed (rpm)</td>
<td>36000</td>
<td>36000</td>
<td>96000</td>
<td>60000</td>
<td>48000</td>
</tr>
<tr>
<td>Degrees</td>
<td>0.29</td>
<td>3.17</td>
<td>3.55</td>
<td>1.36</td>
<td>1.42</td>
</tr>
</tbody>
</table>

Table 5.5 shows the angular response of the shafts for various materials at their corresponding resonance frequencies. It can be seen that Boron//Epoxy is having more amplitude of response when compared to all other materials. The Steel and E-Glass/Vinylester are having less amplitude of response compared to other materials.

**5.5 ANALYSIS OF ADHESIVELY BONDED TUBULAR JOINTS**

**5.5.1 Introduction**

In automotive applications it becomes necessary to provide suitable metallic end joints for the composite shafts so that they can be connected to the other parts of the power train. There are different methods by which a
composite shaft can be connected to a metallic end piece such as bolted joints, splines, riveted joints and bonded joints. Bonded joints are preferred over other mechanical fastening methods, since they can transfer load more evenly over a large area and they do not damage the composite structure and create stress concentrations. In this chapter bonded joints are analyzed for various geometries and configurations using FEA. The preliminary tubular joint design discussed in Chapter-3, which is useful in getting some initial estimates of joint design parameters but it does not completely capture all the joint details such as fillet effects, tapered adherends as well as complex stress state at the joint edges. Thus, once the preliminary estimates for a joint design have been obtained, it is prudent to perform a detailed finite element analysis of the joint.

5.5.2 Finite element model

Finite element analysis was performed on the bonded end joint models with aluminum and composite adherends to analyze the shear stress variation in the adhesive layer with change in bonding length and adhesive thickness. A finite element model using ANSYS 5.4 of a typical aluminum yoke-to-composite shaft tubular lap joint is presented here. Figures 5.28 and 5.29 show FE model of the tubular joint and representation of single lap joint model respectively. The material properties are the same as considered in Chapter -3. The optimal stacking sequence and thickness of the different composites shafts were considered in Chapter- 4. The shear stress analysis on the FEA model was performed by varying adhesive thickness and bonding length. Adhesive thicknesses of 0.2mm, 0.25mm, 0.3mm, 0.35mm and 0.4mm were analyzed. Lap lengths of 20mm, 25mm, 30mm, 35mm and 40 mm were tested. In all cases the shear stress in the adhesive layer was plotted to find the maximum shear stress.
Figure 5.28 Finite element model of the tubular lap joint

Figure 5.29 Representation of single lap joint model
5.5.3 Boundary conditions

The boundary conditions applied were similar to the real situation. That is the joint was subjected to axial torsion. The yoke end was fixed with all degrees of freedom arrested at the end nodes. The shaft end nodes were rotated to cylindrical coordinate system and torque is applied to them as tangential forces in Y direction. A fixed torque of 3500 Nm is applied for both single and double lap joints.

5.5.4 Shear stress Analysis by Varying Adhesive Thickness and Bonding Length

The finite element analysis of the torque transmission capabilities is performed by ANSYS 5.4. The element used for the stress analysis is three-dimensional solid layered 46 elements. Since the stress state of the composite drive shaft tube is axi-symmetric, only one-fourth part of the composite drive shaft with metal joint was modeled. Here, all degrees of freedom (DOF) are constrained at one end of the shaft and the active co-ordinate system was changed into global cylindrical system. Another end of the shaft, the nodes have to be rotated and the force is applied at the Y-direction to apply static torsion.
Figure 5.30  Shear stress in XY-Plane in E-Glass/Epoxy shaft and adhesive layer with lap length 20mm and adhesive thickness 0.2mm

Figure 5.31  Shear stress in XZ-Plane in E-Glass/Epoxy shaft and adhesive layer with lap length 20mm and adhesive thickness 0.2mm
Figure 5.32  Shear stress in YZ-Plane in E-Glass/Epoxy shaft and adhesive layer with lap length 20mm and adhesive thickness 0.2mm

Figure 5.33 Finite element model of the double lap joint
Figure 5.34  Shear stress in XY-Plane in E-Glass/Epoxy shaft and adhesive layer with double lap length 20mm and adhesive thickness 0.2mm

Figure 5.35  Shear stress in YZ-Plane in E-Glass/Epoxy shaft and adhesive layer with double lap length 20mm and adhesive thickness 0.2mm

Figure 5.36  Shear stress in XZ-Plane in E-Glass/Epoxy shaft and adhesive layer with double lap length 20mm and adhesive thickness 0.2mm
Shear Stress in E-Glass/Epoxy Lap Joints by Varying the Adhesive Thickness

Figure 5.37 Shear stress in E-Glass/Epoxy single lap joint by varying the adhesive thickness

Figure 5.38 Shear stress in E-Glass/Epoxy Double lap joint by varying the adhesive thickness
For E-Glass/Epoxy in the maximum shear stress is reduced drastically by using double lap joint. The minimum stress is obtained with 0.4mm adhesive thickness and it is at 0.35mm for the double lap joint.

![Figure 5.39 Shear stress in E-Glass/Epoxy single lap joint by varying the lap length](image)

![Figure 5.39 Shear stress in E-Glass/Epoxy double lap joint by varying the lap length](image)
Figure 5.40  Shear stress in E-Glass/Epoxy double lap joint by varying the lap length

In single lap joint, the minimum shear stress is found to be at 30mm lap length and it is found to be the same for double lap joint.

Figure 5.41  Shear stress in E-Glass/Vinylester single lap joint by varying the adhesive layer thickness
Figure 5.42 Shear stress in E-Glass/Vinylester double lap joint by varying the adhesive layer thickness

In E-Glass/Vinyl Ester shaft the shear stress is reduced by the use of Double lap joint. The maximum shear stresses for single lap and double lap joints are obtained at adhesive thicknesses 0.25mm and 0.3mm respectively.

Figure 5.43 Shear stress in E-Glass/Vinylester single lap joint by varying the bonding length
Figure 5.44  Shear Stress in E-Glass/Vinylester double lap joint by varying the bonding length

The Double lap joint in the E-Glass/Vinylester shaft does not show much stress variation with change in lap length. The minimum stress occurs at 20mm lap length. The single lap joint gives the minimum stress at 20mm but it increases drastically with increase in lap length.

Figure 5.45  Shear stress in Boron/Epoxy single lap joint by varying the adhesive layer thickness

Figure 5.46  Shear stress in Boron/Epoxy double lap joint by varying the adhesive layer thickness
Figure 5.46  Shear stress in Boron/Epoxy double lap joint by varying the adhesive layer thickness

With Boron /Epoxy shaft the shear stress levels are reduced for the same adhesive thickness with the use of Double lap joint. The minimum shear stresses are obtained at 0.2mm and 0.4mm adhesive thicknesses respectively.

Figure 5.47  Shear stress in Boron/Epoxy single lap joint by varying the bonding length
Figure 5.48  Shear stress in Boron/Epoxy double lap joint by varying the bonding length

In Boron/Epoxy shaft the double as well as single lap joints do not show any drastic variations with lap length. The double lap joint has shown lesser shear stress development in the adhesive layer to almost half and less.

Figure 5.49  Shear stress in HM-Carbon/Epoxy single lap joint by varying the adhesive layer thickness
Figure 5.50 Shear stress in HM-Carbon/Epoxy double lap joint by varying the adhesive layer thickness

With HM-Carbon/Epoxy shaft, the shear stress is reduced drastically by double lap joint. The minimum shear stress occurs at 0.4mm adhesive thickness in both the cases.

Figure 5.51 Shear stress in HM-Carbon/Epoxy single lap joint by varying the bonding length
Figure 5.52 Shear stress in HM-Carbon/Epoxy double lap joint by varying the bonding length

From the analysis results, important observations are made for both single as well as double lap joints. The shear stress for all composite material shafts in XY plane is found to be considerably less than those in the other two planes. This is because the shaft was subjected to pure torsion and the shear force due to the torque is shared by the XZ and YZ components of the shear stress. This shear stress component seems to be not much affected by changes in the adhesive thickness and the bonding length. The shear stress in the adhesive layer decreases with increase in adhesive layer thickness. So maximum thickness is to be given for the adhesive layer. However it is assumed in the analyses that the shear stress distribution in the adhesive layer along the thickness is uniform. If the thickness is increased beyond a limit there will be non-linearity in the stress distribution and may cause cracking and failure of adhesive layer. In this research work, the maximum thickness is limited to 0.4mm. Increasing the lap length aids in reducing the shear stress in adhesive
layer to some extend after which it has no effect or may slightly increase the stress. The stiffness of the adherends and their thicknesses also affect the stress levels in the adhesive layer. To keep the shear stresses to be low, the thickness and stiffness of the adherends should be maintained minimum. Also, the difference in stiffness between adherends should be kept minimum. So with less stiff composites like E-Glass/Vinylester and E-Glass/Epoxy, a less stiff adherend like Aluminum should be used for end joints and with HM Carbon/Epoxy and Boron/Epoxy, steel may be used for end fixtures.

For the same adhesive thickness and lap length, double lap joint gives lesser shear stress in adhesive layer. In the analysis, there are some exceptions to this because the thickness of the metal adherend is doubled increasing the stiffness of the total metal structure. With Boron/epoxy in the double lap joint the shear stress is reduced considerably, since the difference in stiffness of the shaft and that of the metal end joint is less. The double lap joint also prevents the delamination of the composite shaft at end joints because the ends of the shaft are enclosed in the joint.

### 5.5.5 Recommendations for lap joints

Based on preliminary tubular joint design discussed in chapter-3 and FEA results, the recommendations for various composites and types of joints are made as in Tables 5.6 and 5.7 for single lap and double lap joints respectively.

#### Table 5.6 Recommendations for single lap joints

<table>
<thead>
<tr>
<th>Material</th>
<th>Adhesive thickness (mm)</th>
<th>Lap length (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>E-Glass/Epoxy</td>
<td>0.35</td>
<td>30</td>
</tr>
</tbody>
</table>
Table 5.7 Recommendations for double lap joints

<table>
<thead>
<tr>
<th>Material</th>
<th>Adhesive thickness</th>
<th>Lap length (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>E-Glass/Epoxy</td>
<td>0.35</td>
<td>30</td>
</tr>
<tr>
<td>E-Glass/Vinylester</td>
<td>0.35</td>
<td>30</td>
</tr>
<tr>
<td>HM Carbon/Epoxy</td>
<td>0.25</td>
<td>30</td>
</tr>
<tr>
<td>Boron/Epoxy</td>
<td>0.4</td>
<td>30</td>
</tr>
</tbody>
</table>

5.6 CONCLUSIONS

1. The variations of stresses across the thickness of steel and composite drive shaft tubes are predicted.

2. The finite element analysis of steel and composite drive shafts is done for torsional buckling strength and the natural frequencies of steel and composite drive shafts are determined and compared.

3. Stress distributions through the thickness of the shaft due to bending are found to be within allowable limit.

4. The effects of specimen geometries and material properties, on the shear stress distribution in the adhesive layer are investigated.
5. A tubular lap joint consisting of fiber reinforced composite shaft and an aluminum yoke, with epoxy adhesive are analyzed with FEA.