The influence of number and orientation of ply on tensile properties of hybrid composites

Anik Das Anto, Sobahan Mia and Md Abdul Hasib
Department of Mechanical Engineering, Khulna University of Engineering & Technology, Khulna-9203, Bangladesh
E-mail: antoanikdas@gmail.com
Keywords: hybrid composite, finite element (FE), orientation of ply

Abstract
The main concern of this work is to investigate the influence of number and orientation of ply on tensile properties of hybrid composites. Hybrid composites are those materials which consist of two or more distinct fibers embedded in the same matrix. For this analysis, the elastic behavior of individual fiber-reinforced composite lamina is required. The properties of a single lamina of composites can be determined by the micromechanical homogenization. From this analysis, it is seen that elastic modulus, Poisson’s ratio, and shear modulus are the functions of the volume fraction of the fiber and matrix. These properties can be further used for analysis of hybrid composite laminate. In this work different hybrid composites material with a different number of plies and orientations are used to simulate with finite element software. At first, a 3D model of three-ply is used for the simulation. The other model with four, five and six-ply has been used. The influence of number and orientation of ply on tensile properties of hybrid composites is the main goal of the work. The comparison of stiffness between different ply models has established that the number ply influences the strength of hybrid composites. From this work, it is seen that as number ply increases the tensile strength also increases. For the investigation of the influence of the ply orientation, different orientations of ply are used. From different simulation results, it is seen that ply orientation with 0° is the best for the tensile strength.

1. General introduction

In the mission for enhanced execution, material researchers, engineers, and scientists are always determined to produce either improved traditional materials or completely novel materials. Composites are one of the enhanced classes of materials. Composite materials are the most dominant engineering substances for their outstanding properties, i.e., high rate of stiffness, low weight, strength, high chemical resistance, machinability, and many other properties. Composites are greatly used in numerous fields of engineering which includes high-end applications as aircraft, wind turbines, and so on. Those components are always cyclically loaded resulting in unexpected failure, which may cause heavy loss. Composites could prevent this unexpected failure [1]. The composites industry has started to perceive that the industrial applications of composites can provide considerably larger business opportunities than the aerospace sector because of the sheer size of the transportation industry. Composite materials have been used in recent years to meet the challenges of the aerospace industry. Composites have also poured down to catering to domestic and commercial applications [2]. In recent years the cost of energy has been on the rise and has pressured engineers to increase the energy efficiency of their products. This evolution is taking place worldwide and can be found in every engineering discipline. A reduction of physical testing helps keep design costs down and can only be accomplished with an increased understanding of computational analysis methods [3]. In today’s automotive and aircraft design industry the majority of computational analyses are completed using commercially available finite element packages. Finite element analysis (FEA or FEM) can be used to predict many different types of mechanical behaviors such as stress, strain, displacement, natural frequencies, mode shapes, and various types of failure.
This capability is what allows engineers to design, test, and optimize components prior to the manufacture of a single part. In this thesis work, the main focus is given to designing a composite material based on tensile strength [4, 5].

2. Composite

A composite material consists of two or more materials which are chemically distinct from each another. These totally distinct individual materials are called constituent materials. There are two general categories of constituent materials. One of which is known as matrix and other is known as reinforcement. One or more discontinuous phases are embedded in a continuous phase for building composites. The discontinuous phase of the composite is typically harder and stronger. The discontinuous phase is known as the reinforcement or reinforcing the material. The continuous phase is softer and known as the matrix.

3. Commercial FEA software packages

High-performance composites are created from the layer of the fiber sheet. These sheets have adhered together. The layer from which composite is constructed is known as a ply or lamina. Built-in features are available in various finite element software packages which can be used to model composite laminates. FEA software packages can be divided into two categories. Some packages have built-in pre-processors and post-processors, while some others have separate pre- and post-processors. ABAQUS and ANSYS are two of the most popular packages currently being used in the industry. The thesis work has been done in ABAQUS.

4. ABAQUS software package

ABAQUS has a built-in composite modeling capability. ABAQUS has a variety of options for the analysis of composites. ABAQUS has a ply layup editor, material models, and built-in damage models. ABAQUS can model orthotropic and anisotropic material. A GUI of the ply layup editor is used to set up different ply. ABAQUS provides different element types for meshing a composite laminate. When a laminate geometry is modeled, it can be used for continuum shell elements or 3D solid stress elements. Conventional shell elements can also be used when its geometry is defined by a planar surface. Conventional continuum shell elements are advantageous as less computation time is needed. Continuum shell elements need more computation time [5].

5. Modeling scale

Results of composite material also depend on the modeling scale of the model. It is important to select an appropriate scale for the desired results. For composites, two scales can be considered such as microscale and macroscale (lamina scale and laminate scale). When a micro level of result is needed, micromechanical analysis is used. For this type, the strain and stress are determined at the constituent level. In this type of analysis, modeling is done at a micromechanical level. If the elastic properties of each lamina are known, macro-mechanical analysis can be done. The macro-mechanical analysis can be classified as two types such as lamina scale and laminate scale. The lamina is the layer of the composite material. Laminate is formed with more than one lamina. The laminate level analysis is done assuming the composite is homogeneous.

6. Hybrid composite modelling using FEA step

To model the tests in the FEA software, ABAQUS, the following steps have been taken.

6.1. Creating the part

The ABAQUS FEA package provides built-in composite modeling capabilities. The geometry of the test specimen is a length of 300 mm, with a width of 25 mm, and for gripping, 57 mm each on each end as shown in figure 1. A 3D deformable shell planar part is created. The part is drawn as a 2D shape and the thickness of the part is indicated, applying a conventional shell section.
6.2. Creating the material
To model, the composite material at first material properties is created. By defining the mechanical properties as elastic and setting the type to engineering constant, the individual ply material properties can be given as input. The Input window is shown in figure 2. The properties of the material are given in table 1.

![Figure 2. Input GUI to create a material.](image)

### Table 1. Material properties of E-glass and carbon fiber epoxy lamina.

| Property | Unit | E-glass epoxy lamina [6] | Carbon fiber lamina [6] |
|----------|------|--------------------------|------------------------|
| $V_f$    |      | 0.60                     | 0.60                   |
| $E_1$    | GPa  | 45.6                     | 126                    |
| $E_2 = E_3$ | GPa  | 16.2                     | 11                     |
| $G_{12} = G_{13}$ | GPa  | 5.83                     | 6.6                    |
| $G_{23}$ | GPa  | 5.79                     | 3.93                   |
| $\nu_{12} = \nu_{13}$ |      | 0.278                    | 0.28                   |
| $\nu_{23}$ |      | 0.4                      | 0.4                    |
| $F_{11}$ | MPa  | 1200                     | 1950                   |
| $F_{16}$ | MPa  | 800                      | 1480                   |
| $F_{21} = F_{31}$ | MPa  | 40                       | 48                     |
| $F_{22} = F_{32}$ | MPa  | 145                      | 200                    |
| $F_6$    | MPa  | 73                       | 79                     |

6.3. Creating composite layup
A GUI is used to define layered composite properties in the ABAQUS. This GUI is known as the composite layup editor. A table is used to define the plies in the layup. This table can be used to assign a name, material, thickness, and orientation to each ply. Figure 3 shows the composite ply layup assignments. The composite layup editor is used to create plies and to assign materials and orientations to these plies. In this step, a composite layup that
represents the specimen layup (figure 4) is created and defined. Each layer of the composite are specified as part of the composite layup section definition.

6.4. Instancing the part
Before creating any boundary conditions or meshes the part is instanced, to be a dependent (mesh) on part instance.

6.5. Boundary condition, load applied on the specimen
The boundary condition (BC) and load must be defined and will be activated during the simulation. In this simulation, the BC will be applied at one end of the specimen. The type of BC applied is Symmetry/Antisymmetry/Encastre in Mechanical category and toggle on ENCASTRE (U1 = U2 = U3 = UR1 = UR2 = UR3 = 0). The load is applied at the other end of the specimen with type traction load on the surface. Figure 5 shows the BC and load applied on the specimen.

6.6. Meshing the part
The S4R Quad shell mesh elements are used for structured meshing. Generally, the more refined the mesh the more accurate the result, but with this simplistic rectangular shape. But it has been found that the factor of refinement and hence the number of elements in the mesh does not affect output, and a largely refined mesh is not needed in this case. The meshed geometry is shown in figure 6.

7. Effect of ply number
The strength of the hybrid composite depends on its ply number. In this work a different model with a different number of ply is used to find out the effect of ply number. It has been established that when the number of ply increases the value of stiffness also increases.

It is clear that the stiffness of the hybrid composite increases with the ply number (figure 7). As the ply number increases, the stiffness also increases (figure 8). So the strength of the composite material is a function of its ply number. So it is established that when the ply number is increased the tensile strength is also increased.
8. Effect of ply orientation

The various combination of ply orientations were simulated in this work. The various model results are described below:

A model with six different ply is used in this work. The 6 ply model shows more tensile strength which is found from the previous simulation. The deformed and undeformed shape for 0° ply model of 6 ply model is given in figure 9.

For all specimens the increase of displacement is linearly constant with the applied load. The stiffness value is very high for the 0° ply orientation. From figures 10 and 11, it is seen that the 0° ply orientation is more stable in the tensile loading.

9. Comparative performance study

The comparative study between normal composite and hybrid composite have been done in the experiment (figure 12). The results shows that the hybrid composite is more suitable for using as the stiffness is higher than the normal composite.
10. Failure analysis

Failure criteria are used to predict failure under multiaxial stress or under uniaxial stress. All failure criteria can detect or predict the first occurrence of failure in any lamina. But failure criteria are unable to track failure propagation [7]. Failure criteria can be presented using a ratio of stress and strength of any point known as failure index, which is used for several FEA packages, and it is defined as

$$ I_F = \frac{\text{Stress}}{\text{Strength}} $$

Failure is predicted when $I_F \geq 1$. When the value of failure index is more than one the first failure is predicted. The inverse of the failure index is also used for some cases to predict the failure. The inverse of the failure index is known as strength ratio.
The strength ratio is similar to a safety factor. Failure is predicted when $R \leq 1$.

10.1. Failure theories in composite materials

(i) Maximum Stress Theory

(ii) Tsai–Wu Theory

10.1.1. MSTRS (maximum stress criterion)

The maximum failure index factor (table 2) is less than one for all ply which ensures that there is no failure for maximum stress. If one of these limits exceeds zero the failure must be initiated in respective ply. From the contour it is established that for zero degree ply orientation there is no failure.

10.1.2. TSAIW (Tsai–Wu)

The maximum failure index factor (table 3) is less than one for all ply which ensures that there is no failure of the 0° orientation six ply model.
Table 2. Maximum Stress Index of six ply model for 0°, 30°, 45° and 90° orientation.

| PLY NO. | 0° Orientation | 30° Orientation | 45° Orientation | 90° Orientation |
|---------|----------------|-----------------|-----------------|-----------------|
| 1       | 0.459          | 4.570           | 8.043           | 10.116          |
| 2       | 0.651          | 8.213           | 14.49           | 18.200          |
| 3       | 0.460          | 4.550           | 8.017           | 10.052          |
| 4       | 0.647          | 8.210           | 14.443          | 18.084          |
| 5       | 0.461          | 4.552           | 7.991           | 9.987           |
| 6       | 0.463          | 8.215           | 14.369          | 17.968          |

Table 3. Tsai–Wu failure Index of six ply model for 0°, 30°, 45° and 90° orientation.

| PLY NO. | 0° Orientation | 30° Orientation | 45° Orientation | 90° Orientation |
|---------|----------------|-----------------|-----------------|-----------------|
| 1       | 0.574          | 6.043           | 9.024           | 10.120          |
| 2       | 0.654          | 9.395           | 15.241          | 18.151          |
| 3       | 0.565          | 6.066           | 9.0530          | 10.061          |
| 4       | 0.650          | 9.380           | 15.239          | 18.042          |
| 5       | 0.562          | 6.091           | 9.0830          | 10.002          |
| 6       | 0.646          | 9.365           | 15.238          | 17.933          |

11. Conclusion

The elastic properties obtained from technique based on micromechanical RVE can be used for other macro-mechanical analysis of any composite or hybrid composite laminates [8]. Using the lamina properties of composites, further macro-mechanical analysis of hybrid composite has been done. The tensile load displacement response of hybrid composites having different layups are presented. From numerical analysis on hybrid composite specimens with different ply, it is seen that the strength of the hybrid composite depends on the ply number. The more ply the more strength. There is another remarkable consequence that ply orientation is important when designing a hybrid composite. The zero degree ply has better strength than any other orientation. The failure analysis of the hybrid composite shows that when the angle of orientation is more than zero degree, there is a failure which is not a good sign for modeling a composite. Hence it is established that only zero degree orientation of ply with more ply as possible would be the convenient design.

ORCID iDs

Anik Das Anto  © https://orcid.org/0000-0002-0815-4747

Reference

[1] Marcal P V and Yamagata N (ed) 2016 Design and Analysis of Reinforced Fiber Composites (Berlin: Springer)
[2] Erden S and Ho K 2017 Fiber reinforced composites Fiber Technology for Fiber-Reinforced Composites (Amsterdam: Elsevier) pp 51–79
[3] Lee S, Park T and Voyiadjis G Z 2003 Vibration Analysis of multi-delaminated beams Composites Part B: Engineering 34 647–59
[4] Barbero E J 2017 Introduction to Composite Materials Design 3rd edn (Boca Raton, FL: CRC Press)
[5] Willis B M 2013 Applying finite element analysis with a focus on tensile damage modeling of carbon fiber reinforced polymer laminates MSc Thesis Ohio State University
[6] Huang Z M 2001 Simulation of the mechanical properties of fibrous composites by the bridging micromechanics model Composites Part A: Applied Science and Manufacturing 32 143–72
[7] Barbero E J 2007 Finite Element Analysis of Composite Materials (Boca Raton, FL: CRC Press Taylor & Francis Group)
[8] Anto A D, Mia S, Hasib M A, Hasan M and Chowdhury T A 2018 Investigation of elastic properties of fiber-reinforced composites by micromechanical modeling method 4th Int. Conf. on Structure, Proc., and Properties of Materials (SPPM) (Dhaka, Bangladesh: BUET)