International Journal of Applied Engineering Research ISSN 0973-4562 Volume 13, Number 2 (2018)

The Mechanisms and Mechanics of the Toughening of Glass Fiber Reinforced Composites

M.ASHOK KUMAR[1]*, SYED ALTAF HUSSAIN[1], B.SIDDA REDDY[1]

[1]Department of Mechanical Engineering, Rajeev Gandhi Memorial College of Engineering & Technology, Nandyal-518501, Kurnool-Dist., Andhra Pradesh, India

P.HARI SANKAR[2]

[2]Department of Mechanical Engineering, G.Pulla Reddy Engineering College, Kurnool-518003,Andhra Pradesh, India

Abstract

The mechanical behaviour of fiber reinforced composite depends upon fiber orientation, fiber volume fraction, fiber spacing and fiber distribution in the matrix. This article deals with the application of new constitutive fracture model for an polymer fiber reinforced composites. The different representative volume elements (RVEs) are developed with a random distribution of fibers. The focus is made to the influence of the interface between the fiber and matrix, as well as to the influence of the polyester matrix, on the strength properties of the composite, damage initiation and propagation under different loading conditions. In the proposed work the micro-mechanical behaviour of uni-directional fiber reinforced composite with glass fiber embedded on vinyl-ester matrix has to be analyzed for various volume fractions (viz., 10% to 70%) and the various engineering constants like, Longitudinal modulus (E_1), Transverse modulus (E_2), Shear modulus (G_{12}) and Major poison's ratio (ν_{12}) are evaluated using Finite element Analysis software package Ansys 17.0. The output obtained is eventually compared to the results obtained by mechanics of solids approach results that are theoretically calculated.

Keywords: micromechanical behaviour, volume fractions, finite element analysis, fiber reinforced composite

NOMENCLATURE

V_f	Fiber volume fraction
r	Radius of the fiber
a	Area of the fiber
E	Young's modulus, MPa
E_1	Longitudinal young's modulus, MPa
E_2	Transverse young's modulus, MPa
ν	Poisson's ratio

G_{12} In-plane shear modulus, MPa

ν_{12} Major Poison's ratio

$\varepsilon_{c,f,m}$ Strains in composite, fiber, and matrix respectively

τ Shear stress, MPa

$\sigma_{c,f,m}$ Stresses in composite, fiber, and matrix resp.., MPa

Introduction

The properties of a composite material are strongly influenced by the properties of its constituents and their distribution and also the quality of interactions among them the most important of all the composites properties are usually the mechanical properties, since whatever the reasons for a choice of a particular composite for some application it must have certain characteristics of shape, rigidity and strength. The mechanical properties of long fibers composites are predicted by "Rules of mixture" and the Halpin-Tsai principles. Continuous fibers offer the highest mechanical properties and give the possibility of using specific orientations to give the composite directional properties. These includes unidirectional, biaxial, multi-axial, and random [1-3].

The stress-strain relationships, engineering constants and failure theories for an angle lamina were developed using four elastic moduli, the four elastic moduli's are Longitudinal young's modulus (E1), Transverse young's modulus (E2), Major poison's ratio (ν_{12}), in plane shear modulus (G_{12}).

sMechanics of Solids Approach

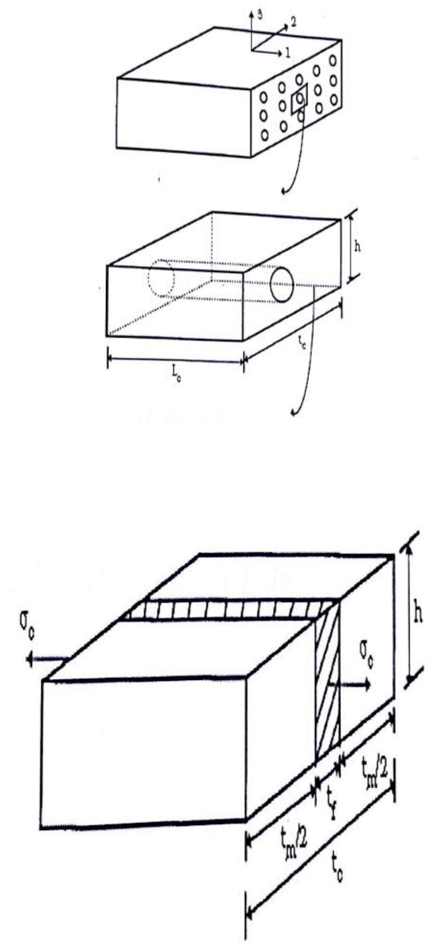


Figure 1 A longitudinal stress applied to the representative volume element to calculate the longitudinal Young's modulus for a unidirectional lamina.

There are many approaches for determining the four elastic moduli, among them Mechanics of solids approach and semi-empirical models are more frequently used. From a unidirectional lamina, take a representative volume element (a representative volume element of a material is the smallest part of the material that represents the material as a whole. It could be otherwise intractable to account for the distribution of the constituents of the material) that consists of a fibre surrounded by the matrix. This representative volume element (RVE) can be further represented as rectangular blocks. The fibre, matrix and the composite are assumed to be of the same width h, and thickness t_f, t_m, and t_c respectively

The area of the fibre is given by

$$A_f = t_f h \qquad \text{….i}$$

The area of the matix is given by

$$A_m = t_m h \qquad \text{….ii}$$

And the area of the composite is given by $A_c = t_c h$ ….iii

The two areas are chosen in the proportion of their volume fraction so

That the fibre volume is defined as

$$V_f = A_f / A_c \qquad \text{….iv}$$

$$= t_f / t_c \qquad \text{….v}$$

And the matrix fibre volume fraction V_m is

$$V_m = A_m / A_c \qquad \text{….vi}$$

$$= t_m / t_c \qquad \text{….vii}$$

$$= 1 - V_f \qquad \text{….viii}$$

The following assumptions are made in the strength of approach materials model. The bond between fibers and matrix is perfect, the elastic moduli, diameters, and space between fibers are uniform the fibers are Continuous and parallel, The fibers and matrix follows Hooke's law, The fibers possess uniform strength, The composite is free of voids.

Longitudinal Young's Modulus

From figure, under a uni-axial load F_c on the composite RVE, the load is shared by the fibre F_f and the matrix F_m so that

$$F_c = F_f + F_m$$

The loads taken by the fibre, the matrix and the composite can be written in terms of the stress in these components and cross sectional areas of these components

$$F_c = \sigma_c A$$

$$F_f = \sigma_f A_f,$$

And

$$F_m = \sigma_m A_m \qquad \text{….ix}$$

Where

$\sigma_{c,f,m}$ =stress in composite, fibre and matrix respectively

$A_{c,f,m}$= area of composite, fibre, and matrix respectively

Assuming that the fibers, matrix and composites follow Hooke's law and that the fibers and the matrix are isotropic , and stress-strain relationship for each component and the composite is

$$\sigma_c = E_c \varepsilon_c \qquad \text{….x}$$

$$\sigma_f = E_f \varepsilon_f, \qquad \text{….xi}$$

And

$$\sigma_m = E_m \varepsilon_m \qquad \text{….xii}$$

Where $\varepsilon_{c,\ f.\ m}$ = strains in composite in , fibre and matrix respectively

$E_{c,f,m}$ = elastic moduli of composite, fibre and matrix respectively

Substitution the equations (x), (xi), and (xii) yields

$E_c \varepsilon_c A_c = E_f \quad \varepsilon_f \quad A_f + E_m \quad \varepsilon_m \quad A_m$ ….xiii

The strains in the composite, fibre, and matrix are equal ($\varepsilon_c=\varepsilon_f=\varepsilon_m$);

From the equation (xii),

$$E_1=E_f \frac{Af}{Ac} + Em \frac{Am}{Ac} \quad \text{....xiv}$$

using equations (iv) to (viii) , for definitions of volume fractions,

$$E_1=E_fV_f+E_mV_m \quad \text{.....xv}$$

Equation xv gives the longitudinal young's modulus as a weighted mean of the fibre and matrix modulus. It is also called the " *rule of mixture"*.

Transverse Young's Modulus

Assume now that, as shown in figure 2 the composite is stressed in the transverse direction. The fibers and matrix are again represented b rectangular blocks shown in figure. The fibre, the matrix, and composite stresses are equal. Thus,

$$\sigma_c = \sigma_f = \sigma_m, \quad \text{..... xvi}$$

where,

$\sigma_{c,f,m}$ = stress in composite, fibre and matrix respectively.

Now, the transverse extensions in the composite Δc is the sum of the transverse extensions in the fibre Δf, and that is matrix' Δm.

$$\Delta_c=\Delta_f+\Delta_m, \quad \text{....xvii}$$

Now, by the definition of normal strain,

$$\Delta_c=t_c\varepsilon_c, \quad \text{... (xvii a)}$$

$$\Delta_f = t_f\, \varepsilon_f, \quad \text{... (xvii b)}$$

And

$$\Delta_m=t_m\varepsilon_m, \quad \text{.... (xviic)}$$

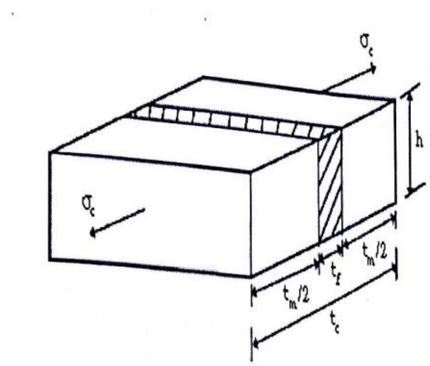


Figure 2: A transverse stress applied to the representative volume element to calculate the transverse Young's modulus for a unidirectional lamina.

Where $t_{c,f,m}$ = thickness of the composite , fibre, and matrix respectively

$\varepsilon_{c,f,m}$ = normal transverse strain in the composite, fibre, and matrix respectively.

Also, by using Hooke's law for the fibre, matrix, and composite the normal strains in the composite, fibre, and matrix are

$\varepsilon_c = \frac{\sigma c}{E2}$ ....xviii(a)

$\varepsilon_f = \frac{\sigma f}{Ef}$ ..... xviii(b)

and

$\varepsilon_m = \frac{\sigma m}{Em}$,

.... xviii (c)

substituting equation (xvi) and equation (xvii) in equation (xviii)

and using equation (xv) gives

$\frac{1}{E2} = \frac{1}{Ef}\frac{tf}{tc} + \frac{1}{Em}\frac{tm}{tc}$ ixxx

Because the thickness fractions are the same as the volume fractions as the other two dimensions are equal for the fibre and the matrix the matrix (Ref. Fig.2).

$\frac{1}{E2} = \frac{Vf}{Ef} + \frac{Vm}{Em}$ xx

The above gives the transverse young's modulus as a weighted mean of the fiber and matrix modulus. It is also called the "*rule of mixture*".

Major Poisson's Ratio

The major poison's ratio is defined as the negative of the ratio of the normal strain in the transverse direction to the normal strain in the longitudinal direction, when a normal load is applied in the longitudinal direction. Assume a composite is loaded in the direction parallel to the fibers, as shown in figure1. The fibers and matrix are again represented by rectangular blocks. The deformation in the transverse direction of the composite (Δ^T_c) is the sum of the transverse deformations of the fiber (Δ^T_f) and the matrix (Δ^T_m) as

$\delta^T_c = \delta^T_f + \delta^T_m$ ..... (xxi)

Using the definition of normal strains,

$\varepsilon^T_f = \delta^T_f / t_f$, ... xxi(a)

$\varepsilon^T_m = \delta^T_m / t_m$ .... xxi(b)

And $\varepsilon^T_c = \delta^T_c / t_c$ .... xxi(c)

Where

$\varepsilon_{c,f,m}$ are transverse strains in composite, fiber and matrix respectively.

$t_c \varepsilon^T_c = t_f \varepsilon^T_f + t_m \varepsilon^T_m$

The poisson's ratios for the fiber , matrix and composite, respectively

Are

$V_f = -\varepsilon^T_f / \varepsilon^L_f$ .... xxii(a)

$\nu_m = -\varepsilon^T_m / \varepsilon^L_m$ .... xxiii(b)

And

$\nu_{12} = -\varepsilon^T_c / \varepsilon^L_c$ .... (xxiii b)

$-t_c \nu_{12} \varepsilon^L_c = -t_f \nu_{12} \varepsilon^L_f + -t_m \nu_{12} \varepsilon^L_m$ ...... (xxiv)

Where

$\nu_{12,f,m}$ =Poison's ratio of composite, fiber, and matrix respectively

$\varepsilon^l_{c,f,m}$= longitudinal strains of composite, fiber and matrix respectively.

However, the strains in the composite, fiber, and matrix are assumed to be equal to the longitudinal direction $\varepsilon^L_c = \varepsilon^L_m = \varepsilon^L_f$, which, from equation gives

$$t_c \nu_{12} = t_f \nu_{12} + t_m \nu_{12}$$

$$\nu_{12} = \frac{tf}{tc}\nu_{12} + \frac{tm}{tc}\nu_{12} \quad \text{... (xxv)}$$

Because the thickness fractions are the same as the volume fractions as per equations,

$$V_{12} = V_f \nu_f + V_m \nu_m \quad \text{.... (xxvi)}$$

In- Plane Shear Modulus

Apply a pure shear stress τ_C to a lamina as shown in figure. The fibers and matrix are represented by rectangular block as shown (Ref. Fig.3)

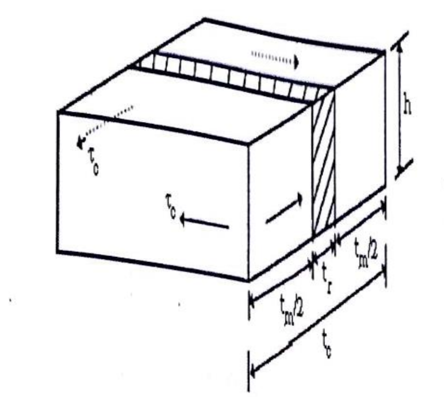


Figure 3: An in-plane shear stress applied to a representative volume element for finding in-plane shear modulus of a unidirectional lamina.

The resulting shear deformations of the composite δc the fiber δf, and the matrix δm are related by

$$\delta_c = \delta_f + \delta_m \quad \text{.... xxvii}$$

From the definition of shear strains,

$$\delta_c = \gamma_c\, t_c \quad \text{... xxvii(a)}$$

$$\delta_f = \gamma_f\, t \quad \text{.... xxvii(b)}$$

And

$$\delta_m = \gamma_m\, t_m \quad \text{.... xxvii(c)}$$

Where

$\gamma_{c,f,m}$ = shearing strains in the composite, fibre, and matrix respectively

$t_{c,f,m}$ = thickness of the composite, fibre, and matrix respectively

From Hooke's law for the fibre, matrix and the composite,

$$\gamma_c = \frac{\tau c}{G12} \quad \text{... xxviii(a)}$$

$$\gamma_f = \frac{\tau f}{Gf} \quad \text{...... xxviii(b)}$$

And

$$\gamma_m = \frac{\tau m}{Gm} \quad \text{xxviii(c)}$$

Where

$G_{12,f,m}$ = shear moduli of composite, fiber, and matrix respectively.

$$\frac{\tau c}{G12} t_c = \frac{\tau f}{Gf} t_f + \frac{\tau m}{Gm} t_m \quad \text{..... ixxx}$$

The shear stresses in the fiber, matrix and composite are assumed to be

Equal ($\tau_c = \tau_f = \tau_m$), giving

$$\frac{1}{G12} = \frac{1}{Gf}\frac{tf}{tc} + \frac{1}{Gm}\ \frac{tm}{tc} \qquad \text{.... xxx}$$

Because the thickness fractions are equal to the volume fractions

$$\frac{1}{G12} = \frac{Vf}{Gf} + \frac{Vm}{Gm} \qquad \text{... xxxi}$$

FEM is widely used in almost all fields of Science and Engineering. It is used to analyze problems of structural, heat transfer, fluid flow, seepage, lubrication, electric and magnetic fields, fracture mechanics and in many other fields. Numerous software packages based on FEM or FEA have been developed such as NASTRAN, ANSYS, STAAD, SDRC/I-DEAS, ABAQUS and COSMOS etc. ANSYS supports almost all types of elements with many facilities for all fields of engineering.

The solution of a continuum problem by finite element method usually follows an orderly step by step process the following steps shown in general how the finite element method works [4-7].

Discretization: Discretization of a given continuum. The essence of the finite element method id to divide a continuum (i.e. problem achieved by replacing the continuum by a set of key points called nodes) which when connected properly produced the elements. The collection of nodes and elements forms the finite element mess. A variety of element shapes are available. The analyst are designer can mix the elements types to solve one problem. The larger number of elements and nodes, the most accurate the finite element solution, but the more expensive the solution is, more memory space fed to store the finite element model, and more and more computer time is needed to obtain the solution.

Select the Suitable Solution Approximation: The variation of the unknown (called field variable) in problem is approximated within each element by a polynomial. The field variable may be scalar (temperature) or a vector (horizontal and vertical displacements). Polynomials are usually used to integrate and differentiate. The degree of polynomial depends in the number of nodes per element, the number of unknowns at each node, and certain continuity requirement along element boundaries.

Developing Elements Matrices and Equations: The finite element formulation presented in the text section involves transformation of governing equilibrium equations from the continuum domain to the element. Once the nodes and material properties of a given element are define, its corresponding matrices.

Four methods are available to derive element matrices and equations. They are, Direct method, Vibrational method, Weighted residual method, and Energy method.

Assemble the Element Equation: The individual element matrices are added together by summing the equilibrium equations of the elements to obtain the global matrices and system of algebraic equations. If boundary conditions are not given, wrong results are obtained, or singular systems of equations may result.

Solving the Unknowns: Solve for the unknowns at nodes. The global system of algebraic equations is solved via gauss elimination methods to provide the values of the field variables at the nodes of the finite element mesh. Field variables and their derivatives of the nodes of complete finite element solution of original continuum problem before discretization described. Values at other points inside the continuum other than the nodes are possible to obtain, although it is not customarily done.

Interpret the Result: The final step is to analyze the solution and the result obtained from the previous setup to make design decisions. The correct interpretation of these results requires a sound background in engineering and FFEA.

The basic premise of the FEM is the solution and region can be analytically modeled or approximated by replacing with an assemblage of discrete elements. Since these elements can put together in a variety of ways, they can be used to represent exceeding complex shapes. The important features to the FEM which sets apart from the other approximate numerical methods is this ability to format solution for individual elements before pulling them together to represent the entire problem. Another advantage of FEM is variety of ways in which one can formulate the properties of individual elements.

In order to reduce laboratory and experiments expenses, one would try to make predictions of a new materials behaviour and response by numerical simulations, with the chief goal being to speed up the trial and error experimental testing and to be able to stimulate real phenomena that occur at the micro level of the composites that cannot be accurately implemented in the existing analytical models. The recent dramatic increase in computational power available for mathematical modelling and simulation raises the possibilities that modern numerical methods can play a significant role in analysis of heterogeneous microstructures.

ANSYS is one of the finite element analysis (FEA) software which is working on computer based numerical technique for calculating the strength and behaviour of engineering structures. It can be used to calculate deflection, stress, vibration, buckling behaviour and many other phenomena. It can be used to analyze either small or large-scale deflection under loading or applied displacement. It can analyze elastic deformation, or permanently bent out of shape plastic deformation. The computer is required because of astronomical number of calculations needed to analyze large structure. The power and low cost of modern computers has made Finite Element Analysis available to many disciplines and companies. These are useful for problems with complicated geometries, loadings, and material properties where analytical solutions cannot be obtained.

In the finite element method, a structure is broken down into many small simple blocks or elements. The behaviour of an individual element can be described with a relatively simple set of equations. Just as the set of elements would be joined together to build the whole structure, the equations describing the behaviour of individual elements are joined into an extremely large set of equations that describe the behaviour of the whole structure. The computer can solve this set of simultaneous equation. From the solution, the computer extracts the behaviour of individual elements. From this, it can get the stress and deflection of all parts of the structure. The stress will be used, to see if the structure is strong enough. The basic concept in the physical interpretation of the FEM is the subdivision of the mathematical model into disjoint (non-overlapping) components of simple geometry called finite elements or elements for short. The response of each element is expressed in terms of a finite number of degrees of freedom characterized as the value of an unknown function, or functions, at set of nodal points. FEM is a discretization method; the number of degrees of freedom of a FEM model is necessarily finite. They are collected in a column vector called u. This vector is generally called DOF vector or state vector. The term nodal displacement for u is reserved to mechanical applications [8-17].

The objectives of this research are outlined below:

- To estimate the variation of longitudinal young's modulus and transverse young's modulus for various fiber volume fractions for vinylester and glass fiber.
- To estimate the variation of in plane shear modulus for various fiber volume fractions for the three different types of composite materials.
- To estimate the variation of major poison ratio for various fiber volume fractions for the three different types of composite materials.
- To estimate the variation of interfacial normal stresses for various fiber volume fractions for the three different types of composite materials.
- To estimate the variation of interfacial shear stresses for various fiber volume fractions for the three different types of composite materials.
- To estimate the variation of longitudinal young's modulus, transverse young's modulus, In plane Shear modulus and Major Poisson's ratio of angle lamina whose fiber orientations varies from o to 9o for three different materials.

Methodology

In the study of the Micromechanics of fiber reinforced materials, it is convenient to use an orthogonal coordinate system that has one axis aligned with the fiber direction. The 1-2-3 Coordinate system shown in figure is used to study the behaviour of unit cell. The 1 axis is aligned with the fiber directions, and the 2 axis is in the plane of the unit cell and perpendicular to the plane of the unit cell and is also perpendicular to the fibers. The isolated unit cell behaves as a part of large array of unit cells by satisfying the conditions that the boundaries of the isolated unit cell remain plane. Due to symmetry in the geometry, material and loading of unit cell with respect to 1-2-3 coordinate system it is assumed that one fourth of unit cell is sufficient to carry out the present analysis.

Geometry:

The dimensions of the finite element model are taken as

- X=100 units,
- Y=100 units,
- Z=200 units

The radius of fiber is calculated is varied to the corresponding fiber volume.

A_f cross section area of fiber
a cross sectional area of unit cell
r radius of fiber
l edge length of square unit cell
V_f volume fraction of fiber

Element Type:

The element SOLID186 of ANSYS V17.0 used for the present analysis is based on a general 3D state and is suited for modeling 3D solid structure under 3D loading. The element has 20 nodes having one degree of freedom at each node : translation in the node x, y and z directions respectively. Static analysis is performed by considering pre-determined conditions (i.e. boundary conditions and point load) using ANSYS 17.0 software.

Boundary Conditions:

Due to the symmetry of the problem the following symmetric boundary conditions are used.

- At x=0,Ux=0
- At y=0,Uy=0
- At z=0,Uz =0

In addition, the following multi point constraints are used

- The Ux of all the nodes on the Area at x=100 is same
- The Uy of all the nodes on the Area at y=100 is same
- The Uz of all the nodes on the Area at z=100 is same

Materials:

Fiber reinforced composite material considered in this investigation are vinylester / glass fiber/carbon fiber/Kevlar fiber composites.

Typical properties of Fiber and vinyl ester

Glass Fiber

Longitudinal Modulus = E_1 = 72 GPa

Transverse Modulus = E_2 = 72 GPa

Poisson's Ratio = ν = 0.21

Shear Modulus = G_{12} = 33 GPa

Vinylester matrix

Longitudinal Modulus = E_1 = 19.74 GPa

Transverse Modulus = E_2 = 19.74 GPa

Poisson's Ratio (ν) = 0.324

Shear Modulus = G_{12} = 4.40 GPa

The procedure is explained step by step as follows:

Step1 Preferences→ Structural→ Discipline→ Option h-Method→ O.k

Step2 Preprocessor→ Element type→ Add/Edit/Delete→ Add→ solid→ 20 node 186 → O.k →Close

Step3 Preprocessor→ Material Properties→ Material Model → Structural→Linear→ Elastic→ Orthotropic→ Enter Material Properties→ O.k

Step4 Preprocessor→ Material Properties→ Material Model → Structural→Linear→ Elastic→ Isotropic→ Enter Material Properties→ O.k

Step5 Preprocessor→ Modeling →Create→ Volume→ Block→ By Dimensions→ Give dimensions→ O.k

Step 6 Preprocessor→ Modeling →Create→ Volume→ Cylinder→ ByDimensions→ Give dimensions→ O.k

Step 7 Preprocessor→ Modeling→ Operate→ Booleans→ Subtract → Volume→Select volume→ O.k

Step 8 Preprocessor→ Modeling→ Create→ Volume→ Cylinder→ partial Cylinder→ By Dimensions→ Give dimensions→ O.k

Step 9 Preprocessor→ Modeling→ Operate→ Booleans→ Glue → Volume→Select volume→ O.k

Step 10 Preprocessor→ meshing→ Size Controls→ Manual size →Global →size→ 5→ O.k

Step 11 Preprocessor→ Meshing →Mesh tool→ Mesh →Set→ Material Id→ O.k

Step 12 preprocessor→ Loads→ define loads → Apply→ Structural → displacement→ on nodes→ pick box→ select nodes→ O.k

Step 13 Preprocesssor→ Coupling/ceqn→ couple DoF's→ on Nodes→pick box→ Ux, Uy, Uz → O.k

Step 14 preprocessor→ Loads→ define loads → Apply→ Structural → pressure→ on areas→ select area→ enter value→ O.k

Step 15 Solution→ Analysis type → New Analysis → static → O.k

Step 16 Solution → Solve→ current LS→ O.K→ Close

Step 17 General post Processor→ plot results→ contour plots → nodal solution→ DOF solution→ X,Y displacement → Deformed Shape only→Auto calculated→ O.k

Step 18 General post Processor→ plot results→ contour plots → nodal solution→ Stress → XY-Direction Stress→ Deformed Shape only→Auto calculated→ O.k

Step 19 General post Processor→ plot results→ contour plots → nodal solution→ Stress → XY-Direction Strain→ Deformed Shape only →Auto calculated→ O.k

Calculation Of Radius For 30 % Volume Fraction Of Fiber :

$$\Pi d^2/4 = V_f\,(a^2)$$

Where ' a ' is edge length of square unit cell.

$$a = 100 \text{ units.}$$

$$\Pi d^2/4 = 0.3 * 100 * 100$$

$$= 3000$$

$$d^2 = 3819.71$$

$$d = 61.81 \text{ units}$$

$$r = 30.909 \text{ units}$$

Calculations of mechanical properties by strength of materials approach: Strength of materials approach is also called " rule of mixtures ''.

Longitudinal Young's Modulus: $E_1 = E_f V_f + E_m V_m$

$$E_f = 72 \text{ GPa}$$

$$E_m = 19.74 \text{ GPa}$$

$$V_f = 0.3$$

$V_m = 0.7$

Where V_f is the volume fraction of fiber.

V_m is the volume fraction of matrix.

Then $E_1 = 35.418$ GPa

Transverse Young's Modulus: $\frac{1}{E2} = \frac{Vf}{Ef} + \frac{Vm}{Em}$

$E_f = 72$ GPa

Em = 19.74 GPa

$V_f = 0.3$

$V_m = 0.7$

Where V_f is the volume fraction of fiber.

V_m is the volume fraction of matrix.

Then $E_2 = 25.23$ GPa.

Poisson's Ratio : $V_{12} = V_f\, \nu_f + V_m\, \nu_m$

$V_f = 0.21$

$\nu_m = 0.324$

$V_f = 0.3$

$V_m = 0.7$

Where V_f is the volume fraction of fiber.

V_m is the volume fraction of matrix.

Then $V_{12} = 0.2898$

In- Plane Shear Modulus: $\frac{1}{G12} = \frac{Vf}{Gf} + \frac{Vm}{Gm}$

$G_f = 33$ GPa

$G_m = 4.40$ GPa

$V_f = 0.3$

$V_m = 0.7$

Where , V_f is the volume fraction of fiber.

V_m is the volume fraction of matrix.

Then $G_{12} = 5.94$ GPa.

RESULTS AND ANALYSIS

This chapter discusses the results of static analysis of the unidirectional fiber reinforced viz., Vinyl-Ester/Glass composite. This static analysis of the unidirectional fiber reinforced composite is performed by using ANSYS17.1 for different volume fractions (10%-70%).From the static analysis of the F.E model of FRP composite the Engineering constants like Longitudinal modulus (E_1),Transverse modulus (E_2), In-Plane Shear modulus (G_{12}) and Major poison's ratio (ν_{12}) determined. In addition to that, interfacial stresses at the fiber-matrix interface for various volume fractions of composite materials are determined and the results are validated with Mechanics of Solids approach method.

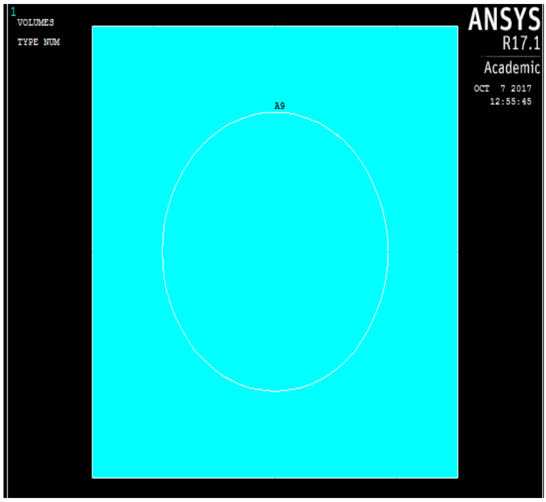


Figure 4: Formed rectangular composite for 30% volume fraction.

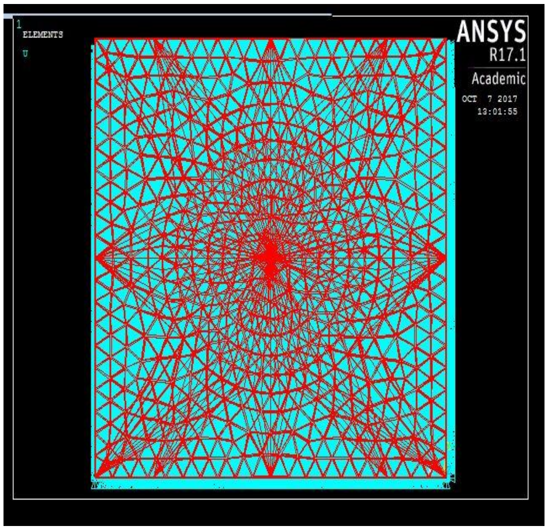


Figure 5: Constant pressure is applied on the composite of 30% fiber volume fraction.

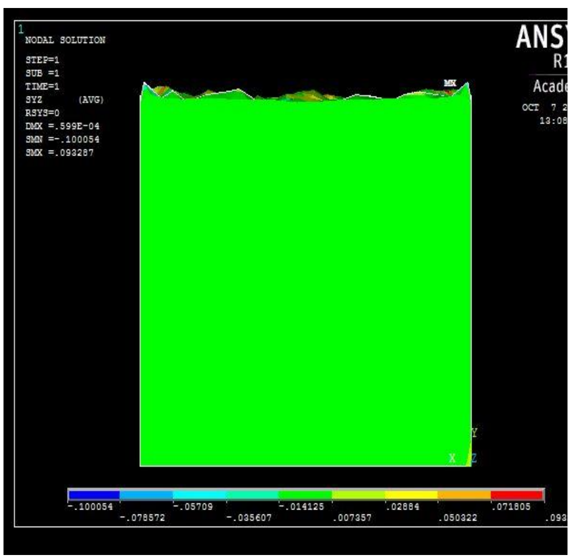


Figure 6: Shear stress in transverse direction for 30% fiber volume fraction.

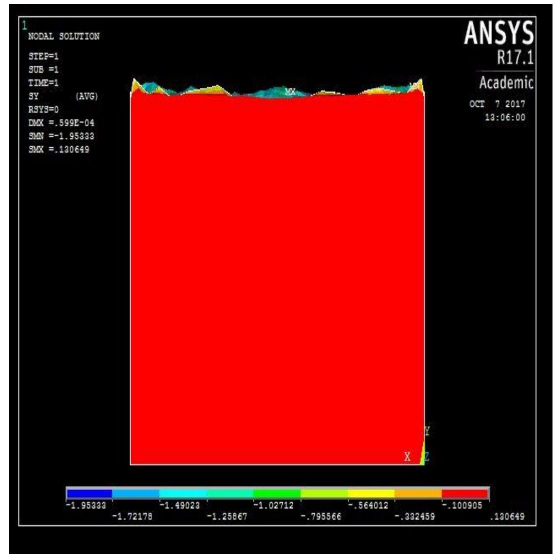
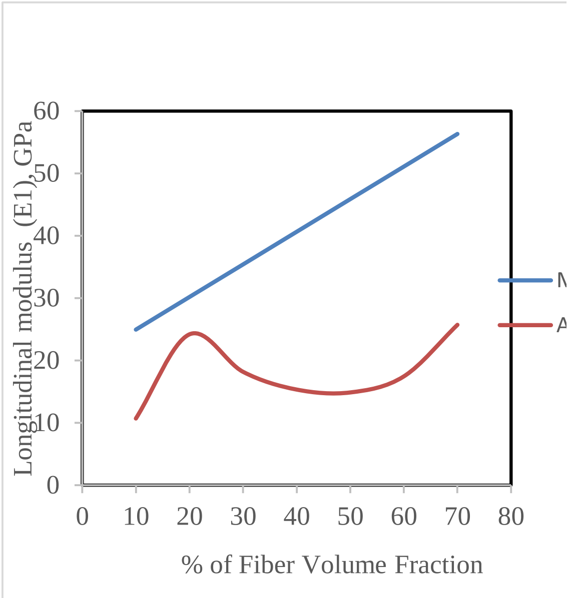


Figure 7: Stress in transverse direction for 30% fiber volume fraction.

<u>Micromechanical Analysis of Vinyl Ester/ Glass Composite:</u>

The F.E model of unidirectional glass fiber reinforced composite material subjected to longitudinal loading has been analyzed using Ansys software package. From the static analysis of composite material at different fiber volume fractions (10% to 70%) different engineering constants like E_1, E_2, G_{12} and ν_{12} are determined. The figures from 4-7 indicate the variation of engineering constants and the interfacial stresses that are induced at the fiber-matrix interface for the material vinylester/epoxy.



igure 8: Variation of E_1 with fiber volume fractions.

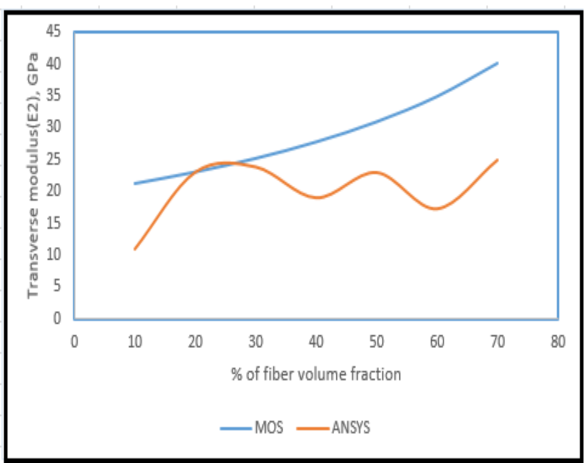


Figure 9: Variation of E_2 with fiber volume fractions.

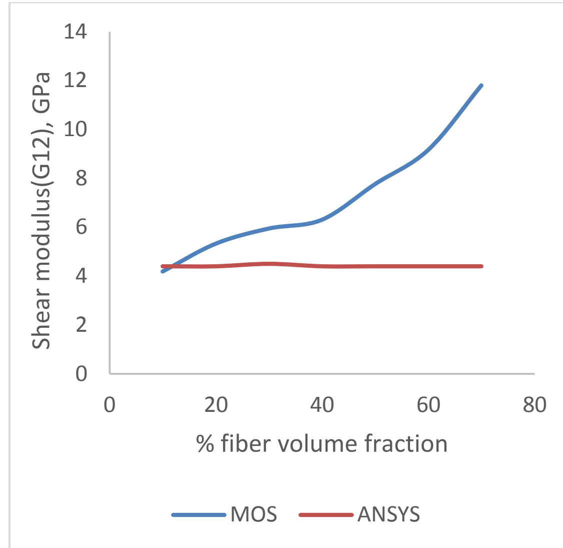


Figure 10: Variation of G_{12} with fiber volume fractions.

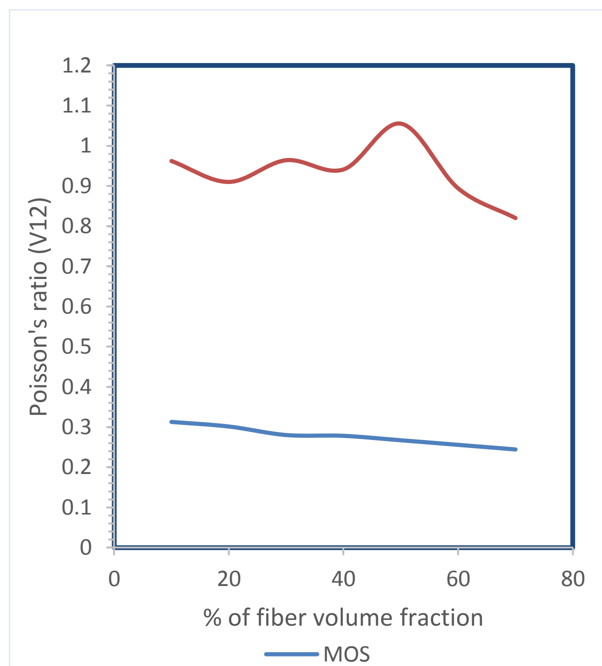


Figure 11: Variation of ν_{12} with fiber volume.

Analysis of Results:

- ✓ From the figure 8 the longitudinal young's modulus E_1 increases linearly with increase in fiber volume fraction.
- ✓ From the figure 9 the transverse young's modulus E_2 increases gradually with increase in fiber volume fraction.
- ✓ From the figure 10 the in plane shear modulus G_{12} increases gradually with increase in fiber volume fraction.
- ✓ From the figure 11 the poison's ratio γ decreases linearly with increase in fiber volume fraction.

CONCLUSION

In this work micromechanical analysis of three different types of uni-directional fiber reinforced (Viz., Vinyl-Ester/Glass) composite materials has been carried out using FEA software package ANSYS17.1 to evaluate several elastic constants viz,. E_1 , E_2, G_{12}, ν_{12}.Results are compared with the results obtained by using the Rules of Mixture. The longitudinal young's modulus (E_1) increases linearly with increase in fiber volume fraction for the fiber reinforced (viz,.Vinyl-Ester/Glass) composite material considered in this investigation. Fiber volume fraction doesn't have much influence on the transverse young's modulus (E_2) for the fiber reinforced composite material considered in this investigation. In-plane shear modulus (G_{12}) gradually increases with Fiber volume fraction for the composite material considered in this investigation. Major poison's ratio (ν_{12}) decreases slightly with increase in the Fiber volume fraction for the composite material considered in this investigation.

REFERENCES

[1] Chen C H & Cheng S, Trans ASME J Appl Mech, 37 (1970) 186-189.

[2] Syed Altaf Hussain, B. Sidda Reddy. and Nageswar Reddy. V, "prediction of elastic properties of FRM composite lamina for Longitudinal loading" Asian Research publishing Network(ARPN), vol. 3 (6), 2008, pp. 70-75.

[3] Taurdon, G. p. " Evaluation of interfacial normal strength in a SCS-0/Epoxy composite with cruciform specimens", Composites and Technology, vol. 60, 2000, pp. 2281-2295.

[4] Wang, Q. and chiang, F. p., "Experimental characterization of interface mechanical properties of composites''. ° Composites part:B, vol. 27B, 1996, pp. 123-128.

[5] Dragan D. Kreculj, "Stress Analysis in a Unidirectional Carbon/Epoxy Composite Material", FME Transactions, vol.36 (3), 2008, pp. 127-132.

[6] Krishna Vihari.N, Phani Prasanthi. P and Bala Krishna Murthy. v "Micro mechanical Analysis of FRP Composites Subjected to Thermal Loading" International Journal of Mechanics and Solids, vol. 6, (3) 2011, pp. 167-174.

[7] Sivaji Bubu et.al "Micro mechenical analysis of FRP hybrid Composite lamina for in plane transverse loading ", Indian journal of engineering and Material science,IJEMS, Vol.15,October 2008, pp. 382-390,

[8] Robertson ,D.D. and mall, S., "Fiber-Matrix interphase Effects upon Transverse behaviour in metal matrix Composites" . Journal of Composites Technology & Research, JCTRER, vo]. 14 (1),spring 1992, pp. 3-11.

[9] Asp, L.E, Berglund. L.A, "Effects of fibre and interface on matrix initiated transverse failure in polymer composites ".Composites Science and Technology, vol. 56, 1996, pages 657-665.

[10] M W Hyer, 1998. Stress Analysis of Composite Materials. WCB McGraw-Hill.

[11] R M Jones. 1975, Mechanics of Composite Materials. Scripta Book Company.

[12] Autar K. Kaw. 2006. Mechanics of Composite Materials. CRC press.

[13] Issac M Danial. 1994 Engineering Mechanics of Composites. Oxford University Press.

[14] P.P.Camanho, F.M.Andrade Pires, S.T.Pinho, Micromechanical analysis of polymer composites reinforced by unidirectional fibres: Part II – Micromechanical analyses, International Journal of Solids and Structures, 50, 11–12, 2013, 1906-1915.

[15] Luis Pablo Canal, Carlos González, Javier Segurado, Javier LLorca, Intraply fracture of fiber-reinforced composites: Microscopic mechanisms and modelling, Composites Science and Technology, 72, 11(28)2012,1223-1232.

[16] Thomas Hobbiebrunken, BodoFiedler, Masaki Hojo, Shojiro Ochiai, Karl Schulte, Microscopic yielding of CF/epoxy composites and the effect on the formation of thermal residual stresses, Composites Science and Technology, 65(10), 2005,1626-1635.

[17] T.H.Hsieh, A.J.Kinloch, K.Masania, A.C.Taylor, S.Sprenger, The mechanisms and mechanics of the toughening of epoxy polymers modified with silica nanoparticles, Polymer, 51, 26(10) 2010, 6284-6294.