Journal of Manufacturing Engineering, 2007, Vol.2, Issue.4

MICRO MECHANICAL ANALYSIS OF FRP LAMINA USING FINITE ELEMENT METHOD FOR TRANSVERSE LOADING

M.S. Durga Prasad, R. Sasikanth
G. Sambasiva Rao and Dr. V. Bala Krishna Murthy*
Department of Mechanical Engineering, Prasad V. Potluri Siddhartha Institute of Technology,
VIJAYAWADA – 520 007.
e-mail ID: vbkm64@rediffmail.com, ms302.mech@gmail.com.

ABSTRACT

The present investigation studies the micromechanical behaviour of the square unit cell of a Fiber Reinforced Composite lamina. A three-dimensional finite element model has been developed from the unit cells of square array of the composite to predict the Young's modulus (E_2) and poison's ratios (ν_{23}, ν_{21}) of Graphite and Boron fiber reinforced lamina. The stresses at the fiber-matrix interface are also determined from this model. The finite element software ANSYS has been successfully executed to evaluate the properties and stresses. The variation of the stresses at the fiber-matrix interface with respect to the angular location is discussed.

1. INTRODUCTION

The Mechanics of Materials deals with stresses, strains and deformations in engineering structures subjected to mechanical and thermal loads. A common assumption in the mechanics of conventional materials such as steel and aluminum is that they are homogeneous and isotropic continua. For a homogeneous material, properties do not depend on the location; and for an isotropic material, properties are same in all the directions. Fiber reinforced composites, on the other hand, are microscopically inhomogeneous and orthotropic. As a result, the mechanics of fiber reinforced composites are far more complex than that of conventional materials.

Micromechanics analysis is intended to study the distribution of stress and strains within the micro regions of the composite under the loading. This study will be particularized to simple loading and geometry for evaluating the average or global stiffness and strengths of the composites [1, 2]. Micromechanical analysis can be carried theoretically using the principles of continuum mechanics, and experimentally using mechanical, photo-elasticity, ultrasonic tests, etc. The results of micromechanics will help

- To understand load sharing, microscopic structure, etc., within composites,
- To understand the influence of microstructure (arrangement of fibers) on the properties of composite,
- To predict the average properties, and
- To design the materials, i.e., constituents and their distribution, for a given situation.

The properties and behaviour of composite are influenced by the properties of fiber and matrix, interfacial bond and by its micro structure. Micro structural parameters that influence the composite behaviour are fiber diameter, length, volume fraction, packing and orientation of fiber. Sun et al [3] established a vigorous mechanics foundation for using a Representative Volume Element (RVE) to predict the mechanical properties of unidirectional fiber composites. Li [4] has developed two typical idealized packing systems, which have been employed for unidirectional fiber reinforced composites, viz. square and hexagonal unit cells to accommodate fibers of irregular cross sections and imperfections asymmetrically distributed around fibers.

In the present work the FRP lamina has been idealized as an array of square unit cells and the mechanical properties E_2, ν_{21} and ν_{23}, and the stresses at the fiber-matrix interface are determined using finite element analysis.

Corresponding author: vbkm64@rediffmail.com

2. SQUARE ARRAY OF UNIT CELLS

A schematic diagram of the unidirectional fiber composite is shown in Fig.1 where the fibers are arranged in the square array. It is assumed that the fiber and matrix materials are linearly elastic. A unit cell is adopted for the analysis. The cross sectional area of the fiber relative to the total cross sectional area of the unit cell is a measure of the volume of fiber relative to the total volume of the composite. This fraction is an important parameter in composite materials and is called fiber volume fraction (V_f).

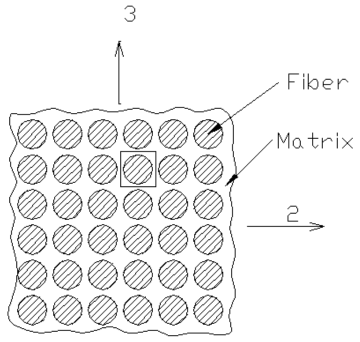

Fig.1. Concept of Unit Cells

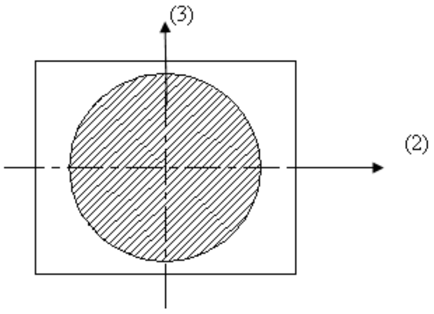

Fig.2. Isolated Unit Cell of Square packed array

3. FINITE ELEMENT MODEL

The 1-2-3 Coordinate system shown in Fig.2 is used to study the behaviour of unit cell. 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.

It is assumed that the geometry, material and loading of unit cell are symmetric with respect to 1-2-3 coordinate system. Therefore, a one-eighth portion of the unit cell is modeled for the present work (Fig.3).

3.1 Geometry

The dimensions of the finite element model are taken as

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

The radius of the fiber is calculated as 50.46 units, so that the fiber volume fraction becomes 0.2.

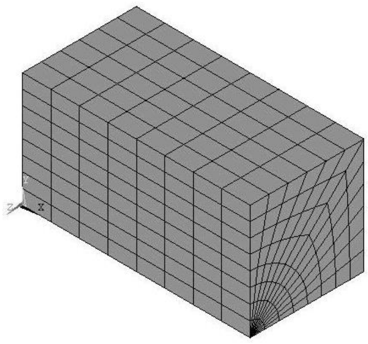

Fig.3. Finite element mesh on one-eighth portion of the unit cell

3.2 ELEMENT TYPE

The element used for the present analysis is SOLID 95 of ANSYS [5] which is developed based on three-dimensional elasticity theory and is defined by 20 nodes having three degrees of freedom at each node: translation in the node x, y and z directions.

3.3 MATERIALS

The properties of the constituent materials used for the present analysis are given in Table 1.

Table1. Properties of Constituents

S. No.	Material	E (GPa)	ν	G (GPa)
1	Boron [6]	400	0.2	--
2	Graphite [7]	E_1=233 E_2=23.1	ν_{12}= 0.2 ν_{23}=0.4	G_{12}=8.96 G_{23}=8.27
3	Epoxy [7]	4.62	0.36	--

3.4 Loading

Uniform Tensile load of 1 MPa is applied on the area at z = -100 units.

3.5 Boundary conditions

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

- At x = 0, $U_x = 0$
- At y = 0, $U_y = 0$
- At z = 0, $U_z = 0$

In addition the following multi point constraints are used.

- The U_x of all the nodes on the line at x = 200 is same
- The U_y of all the nodes on the line at y =100 is same
- The U_z of all the nodes on the line at z = -100 is same

4. RESULTS

The mechanical properties of the lamina are calculated using the following expressions.

Young's modulus, $E_2 = \dfrac{\sigma_2}{\varepsilon_2}$

Poisson's Ratio, $\nu_{21} = \nu_{31} = -\dfrac{\varepsilon_1}{\varepsilon_2}$;

$\nu_{23} = -\dfrac{\varepsilon_3}{\varepsilon_2}$

Where σ_2 = Stress in y-direction.,
ε_2 = Strain in y-direction,
ε_3 = Strain in z-direction

Sufficient number of convergence tests are made and the present finite element model is validated by comparing the Young's modulus E_1 of Graphite-Epoxy lamina predicted using the finite element method with the value obtained from exact elasticity theory. Close agreement is observed (Table 2). Subsequently this finite element model is extended to predict the transverse modulus. Table 3 presents the mechanical properties predicted from the present analysis.

Table 2

Material	**E_2 (GPa)**	**ν_{21}**	**ν_{23}**
Boron-Epoxy	7.631	0.0293	0.5
Graphite-Epoxy	6.691	0.0431	0.5

Table 3

Material	**Exact Elasticity Theory [7]**	**Present FEM model**
Graphite-Epoxy	50.2 GPa	50.3 GPa

The following stresses are also computed at the fiber-matrix interface.

- σ^f_n = Normal stress in the fiber at the interface
- σ^m_n = Normal stress in the matrix at the interface
- τ^f_{ns} = Shear stress in the fiber at the interface.
- τ^m_{ns} = Shear stress in the matrix at the interface.
- σ^f_c = Circumferential stress in the fiber at the interface
- σ^m_c = Circumferential stress in the matrix at the interface
- σ^f_1 = Fiber directional stress in the fiber at the interface
- σ^m_1 = Fiber directional stress in the matrix at the interface

5. DISCUSSION

Fig. 4 shows the variation of normal stress at the fiber-matrix interface. In this case the stresses in fiber and matrix are same for all the values of θ. where θ is the angle measured from direction 2 in the counter-clockwise direction. The stresses decrease with increase in θ with a maximum value at $\theta = 0^0$ for both the materials. The stresses are more in Boron-Epoxy for the values of θ between 0^0 and 52.5^0, and beyond $\theta = 70^0$.

Fig. 5 shows the variation of circumferential stress at the fiber-matrix interface. The stresses in the fiber increase with increase in θ with a maximum value at 90^0 for both the composites, whereas, the stresses in the matrix decrease with increase in θ with a maximum at 0^0. There is no considerable difference in the magnitudes of the stresses in the fiber when compared to the stresses in the matrix. In the matrix, the magnitude of the stresses in Boron-Epoxy are found to be more between $\theta = 0^0$ and 37.5^0.

The variation of the shear stresses at the fiber-matrix interface is shown in Fig. 6. In this case also the stresses are observed to be same in fiber and matrix for both the composites. Magnitudes of these stresses increase with increase in θ up to 37.5^0 and later decreases. It is observed that the magnitude of the shear stresses is more in Boron-Epoxy compared with the magnitude of shear stresses in Graphite-Epoxy except at $\theta = 0^0$ and 90^0 where the stresses are zero in both the materials.

Fig. 7 shows the variation of the normal stress in the fiber direction at the fiber-matrix interface. Stresses in the fibers are observed to be compressive and in the matrix they are tensile for all the values of θ in Graphite-Epoxy, and up to 67.5^0 of θ in Boron-Epoxy. The stresses in the matrix decrease, with increase in θ with a maximum value at 0^0. In fiber, the variation of stresses with respect to θ in both the composites is negligible. The magnitude of the stresses in matrix of the Boron-Epoxy composite are observed to be more than that of Graphite-Epoxy up to $\theta = 45^0$ and beyond 70^0.

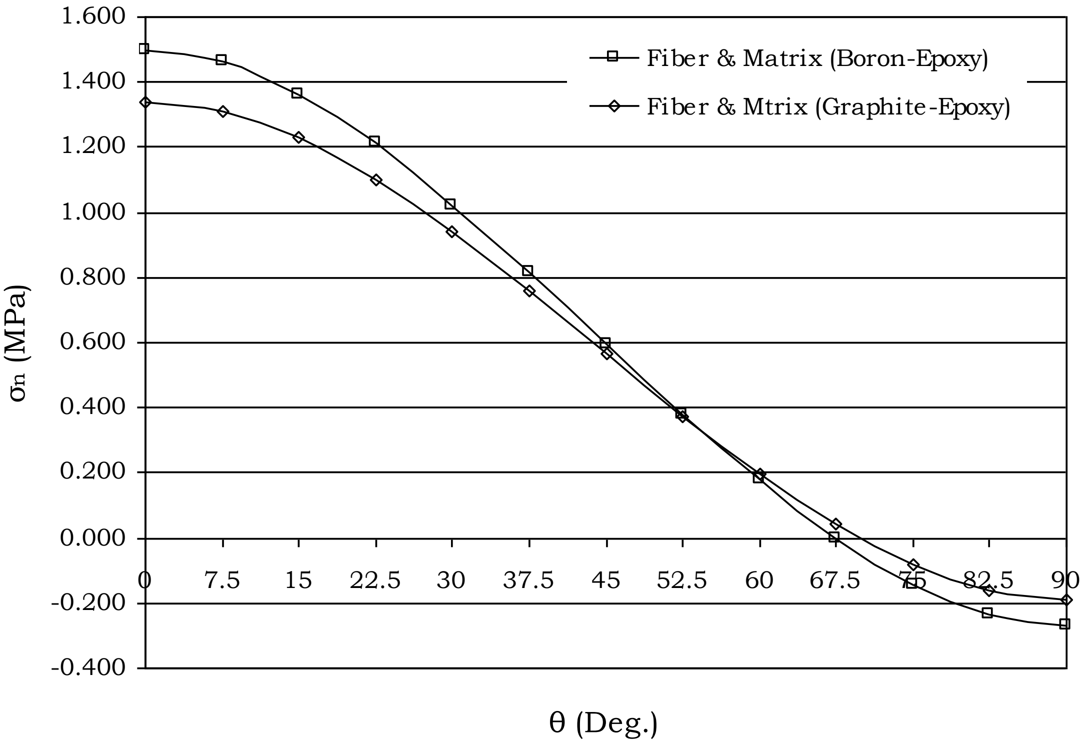

Fig. 4 Variation of Interface normal stress with respect to θ for Fiber & Matrix

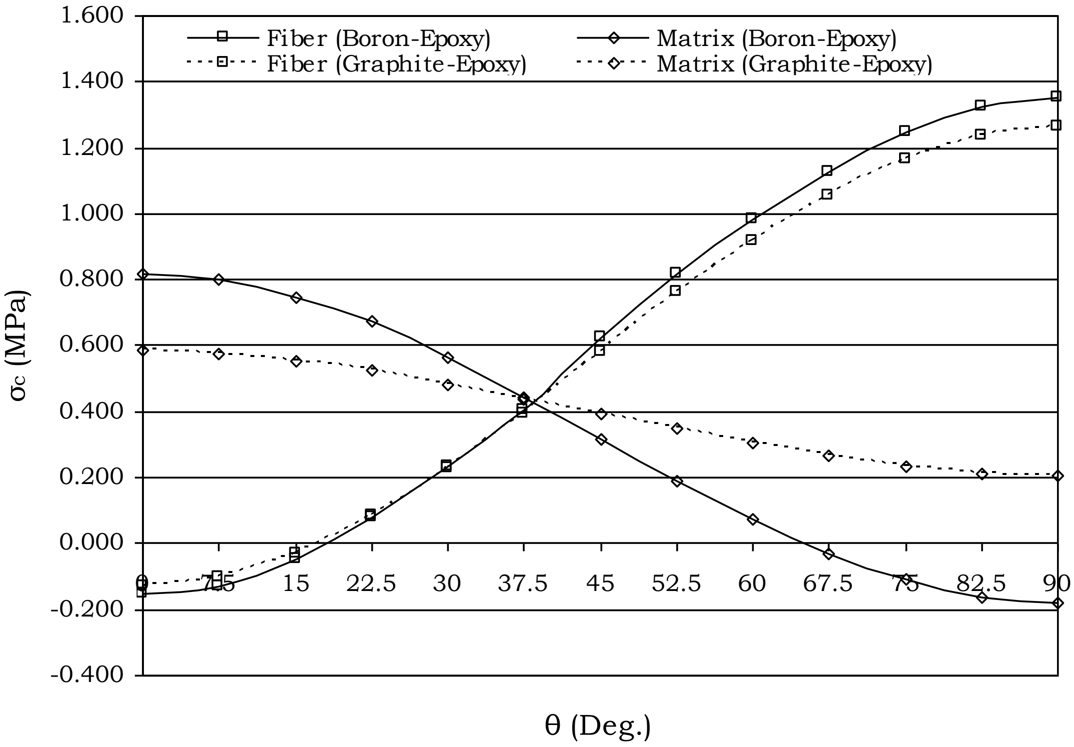

Fig. 5 Variation of Interface circumferential stress with respect to θ

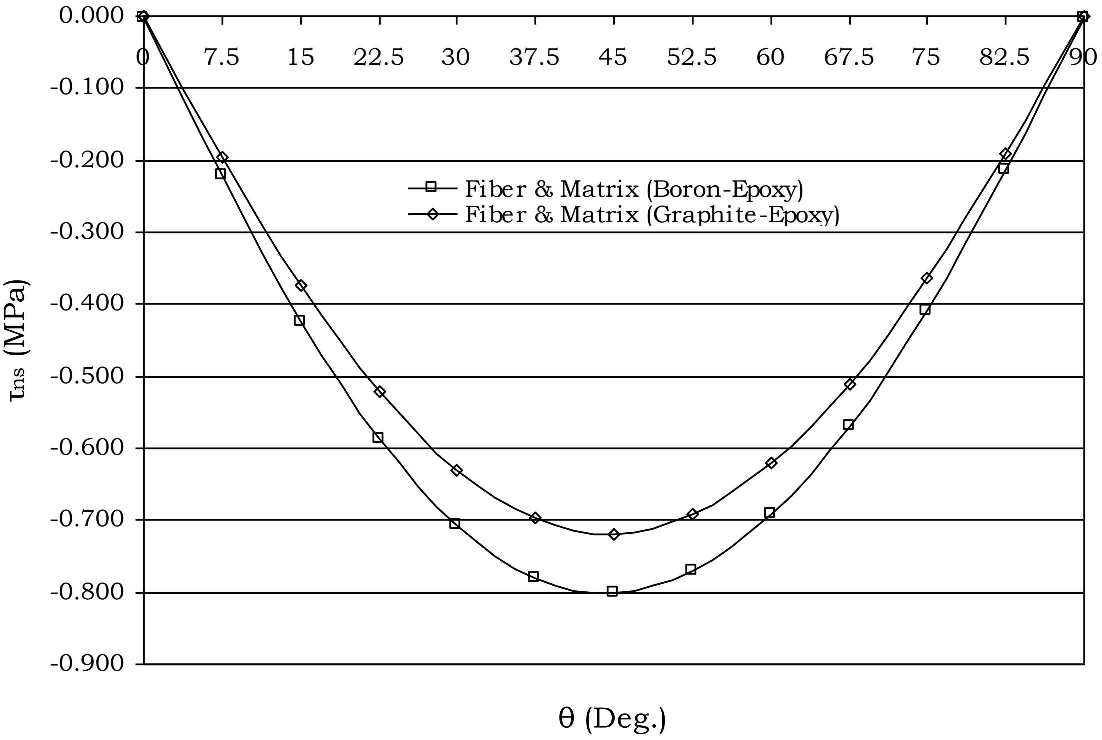

Fig. 6 Variation of Interface shear stress with respect to θ for fiber & matrix

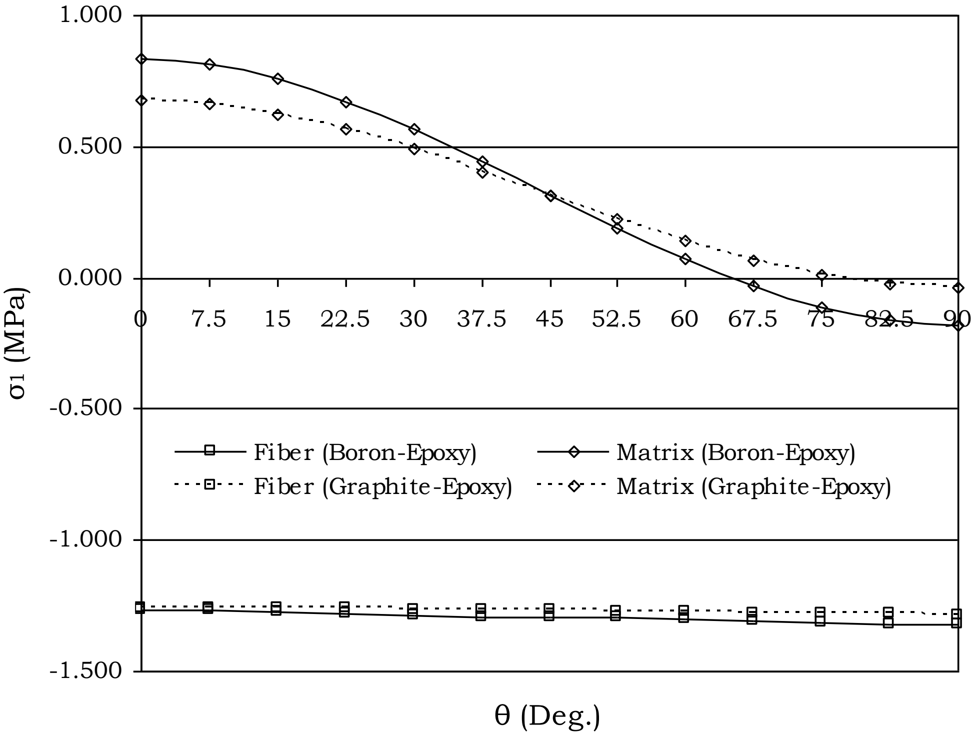

Fig.7 Variation of Interface fiber directional stress with respect to θ

6. CONCLUSIONS

The micromechanical behaviour of FRP lamina has been studied using finite element method. The Young's modulus and poison's ratios of two different materials are predicted for 20% of fiber volume fraction. The stresses at the fiber-matrix interface are also computed. The following conclusions are drawn.

- The Young's modulus (E_2) is found to be more in Boron-Epoxy lamina indicating that this combination possesses good strength.
- The maximum values of normal stresses are observed at 0^0 of θ for both the composites. This may result in the damage of the interface at 0^0 of θ. (Fig.4)
- The circumferential stress of matrix material is found to be maximum at $\theta = 0^0$ for both the lamina. This indicates that the failure of the matrix may originate at this location. Similarly, the circumferential stress of fiber material is found to be maximum at $\theta = 90^0$ for both the composites. This may result in the failure of the fiber at $\theta = 90^0$. (Fig.5)
- The maximum values of shear stress is observed at $\theta=45^0$ indicating that the interfacial damage may occur at this location. (Fig.6)

REFERENCES

1. Mohana Rao K. (1986), "Workshop on Introduction to Fiber-reinforced composite materials", NSTL, pp 24-29.
2. Morais AB., (2000) "Transverse Modulii Continuous fiber-reinforced polymers," Composites Science & Technology, 60, pp 997-1002.
3. Sun CT and Vaidya RS., (1996) "Prediction of composite properties from a representative volume element", Composites Science and Technology, 56, pp.171-179.
4. Li. S., (2000) "General unit cells for micromechanical analyses of unidirectional composites", Composites: part A, 32, pp.815-826.
5. ANSYS Reference manual 2006.
6. Hull D. & Clyne T.W., (1996) "An Introduction to Composite Materials", Cambridge university press.
7. Hyer M.W., (1998) "Stress Analysis of Fiber-Reinforced composite materials", Mc. GRAWHILL International edition.