Tensile Test Simulation of Carbon Fiber Reinforced Plastic

Sanjeev Agrawal[1], Rehan Khan[2], Athar Hussain[3]

[1]*M.Tech. Student, School of Engineering, Gautam Buddha University, Greater Noida-201310, Uttar Pradesh, India.*

[2]*M.Tech. Student, School of Engineering, Gautam Buddha University, Greater Noida-201310, Uttar Pradesh, India.*

[3]*School of Engineering, Gautam Buddha University, Greater Noida-201310, Uttar Pradesh, India.*

ABSTRACT

This work is emphasizing on efficient design of multidirectional carbon fiber reinforced with composite material with the help of finite element simulation. A3D model is formed with the help of abaqus software then finally the simulation of the tensile test on the composite specimen is being done .The aim of this analysis is to compute the stress and strain in different plies. Tensile test was carried then and results were analyzed in order to compute Young's modulus, stress, strain and loads on the composite specimen. In this research work two different composite specimens are used with 15 no of plies which stacked in different orientation. For different ply orientation different mechanical strength, failure load and strain are analyzed. A no of orientation series also be carried to verify the best modeling technique. There is lot of complexity behind the micromechanics of the composite material and its quite expansive and time consuming.

Keywords: composite materials, carbon fiber/epoxy, finite element simulation

1. INTRODUCTION

Composite materials are popularly known for their structural applications. The combination of high strength, high modulus, and low density is the characteristics of composites having continuous fibers as the reinforcement [2]. The main application of this paper is to carry out the simulation on composite specimen. The specimen of shell conventional model and semisolid continuum model are compared on grounds of various curves (reaction force, displacement).for input data we need to calculate the ply properties which are obtained from ABAQUS.

The value of elastic modulus is quite higher when load is applied in the direction of orientation of fiber than it is applied transverse to it.

Composite materials may be defined as the matrix of material having fiber reinforcement. When two or more materials combine, they form a newer material with highly desirable properties [2].

When polymer matrix is combined with high modulus fiber reinforcement it gives a rise to a new material which has greater strength/weight and stiffness/weight ratio as compared to steel and various other alloys.

In our literature review, Saha (in his master's thesis) on carbon fiber with Abaqus create the models shell, semisolid, solid and he conducts some experimental work also. In this paper some ply orientation has altered and best results are carried out.

2. THEORETICAL FRAMEWORK

1. Rule of mixture 2. Halpin/Tsai model

As we know that composite materials are made by arranging several distinct layer of unidirectional lamina [3]. It is too much necessary to calculate stiffness while doing modeling of composite material in abaqus.

1. Stiffness of composite lamina according to the rule of mixture

In rule of mixture it is assumed that fibers are parallel throughout the whole composite in unidirectional lamina and there is no slippage between the interfaces of fiber and matrix material as they are already perfectly bonded. Due to this bonding they experience the same strain and this is called isostrain situation [3].

By the rule of mixture if an axial load is applied in longitudinal direction of specimen then it will be shared by fiber and matrix [3].

$$P_c = P_f + P_m \quad \sigma_c A_c = \sigma_f A_f \Leftrightarrow_m A_m$$

$$E_c = E_f V_f + E_m (1-V_f) = E_{11}$$

E_{11} is the longitudinal elastic modulus of composite lamina

$$E_{33} = E_{22} = E_f E_m / (E_f V_m + E_m V_f)$$

$$G_{13} = G_{12} = G_f G_m / (G_f V_m + G_m V_f)$$

3. HALPIN/TSAI MODEL

In order to approximate the modulus of composite materials by considering the geometry of the reinforcing fiber used, he introduced simple semi empirical equation. Until and unless volume fraction of the fiber does not reach 1, these equations are good enough to be applied [3].

$$E_{11}=E_m(1+\zeta_{11}\eta_{11}V_f)/(1-\eta_{11}V_f),\ E_{22}=E_m(1+\zeta_{22}\eta_{22}V_f)/(1-\eta_{22}V_f)$$

$$G_{12}=G_m(1+\zeta_{12}\eta_{12}V_f)/(1-\eta_{12}V_f)$$

$$\zeta_{11}=2(1t)+40(Vf)^{10};\ \zeta_{22}=2(wt)+40(Vf)^{10};\ \zeta_{12}=(wt)^{1.73}+40(Vf)^{10};$$

$$\eta_{11}=(E_f/E_m-1)/(E_f/E_m+\zeta_{11});\ \eta_{22}=(E_f/E_m-1)/(E_f/E_m+\zeta_{22});\ \eta_{12}=(G_f/G_m-1)/(G_f/G_m+\zeta_{12});$$

ζ is a measure of reinforcement and it depends on the fiber geometry, packing geometry and loading conditions.

$\gamma_{12} = V_f V_{pf} + V_m V_{pm}$ for both models. Volume fraction of fiber and matrix is taken as 0.5.

4. MATERIAL AND METHODS: (FINITE ELEMENT SIMULATION)

Nowadays finite element method is used in various field of engineering. With the help of this advanced numerical calculations can be done quite easily.

In FEM the body to be studied is divided into small number of parts called elements and this is called meshed model. The smaller elements are connected in nodes. With increase in number of nodes accuracy improves to much greater extent. When the force is applied on the element, it acts on the nodes and is represented by load vectors. The nodal deflection will give the solution of the equation system. Abaqus is the finest FEM tool to analyze 3d problems.

4.1Part

The test specimen is a rectangular multidirectional carbon fiber of length 300mm, width 25 mm and for gripping 57mm on each end is taken. The fiber is laminated with a thickness of 2.7mm.Casting of a model is done in two dimension, deformable shell planar feature for the shell model and another three dimensional. With the help of this double geometry, comparison can be made in stresses and strains and load value of different modeling structure.

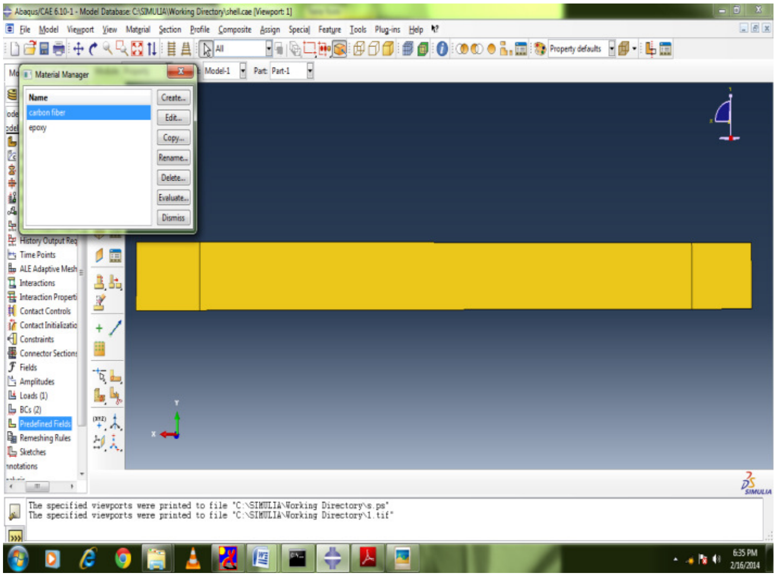

Fig:1 part of the specimen

4.2 Material Property

For the analysis of the result, two different types of elastic lamina type materials are created one for carbon fiber and the other for E-glass composite material. By applying Rule of mixture and Halpin/Tsai method mechanical properties of the plies are calculated. The materials taken are orthotropic and linear in nature.

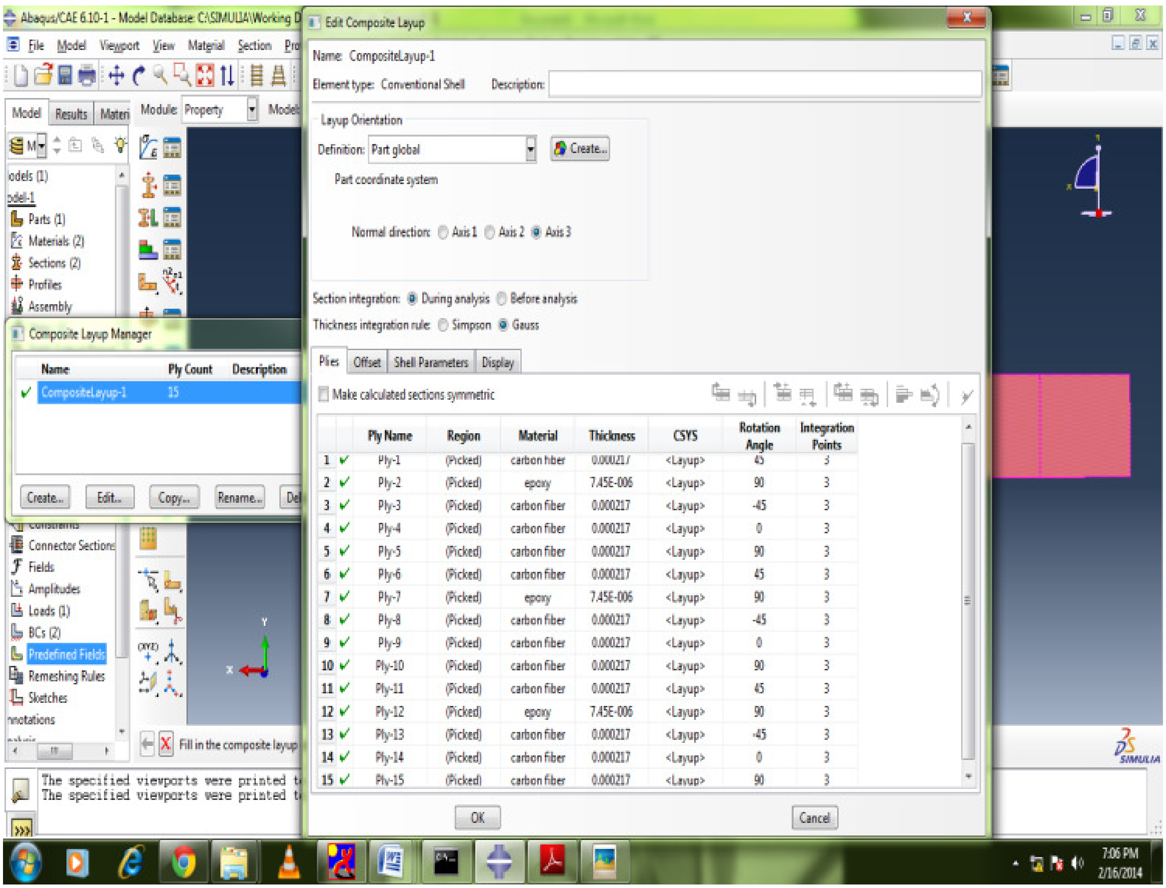

Fig:2 material properties of specimen

4.3Boundary condition and Load

The boundary condition in this test, the specimen is clamped at both ends lower and upper grip. Lower grip is clamped in all direction and upper grip is also clamped but free in longitudinal direction, in which direction the load is applied. In this simulation surface traction load (40KN/m) scenario is used and for shell model and solid model load is applied at one surface and two surfaces respectively.

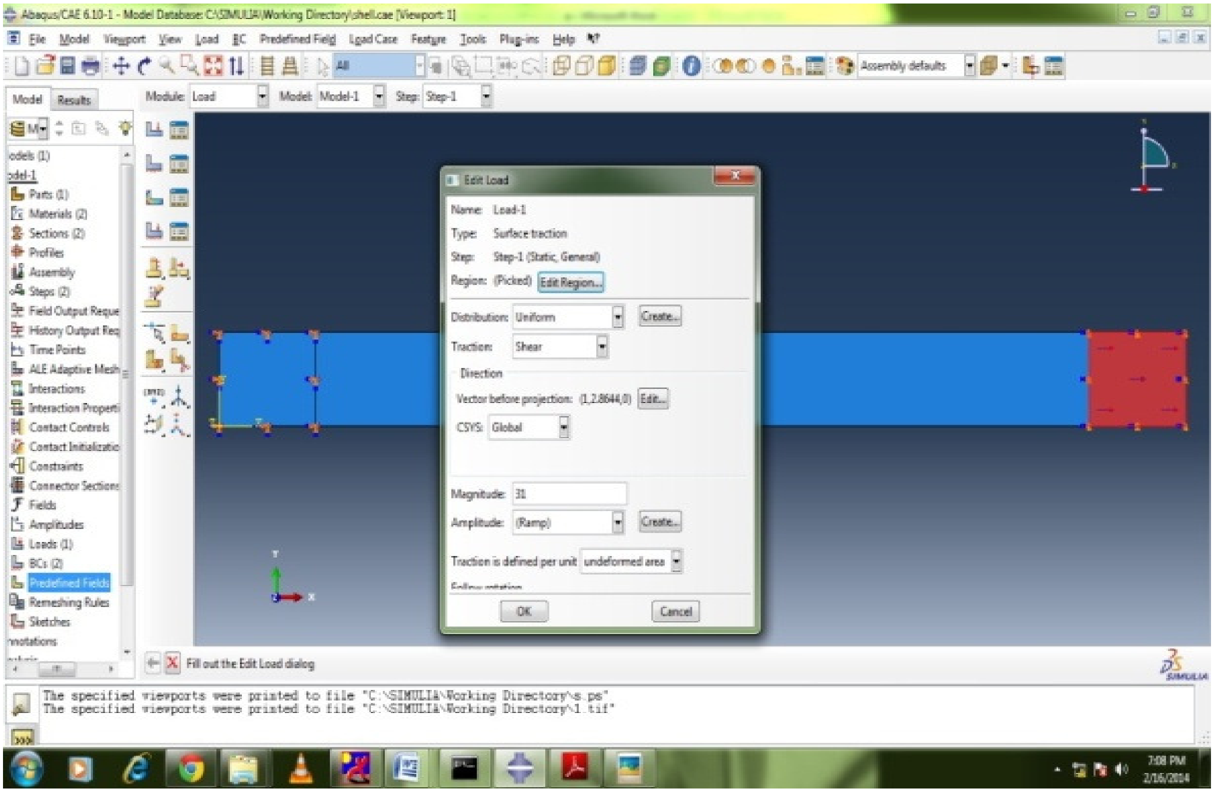

Fig:3 boundary condition and load applied on the specimen

3.4Mesh

Meshing is one of the most important aspects of engineering simulation. With the help of this meshing model is divided into smaller number of elements. If mesh density is low then there are chances that inaccurate result may come. If smaller size elements are there more accurate result will come but it will go to take long time. Care should be taken to optimize the mesh size so as to get reliable result.

For the shell model S4R (linear, four node general purpose shell, reduced integration), for the solid continuum shell model SC8R (linear, eight node hexahedron, reduced integration) elements are used. Abaqus provide hourglass control formulation only for S4R and SC8R elements. The enhanced hourglass formulation is used with composite shell. in shell model total number of node and elements are 366 and 300 respectively. In solid continuum model nodes are 732 and elements are 300.

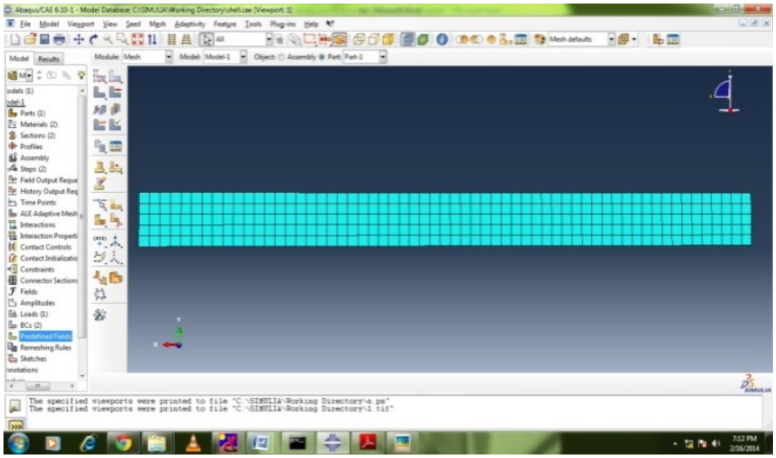

Fig:4 mesh model of specimen

Results and discussion: These material, properties used in this research work extracted from SP.

Carbon fiber-
E_f =220 GPa
v_f=0.2 (Poisson ratio)
V_f=0.5 (volume fraction)

E-glass fiber
E_f=73GPa
v_f=0.25
V_f=0.5

Matrix (Epoxy Vinyl Ester Resins)

E_m =4.5GPa
V_m =0.4(volume fraction)
v_m =0.5 (Poisson ratio)

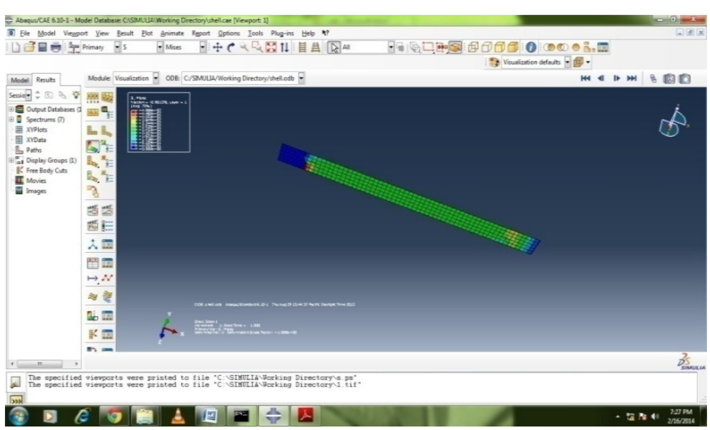

Fig:5 visualization of deformation results of specimen

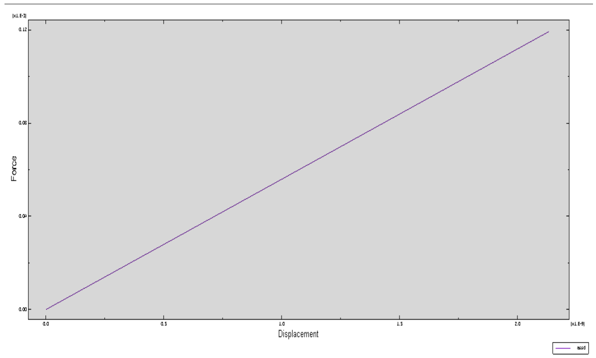

Fig:6 plot of reaction force vs. displacement of shell model

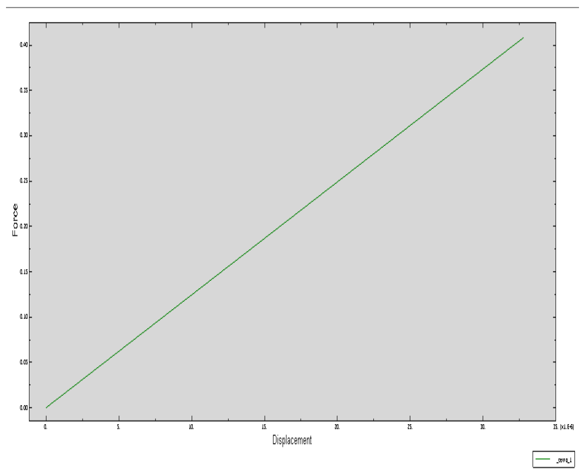

Fig:7 plot of reaction force vs. displacement in semisolid model

Discussion: In Saha's thesis project the results (displacements and reaction force) are greater. So this paper needs to be analyzed and best results have been extracted. The two models (shell, semisolid) are compared in fig:6 and fig:7. the semisolid model gives the best results.

Table-1

Ply no	**Ply orientation for different plies**		
	0° Specimen	45° Specimen	90° specimen
1	45	90	-45
2	90	-45	0
3	-45	0	45
4	0	45	90
5	90	-45	0

6	45	0	-45
7	90	-45	0
8	-45	0	45
9	0	45	90
10	90	-45	0
11	45	0	-45
12	90	-45	0
13	-45	0	45
14	0	45	90
15	90	-45	0

5. NOTATIONS AND ABBREVIATIONS:

A Area
ε Strain
ε_c Strain of composite
ε_f Strain of fiber
ε_m Strain of matrix
η Shear stress component
η_c Shear stress of composite
η_f Shear stress of fiber
η_m Shear stress of matrix
ζ Normal stress component
ζ_c Stress in composite
ζ_f Stress in fiber
ζ_m Stress in matrix
wt Fiber width/thickness ratio
δ_c Elongation of composite
δ_m Elongation of matrix
δ_f Elongation of fiber
Δ_c Deformation of composite
Δ_f Deformation of fiber
Δ_m Deformation of matrix
F Load
G Shear modulus
E_f Fiber Elasticity Modulus
V_f Fiber volume fraction
v_{pf} Fiber Poisson's ratio
E_m Matrix Elasticity Modulus
V_m Matrix volume fraction
v_{pm} Matrix Poisson's ratio
lt Fiber length/thickness ratio

REFERENCES

[1] Adis Avdic and Ujjal Kumar Saha, "Simulating a Tensile Test of Carbon Fiber Composite" Masters Thesis, Department of Technology and Society, SKÖVDE HÖGSKOLAN, Skövde, Sweden, 2011.

[2] . M.Daniel, and O. Ishai, "Engineering Mechanics of Composite Materials", Oxford University Press, New York, 1994.

[3] J. C.Halpin, and S.W.Tsai, "Effects of Environmental Factors on Composite Materials" AFML-TR, 67, page:129, 1969.

[4] Berbinau, C. Soutis*, I.A. Guz; "Compressive Failure of Unidirectional Carbon Fiber Reinforced Plastic (CFRP) Laminates by Fiber Microbuckling", 1999.

[5] David Roylance, "Introduction to Composite Materials", Department of Materials Science and Engineering Massachusetts Institute of Technology Cambridge", MA 02139;2000.

[6] M. Shioya, M. Nakatani, "Compressive Strengths of Single Carbon Fibers and CompositeStrands", Department of Organic and Polymeric Materials, Tokyo Institute of Technology, 2-12-1 O-Okayama, Meguro-ku, Tokyo 152-8552, Japan;2011

[7] R.M.Jones., "Mechanics of Composite Materials", McGraw-Hill, New York, 1975.