Asian Journal of Applied Science and Technology (AJAST)
(Open Access Quarterly International Journal) Volume 1, Issue 9, Pages 142-146, 2017

Damage Assessment on Laminated Composites Under Tensile Test Using Finite Element Method

P.Arul Jose[1], T.Sasi kumar[2], P.Arun Bose[3] and Ashok KB[4]

[1,2,3]*Department of Mechanical Engineering, Lord Jegannath College of Engineering and Technology, Kanyakumari, Tamilnadu, India.*
[4]*Department of Mechanical Engineering, Kottayam Institute of Technology and Science, Kottayam, Kerala, India.*

Article Received: 05 August 2017 *Article Accepted: 21 September 2017* *Article Published: 27 October 2017*

ABSTRACT

The present work gives an efficient tool for design of glass fiber reinforced plastic (GFRP) material using ABAQUS software. To predict the displacement and strength of composite materials, simulation of the tensile test of 16 ply GFRP specimens with different orientations were analyzed. This analysis reveals that the role of ply orientation on the strength of composite materials. Glass fiber with cross ply lamination gives better strength than angle ply lamination.

Keywords: Glass fiber reinforced plastic (GFRP) material, ABAQUS software and Glass fiber.

1. INTRODUCTION

Composite materials are widely used in automotive, aeronautical, naval, construction etc., fields instead for metals [1]. Fiber reinforced polymer matrix are most & valuable among all other materials [2]. Glass fibers used as structural and skin material [3]. Glass fiber reinforced plastic is a strong, lightweight, material [4]. Finite Element Analysis (FEA) is an important tool for numerical stress investigation, with a benefit of being appropriate to solids of irregular geometry that contain various material properties. This type of technique may provide better information on the behaviour of individual tissues [6, 7]. While choose the proper numerical model, part type and degree of discretization are important to attain accurate, time and cost efficient solution.

Finite element analysis has confirmed itself an conventional mathematical analysis with a principal significance is not only aerospace, production engineering and the automotive fields, but in health care too. (8) Stanova et al. generate numerical model in the FEA software ABAQUS/Explicit towards to predict the behavior of the multi-layered mat under tensile loads [9]. The finite element simulation was done on a CFRP tensile specimen by ABAQUS, which solve issues from linear to non linear behaviour [5]. The main aim of this investigation is to predict the ply based strength of GFRP through simulation software package ABAQUS.

2. MATERIAL DEFINITION

Composite materials are principally ready by collecting or binding many distinctive layers of one-way plate or ply such the fiber orientation will vary among the plate. Fig. one represents a one-way laminate or ply created up matrix and fiber with longitudinal and transversal directions. So as to simulate the mechanical behavior of composite specimen, stiffness of every plate is to be calculated. The stiffness of the plate and laminates depends upon numerous factors like basic mechanical properties of the matrix and fiber, quantity or volume fraction of the matrix and fiber within the material, sort and orientation of reinforcement utilized like continuous or discontinuous fiber.

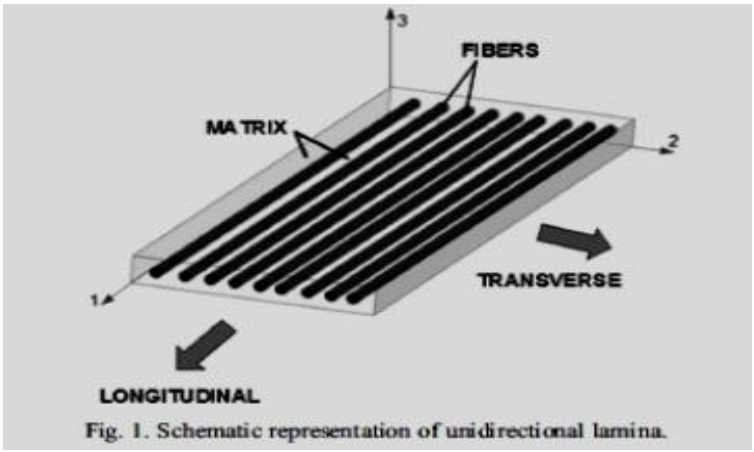

Figure 1

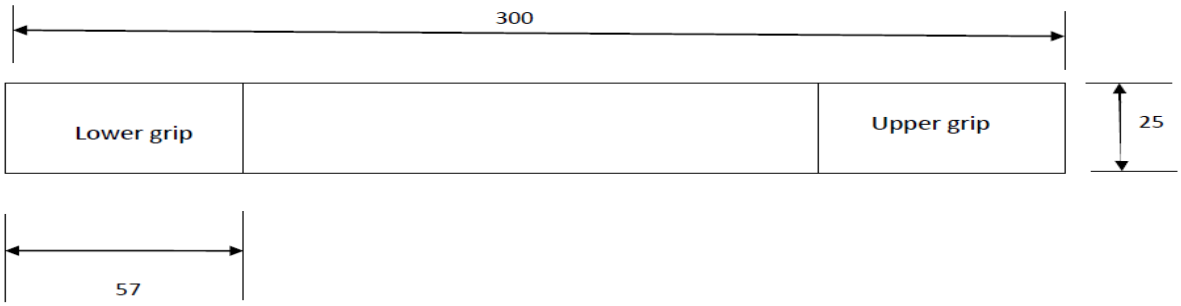

Figure 2: Tensile test specimen geometrical model

The load during this explicit simulation was applied victimization surface grip situation keeping in mind the very fact that displacement control leads to a far additional gradual failure as compared to loading done by applied forces methodology. It is determined that once a straightforward structure, like composite plate begins to fail underneath the action of applied force the structure fails terribly as a result of the load continues to extend thereby decreasing the load carrying capability of the structure. On the opposite hand in displacement controlled loading the load carried by the structure decreases because the structure fails that permits for a slower rate of failure.

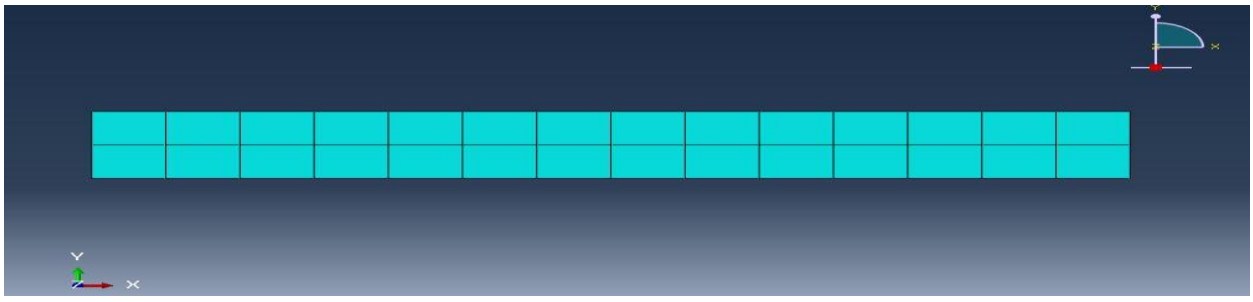

Figure 3: Tensile test specimen (FE model)

Fig. 2 shows geometric model of the rectangular specimen. In order to collect stress distribution result, S4 shell model have been prepared.Fig.3 represents FE model of test specimen.

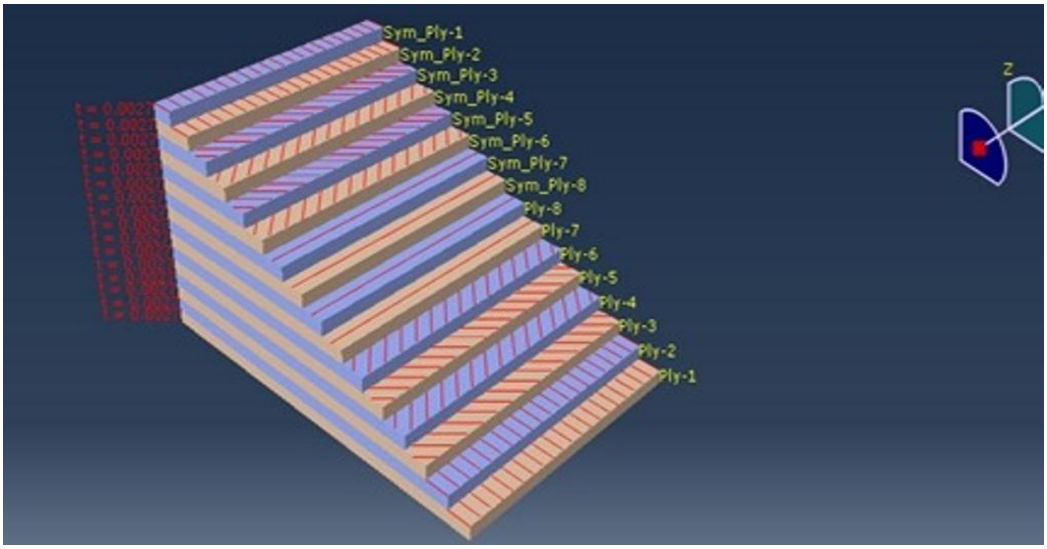

Figure 4: Ply sequence and orientation of the GFRP laminates

Table 1: Ply sequence, thickness and orientations

PLY NO	THICKNESS	ORIENTATION
1	0.0027	0
2	0.0027	0
3	0.0027	-45
4	0.0027	+45
5	0.0027	-45
6	0.0027	+45
7	0.0027	90
8	0.0027	90
9	0.0027	90
10	0.0027	90
11	0.0027	+45
12	0.0027	-45
13	0.0027	+45
14	0.0027	-45
15	0.0027	0
16	0.0027	0

Table 2: Mechanical properties of constituent

Constituent	E (Gpa)	υ	V_f
E-glass fiber	73	0.25	0.5
Matrix (epoxy vinyl ester resin)	4.5	0.4	0.5

Table 3: Engineering properties of Glass fiber/epoxy

Property	E-glass epoxy
E_{11} (GPa)	38.7
$E_{22=}E_{33}$ (GPa)	8.4
$G_{12=}G_{13}$ (GPa)	3
υ_{12}	0.314

In this research work glass/epoxy with 16 ply has been investigated. Table 2 & 3 shows engineering and mechanical properties of glass fiber epoxy ply. Fig. 1 shows 16 ply GFRP and their orientation. Every ply was assigned its region, thickness and orientation. The gauss integration rule was most liked during this simulation.

3. RESULTS & DISCUSSION

This topic is based on the analysis of GFRP test specimen using shell mode. The results evidenced the displacement occurred at free end while maximum reaction force occurred. Table 4 shows the average values of cross ply parameter.

Table 4

Parameter	Results
Reaction force(KN)	**8.3**
Displacement(mm)	**2.1**
Strain(major)%	**4.1**
Strain(minor)%	**-1.02**

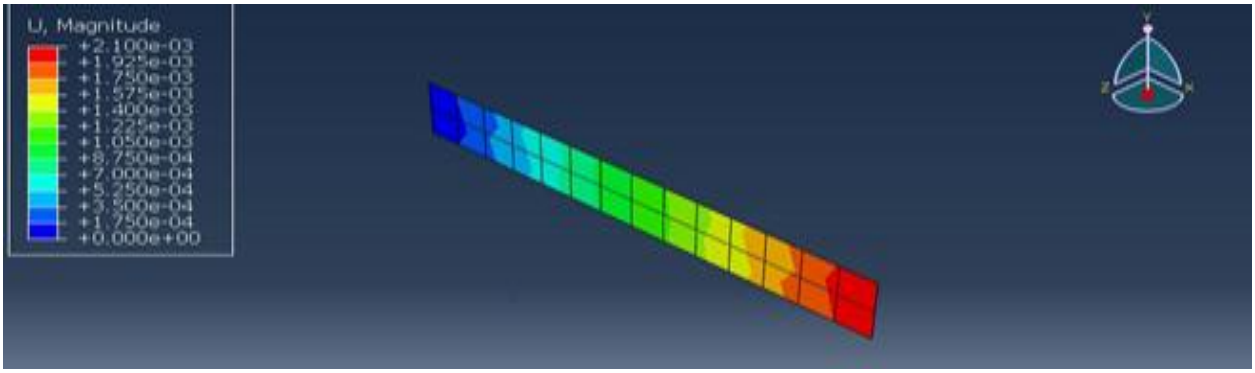

Figure 5: Displacement on GFRP test specimen shell model

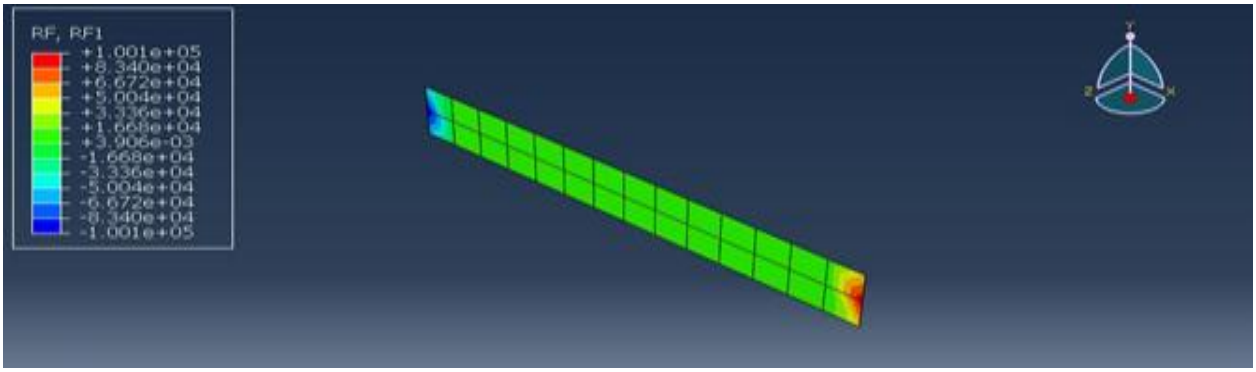

Figure 6: Reaction force on GFRP test specimen shell model

The different ply orientations were simulated to predict finest in the sort of strength and stiffness as shown in fig 7. From the results it is noticed that when considered displacement cross ply deflection is lesser than the angle ply.

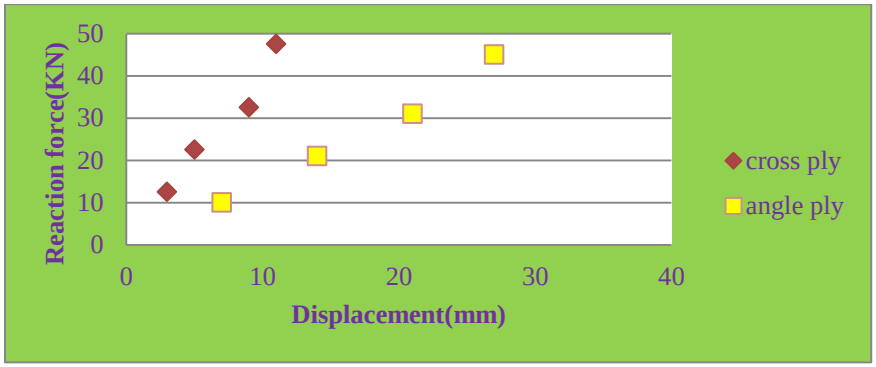

Figure 7: Reaction force vs displacement

4. CONCLUSION

The load applied in the form surface traction is applied to the free end. The displacement is maximum at free end but Zero at other side. This research revealed the effect of orientation and following conclusions were found: (i) Cross ply laminates were found to be strength than angle ply lamination (ii) And when displacement is concerned, the angle ply deflection is more than cross ply lamina.

REFERENCES

[1] Rafel CS, Recardo la, Antonio FA, Marco A, Mechanical characterization of Glass/epoxy composite material with nanoclay. In proceedings of COBEM 2007.

[2] Mouritz AP, Gallagher J, Goodwin AA, Fexural strength and inter-laminar shear strength of stitched GRP laminate following repeated impacts. Composite science and technology, 1997, 57:509-22.

[3] Why aircraft fail, FINday, S.J & Harrison, N.D, Elsevier, Nov 2002, Material today, vol:15, pp:18-25.

[4] Sunit Babu L, H.K.Shivanand, Impact analysis of laminated composite on glass fiber and carbon fiber, International journal of Engineering and technology & Advanced Engineering, Vo:4: issue 6, June 2014.

[5] Avdic. D, Sha U.K, Simulating a tensile test of a carbon fiber composite test specimen in Abaqus, M.Tech, Skovde Hogg Kolan, Skovde, Sweden.

[6] Geng JP, Tan KB, Liu GR (2001) Application of finite element analysis in implant dentistry: a review of the literature. J Prosthet Dent 85: 585-598.

[7] Kazuo Tanne, Mamoru Sakuda, Charles J (1987) Three dimensional finite analyses for stress distribution in the periodontal tissue by orthodontic forces. Am. J. Orthod. Dentofacial Orthop. 92: 499-505.

[8] Basic Concepts of Finite Element Analysis and its Applications in Dentistry: An Overview Mohammed SD and Desai H Oral Hygiene & Health 2014.

[9] Anıl, A. S., (2011) Investigation of Fatigue Life of Axial Loaded Wire Rope Strands In Computer Environment, Master of Science Thesis, Istanbul Technical University.