International Journal of Applied Engineering Research ISSN 0973-4562 Volume 11, Number 1 (2016) pp 380-384
© Research India Publications. http://www.ripublication.com

Analysis and Characterization of Tensile Property of the Composite Specimen using ANSYS

K.Krishnamoorthy[1]
[1]Research Scholar, Department of Mechanical Engineering,
Lord Jegannath College of Engineering & Technology, Kanyakumari, India.
Email: sangeethakrishnamoorthy1972@gmail.com

T.Sasikumar[2]
[2]Professor, Department of Mechanical Engineering,
Lord Jegannath College of Engineering & Technology, Kanyakumari, India.

Abstract
The objective of the this research paper is to predict the deformation length of glass/epoxy composite laminates using Finite Element Analysis (FEA) and compare with the Universal Testing Machine (UTM) deformation results through the tensile load. The laminate was made by six layered glass fiber (in woven mat form) with epoxy as the binding medium by hand lay-up technique, cured at room temperature at the pressure of 30 kg/cm^2. Tensile specimens with ASTM D3039 standarddimensions were cut from the laminate. These specimens were subjected to uni-axial tension under the 10 ton capacity UTM. The specimen's FEA models were created using ANSYS 12 and the deformation length data were obtained. The actual deformation results were compared with the predicted values obtained from the FEA. The results were ensures that the FEA results has well agreed with the original deformation length with in the acceptable error margin.

Keywords: Glass/Epoxy Composite laminates,FEA, Tensile Testing

Introduction
Dramatic development of new structural materials has followed the way of substituting metals for composite in various industries, such as aviation, ship building chemical-petroleum, civil engineering etc. Fiber reinforced composite materials have been increasingly used as structural members in many structures such as airplane, which in flight condition undergoes temperature as low as -60°C or in acryogenic tank which may be exposed to temperature below -150°C. The advantages of these materials arederived from their high strength, stiffness anddamping together with low specific weight. On the other hand, composite materials have the potential of reducing costs in construction, operation and development while improving structural reliability and enhancing safety. Because of these unique specifications, they are widely used in high technology structural applications, such as aeronautic and aerospace.

Composite is a mixture of two or more constituents/materials (or phases) with different physical/chemical properties at the macroscopic or microscopic scale[1]. In general composites have two or more constituents, fiber and matrix. Composites are classified by the geometry of the reinforcement: particulate, flake, and fibers or by the type of matrix: polymer, metal, ceramic, and carbon. The basic idea of the composite is to optimize material properties of the composite, i.e., the properties of the matrix are to be improved by incorporating the reinforcement phase. Fibers are the principal load-carrying constituents while the surrounding matrix helps to keep them in desired location and orientation and also act as a load transfer medium between them.

Experimental Procedure
A. Specimen Fabrication
GFRP composite laminate of dimensions 250 × 400 mm are fabricated using hand layup method. Six layers of bi-directional glass fiber mat along with LY556 epoxy are employed forth purpose of fabrication of the laminates. The 14 tensile specimens of size 250×25×2.5 mm were cut from the laminate in accordance with ASTMD 3039 standard. GFRP composite tabs are provided as per ASTM standard on both ends. Diamond cutting was employed to avoid the machining defects and to ensure good surface finish along the cutting edges. Tensile test specimens werefabricatedin accordance to ASTM D 3039 standards as shown in figure1&2.

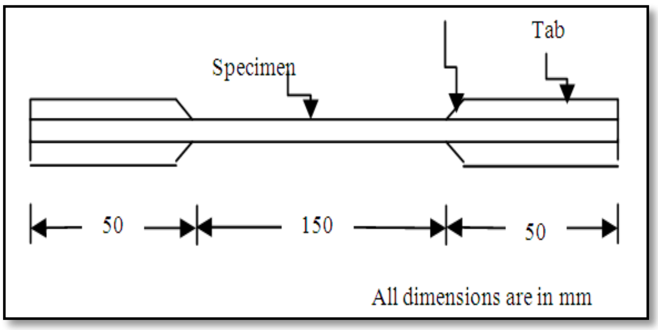

Figure 1: Specimen's Specification

Figure 2: Sample Specimens

B. Tensile testing procedure

The tensile specimens obtained from the laminates are subjected to uni-axial tension using DAKuniversal testing machine (Fig 3). Fourteen specimens were tested. The crosshead speed was maintained at 1.5 mm/min throughout the testing process.

Figure 3: UTM with Specimen

Tensile properties of bidirectional glass/epoxy composite are determined through testing. The testing procedure adheres to the ASTM forth Tensile Properties of Epoxy Matrix Composite Materials, D3039. Figure4 shows the prepared tensile test specimens with strain gages and wiring to obtain the strain values during the testing process.

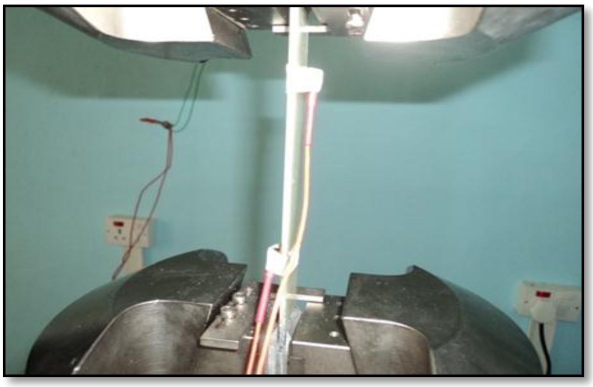

Figure 4: Strain Gauges

Finite Element Analysis Using Ansys

ANSYS is general purpose finite element analysis (FEA) software package which is engineering simulation software (computer- aided engineering, or CAE) that utilizes Finite element Analysis. It is a numerical method in which a complex system is divided into very small pieces called elements. The software uses equations that generate the behavior of these elements and solves them all.

A. Parameters chose as input to FEA

The engineering data and materials properties are required to analyze the prepared composite specimens so that we can create one model and use it for the analysis purpose. In ANSYS software the parameters for analysis purpose chosen were based on the matrix material and the reinforcement. Static structure and advanced composite tools were also selected for the purpose. The values of poison's ratio (υ) and young modulus of elasticity (E) were determined experimentally(Table 1) and same were used in finite element analysis. With the advancement of computers, finite element analysis [2]has become one of the most important tools available to an engineer for design analysis. The finite element method is one of the most general procedures for solving complex analysis problems. For performing finite element analysis the material was considered to be isotropic in nature and the boundary condition and load conditions applied were similar to the experimental condition. The element type used for ANSYS work was solid 8-node 45[3]and the values of young modulus and Poisson ratios were taken from experimental results. The figure 5 and 6 shows the meshing details of the FEA models. The figure 7 and 8 shows the deformational FEA model for the specimen number 2 in two dimensional and three dimensional views respectively.

Table 1: Experimental Values

Specimen	Failure Load in KN	Young Modulus (Gap)	Poisson's Ratio(υ)
SP-1	8.63	11.13	0.24
SP-2	11.51	12.12	0.24
SP-3	10.74	11.3	0.24
SP-4	9.64	12.44	0.24
SP-5	10.85	12.07	0.24
SP-6	12.51	12.2	0.24
SP-7	12.91	12.01	0.24
SP-8	12.54	11.67	0.24
SP-9	10.53	11.08	0.24
SP-10	12.40	11.02	0.24
SP-11	10.47	9.52	0.24
SP-12	8.90	9.89	0.24
SP-13	11.55	11.55	0.24
SP-14	9.27	10.91	0.24

International Journal of Applied Engineering Research ISSN 0973-4562 Volume 11, Number 1 (2016) pp 380-384

Figure 5: Details of Meshing(3D)

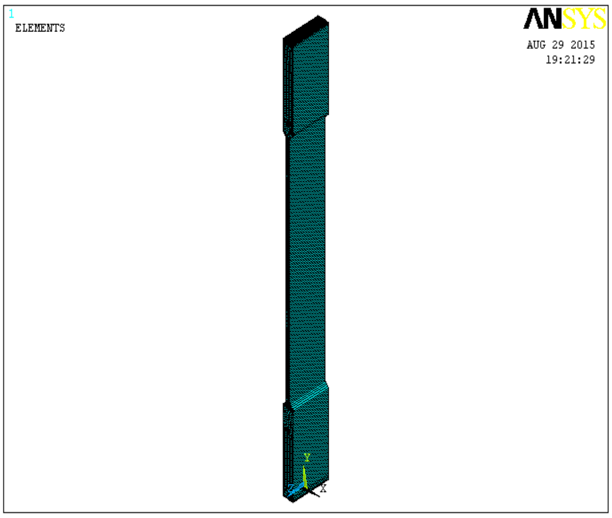

Figure 6: Details of Meshing(3D)

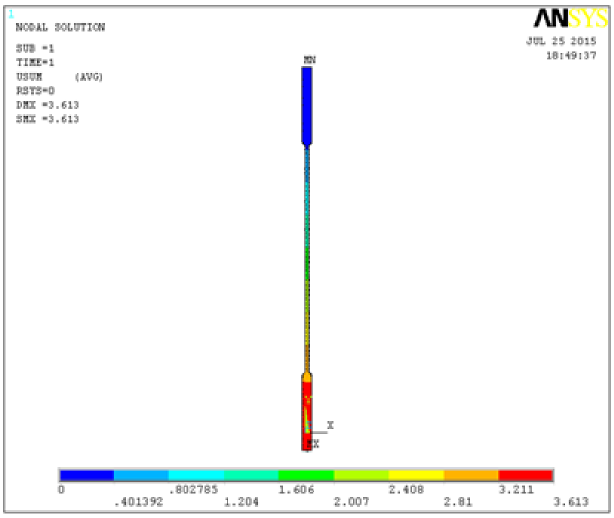

Figure 7: Nodal solution of tensile test(2D)

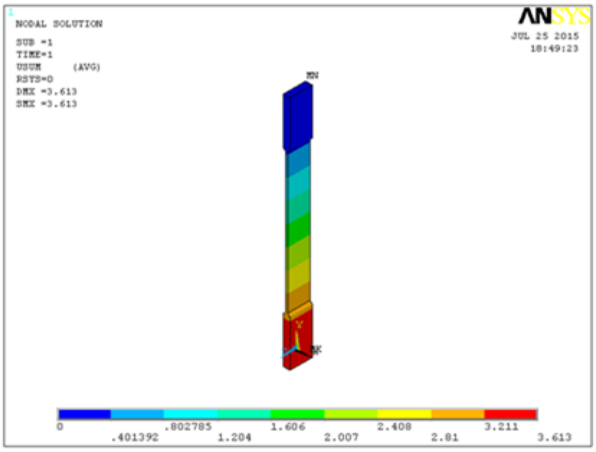

Figure 8: Nodal solution of tensile test(3D)

Result and Discussion

Many researchers have analytically and experimentally investigated [4-6] the mechanical properties (tensile, flexural, toughness, fatigue etc.) of FRP composites and other used finite element analysis [7-10] to predict the behavior of FRP and their mechanical properties. Ghassemieh [11, 12] developed a micro-mechanical model to understand the behavior of fiber and particulate reinforced polymeric composites. This model was used to simulate stress distribution and to identify the maximum stress concentrations locations. The interfacial stresses evaluated by model were compared with the well-known shear lag and modified shear lag models. Horsta *et al.* [13] developed a finite element model to predict the interfacial tensile and shear stress and validated experimentally. Perfect bonding between fiber and matrix was considered. Caporale *et al.* [14] examined the behavior of unidirectional fiber-reinforced composites with imperfect interfacial bonding with the aid of finite element method.

However a few efforts were made to relate the experimental results of mechanical properties of FRP with the finite element analysis results while considering the isotropic behavior of composites [15 &16]. The main advantage of using finite element analysis is to generate the quantitative data about the failure morphology of the composites and to understand the deviation of results from the experimental results. In present work an attempt is made to relate the experimental and numerical results of tensile by considering the isotropic behavior of composites. The experimental results were reproduced by applying the same boundary and loading conditions in ANSYS 12 (non-commercial version) and comparisons were made as shown Table 2. From this table SP-13 has maximum error percentage and the SP-9 has minimum error percentage. But all specimens has error percentage with in the acceptable limit. Results revealed that tensile results for numerical analysis are better than experimental results as shown in figure 9. This deviation of results occurred due to manufacturing defects of composites like blow holes, porosity etc. Further in analysis part isotropic behavior was considered, but it is not possible to achieve isotropic behavior practically using random oriented fibers due to stress concentrations at fiber ends.

Table 2: Comparison of Actual Elongation with Predicted Elongation

Specimen	Actual Elongation in mm from Tensile Test	Predicted Elongation in mm from ANSYS	% of Error
SP-1	3.10	2.97	4.19
SP-2	3.83	3.61	5.74
SP-3	3.75	3.50	6.67
SP-4	3.10	2.95	4.84
SP-5	3.61	3.22	10.80
SP-6	4.10	3.68	10.24
SP-7	4.28	3.89	9.11
SP-8	4.30	3.96	7.91
SP-9	3.83	3.67	4.18
SP-10	4.52	4.06	10.18
SP-11	4.40	4.07	7.50
SP-12	3.63	3.32	8.54
SP-13	3.99	3.33	16.54
SP-14	3.41	3.21	5.87

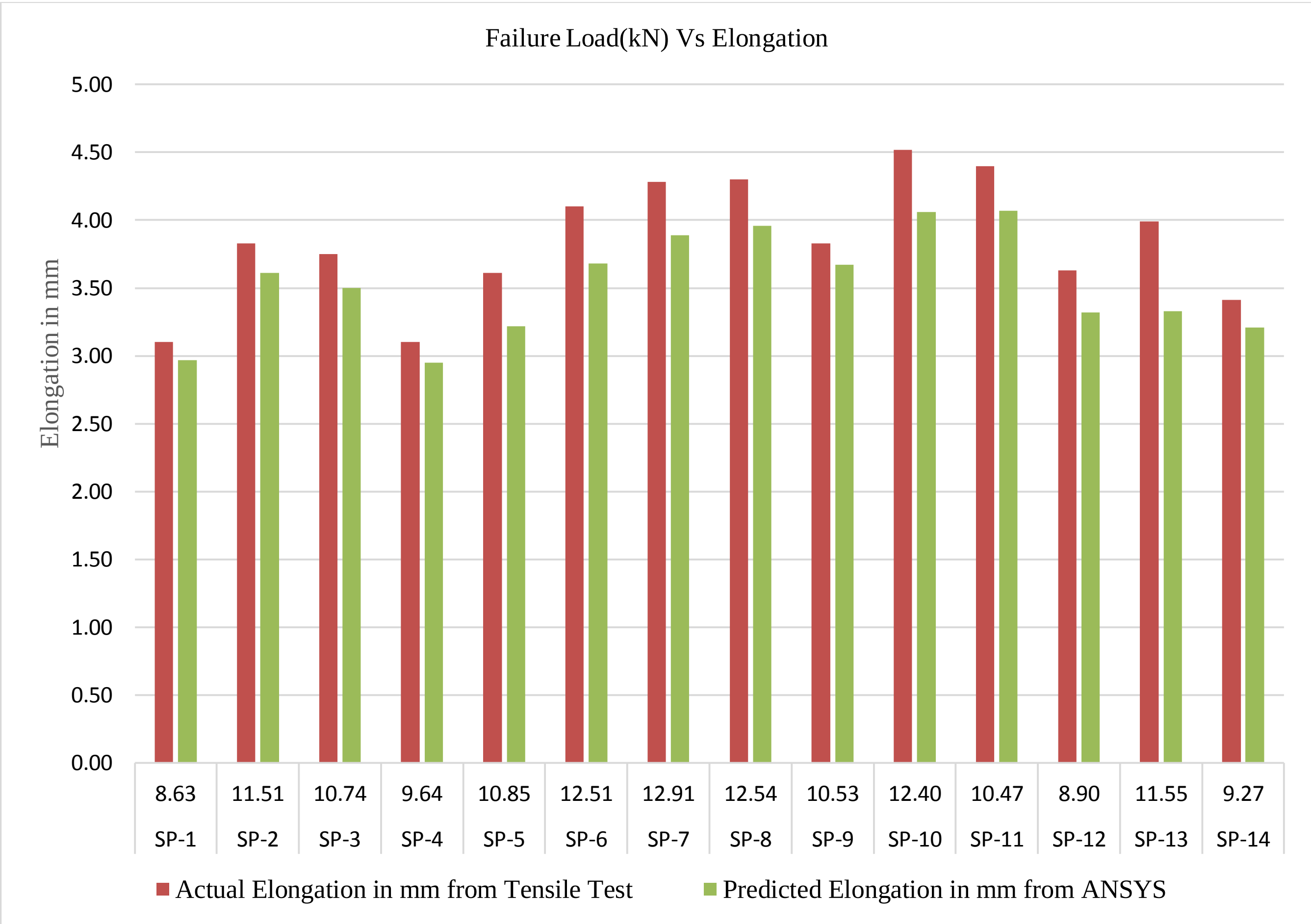

Figure 9: Comparison of Actual and Predicted Elongation

Conclusions

The characterization of mechanical properties of the composite reinforced with
E-glass fiber, reinforced epoxy composites is reported and the results are summarized. The results obtained experimentally were compared with the simulation in ANSYS software. The experimental values and ANSYS values match reasonably validating the experimental results.

References

[1] Kaw A.K.; *Mechanics of composite materials*, Chapter 1, CRC Press: Taylor & Francis Group, USA, 2006, 2nd Edition. ISBN: 0-8493-1343-0.

[2] Sun, W.; Lin, F. *Journal of Thermoplastic Composite Materials*, Computer Modeling and FEA Simulation for Composite Single Fiber Pull-Out, 2001, 14, 327. DOI: 10.1106/YKDM-PX8K-NF6Q-L7FK

[3] Gilchrist, M.D.; Kinloch, A.J. Composites Science and Technology, Mechanical performance of carbon-fibre- and glass-fibre-reinforced epoxy I-beams, 1996, 56, 37. DOI: 10.1016/0266-3538(95)00126-3

[4] Ascione, F.; Feo, L.; Maceri, F. Composites Part B: Engineering. An experimental investigation on the bearing failure load of glass fibre/epoxy laminates, 2009, 40, 97. DOI: 10.1016/j.compositesb. 2008.11.005

[5] Shokrieh, M.M.; Omidi, M.J. Composite Structures, 2009, 88, 595. DOI: 10.1016/j.compstruct.2008.06. 012

[6] Haider, A.Z.; Zhao, X.L.; Riadh, A.M. *Procedia Engineering*, 2011, 10, 2453. DOI: 10.1016/j.proeng. 2011.04.404

[7] Segurado, J.; LLorca J. *Mechanics of materials*, Computational micromechanics of composites:The effect of particle spatial distribution, 2006, 38, 873. DOI: 10.1016/j.mechmat.2005.06.026

[8] Zhang, Y.X.; Yang, C.H. *Composite Structures*, Recent developments in finite element analysis for laminated composite plates, 2009, 88, 147. DOI: 10.1016/j.compstruct.2008.02.014

[9] Kabir, M.R.; Lutz, W.; Zhu, K.; Schmauder, S. *Computational Materials Science*,Fatigue modeling of short fiber reinforced composites with ductile matrix under cyclic loading, 2006, 36, 361. DOI: 10.1016/j. commatsci.2005.09.004

[10] Goh, K. L; Aspden, R. M; Hukin, DWL. *Composite Science and Technology*, 2004, 64,1091. DOI: 10.1016/j.compscitech.2003.11.003

[11] Houshyar, S.; Shanks, R.A.; Hodzic, A. *Express Polymer Letters*, Modelling of polypropylene fibre-matrix composites using finite element analysis, 2009, 3, 2. DOI: 10.3144/expresspolymlett.2009.2

[12] Ghassemieh, E.; Nassehi, V. *Polymer Composites*, Prediction of failure and fracture mechanisms of polymeric composites using finite element analysis. Part 2: Fiber reinforced composites, 2001, 22, 542. DOI: 10.1002/pc.10558

[13] Ghassemieh, E.; Nassehi, V. *Advances in Polymer Technology*, Stiffness analysis of polymeric composites using the finite element method, 2001, 20, 42. DOI:10.1002/1098-2329(200121)20:1<42::AID-DV1003>3.0.CO;2- F

[14] Horsta, J. J.; Salienkob, N. V.; Spoormaker, J. L. *Composites Part A*, 1998, 29, 525. DOI: 10.1016/ S1359-835X(97)00125-5

[15] Caporale, A.; Luciano, R.; Sacco, E. *Computers and Structures*, Micromechanical analysis of interfacial debonding in unidirectional fiber-reinforced composites, 2006, 84, 2200. DOI: 10.1016/j. compstruc. 2006.08.023

[16] Naik, K.N. *Woven Fabric Composites*, Introduction, Technomic Publishing Company Inc., Pennsylvania (1994).