LOAD-DISPLACEMENT BEHAVIOR OF CARBON-EPOXY LAMINATED COMPOSITES DURING DELAMINATION-A NUMERICAL STUDY BY FINITE ELEMENT METHOD

M.Kishore[#1], Dr.Ch.SrinivasaRao[*2], Ch.Sekhar[#3]

[#]*M.kishore-Ch.sekhar (Associate professor) Mechanical Engineering, M.kishore-Ch.sekhar*
Jntu Kakinada University
Narasaraopeta, A.P. INDIA
[1]kishore.mightymech@gmail.com
[3]sekhar-mech348@yahoo.co.in
[2]*Dr.Ch.SrinivasRao, Professor*
Guntur, A.P. INDIA
[2]chepuri-srao@yahoo.co.in

Abstract: **Laminated composites are assemblies of layers of fibrous composite materials joined to provide various engineering properties like in-plane stiffness, bending stiffness, strength, and coefficient of thermal expansion. The individual layers consist of high-modulus, high-strength fibers in a polymeric, metallic, or ceramic matrix material.**

The determination of load bearing capability of laminated composites is vital in the design and fabrication of structural and other engineering composites. Delamination in laminated composites is one type of failure that has long been the centre of materials research. The failure is known due to separation at an interphase region, caused by manufacturing defects, object impacts, or high stress concentrations from geometrical discontinuity. The load-displacement characteristic of a material is the finger print of all the mechanical properties and estimation of such properties needed extensive experimental facilities.

Finite Element Method (FEM) is one of the numerical techniques used to study the load-displacement behavior of engineering materials with less expenditure and the results obtained will be in good agreement with experimental results.

In the present work, FEM approach using ANSYS software is employed to simulate the delamination process in Carbon/Epoxy laminated composite and to extract the Load-displacement characteristics during the failure of the laminate by delamination process. The laminated composite is assumed to be a four layered Double Cantilever Beam (DCB) with the composite material properties given as input. Parametric study is carried out for a range of displacement of the beam and displacement conditions are varied from 10mm to 18mm with a variation of 2mm.

Keywords: Load-displacement characteristics, laminated composites, Delamination, Finite Element Method.

1. INTRODUCTION

Delamination is a common kind of failure observed in various composites. Various experimental techniques and numerical methods have been used to study the behavior of delamination in composite materials. Finite element (FE) models have been utilized in the past to study the stress distribution and the strain energy release rate in cracked composite laminates subject to Mode I, Mode II, and mixed-mode loadings. Vander Zande and Grootenboer [1] have performed finite element modeling of delamination to study the interface cracks and compared their model of an interface crack in an isotropic bi-material with Comninou's results [2,3]. They model contact zones at the crack tips with special contact elements, which do not permit wrinkling of the two faces and transmit the compressive stresses in the contact zone of the crack. Hwu et al. [4] use a FE model for delamination analysis of glass/epoxy multidirectional specimens, with various lay ups and varying the layers between 3 to 6.

Vol.2, No.1 ISSN Number (online): 2454-9614

Proceedings of International Conference on Recent Trends in Mechanical Engineering-2K15(NECICRTME-2K15), 20th – 21st November,2015

An analytical crack tip element based on classical plate theory was introduced by Davidson et al. [5]. The virtual crack closure technique was used by Davidson [6] and Polaha et al. [7].

The advancement of the finite element methods has provided a robust and flexible tool to solve the nonlinear problems [8-10]. For example, a global-local FE analysis used 3D FE models with either layered shell or solid brick elements in the fracture critical zones with the boundary conditions obtained from the global analysis [11].

2. PROBLEM STATEMNT

The specimen of the laminated composite specimen is assumed with 4 layers with a length of 100mm, height .5mm, width 20mm and a debonding length 'a' of 30mm in delaminated layers. The specimen is fixed at one end and displacement loading is applied at the other end. Necessary orthotropic material properties are assumed.

3. FINITE ELEMENT MODELLING

The length of the specimen is aligned along x-axis, height along y-axis and width is aligned along z-axis. Four layers are used for analysis and contact layers are used in intermediate layers for bonding. A debonding length of 30mm is provided in delaminated layer, and the displacement is varied from 10 to 18 mm in y-direction. The layered composite with specifications is as shown in figure Figure-1.

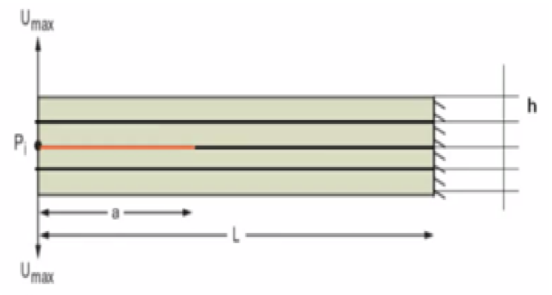

Fig-1: Four layered laminated composite

3.1 Meshing:

The finite element mesh is generated using a 3-D 8-noded solid element OLID185. It is a is a higher order element that exhibits quadratic displacement behaviour. The element is defined by 8 nodes having three degrees of freedom per node: translations in the nodal x, y, and z directions.
The FE mesh of the model is shown in figure-2

3.2 Loading and Boundary conditions

The structural displacement load is applied in the y- direction, with displacement of 10mm is given on the front surface of the elemental structure. One side of rectangular surface is fixed with all degrees of freedom to be zero and other end forces are applied to rip apart the areas. This kind of loading condition is said to be DCB loading. In this case Displacements are varied from 10mm to 18mm with a variation of 2mm.

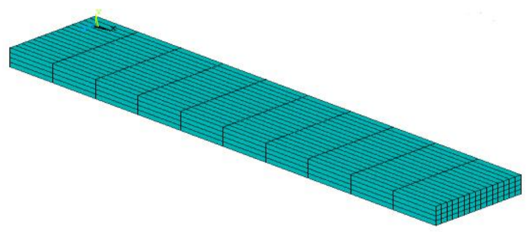

Fig-2: Finite Element Mesh

3.3 Materials Properties:

The following orthotropic properties of Carbon Epoxy materials are used in the present analysis. E_1 =126Gpa, E_2=9.5Gpa, E_3=9.5Gpa, V_{12}=0.263, V_{23}=0.263, V_{13}=0.27, G_{12}=1.07Gpa, G_{23}= 0.8063Gpa, G_{13} = 1.07Gpa

4.0 RESULTS AND DISCUSSION

Numerical results are obtained for extracting load values with variation of displacement in Y-direction. The Reaction forces obtained in Y-direction are the load values of the Delaminated Composite for a given value of displacement. The results also include the stresses and strains developed in the composite.

4.1.1 Displacement Vs .Load

The displacement is given to the top layer of the laminated composite in Y-direction. For the given input displacement load, the output reaction force has been obtained by simulation. The displacement is changed from 10 mm to 18 mm in multiples of 2 mm. The load Vs. displacement curve is shown in figure-3. For the given debonding length of 30 mm and the displacements of 10 mm, 12 mm up to 18 mm the deformed shapes of the laminated composite are shown in figures-,4,5,6 and 7. The stresses developed at the maximum displacement of 18 mm are shown in figure-8.

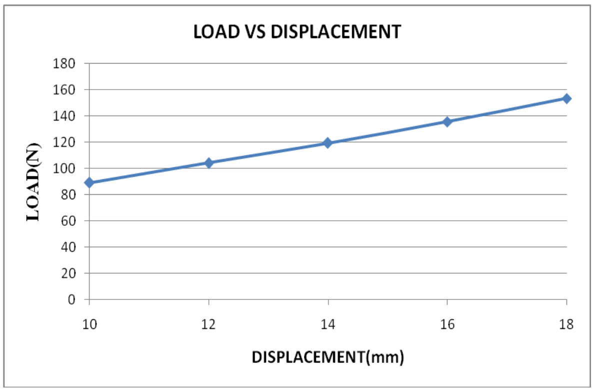

Fig-3: Load-Displacement Curve

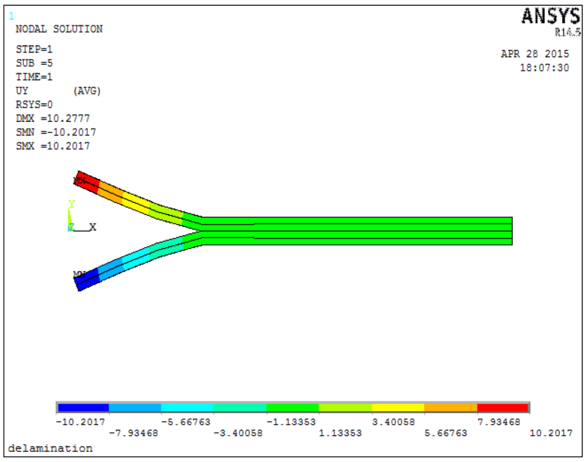

Fig- 4: Y-component of displacement (10mm)

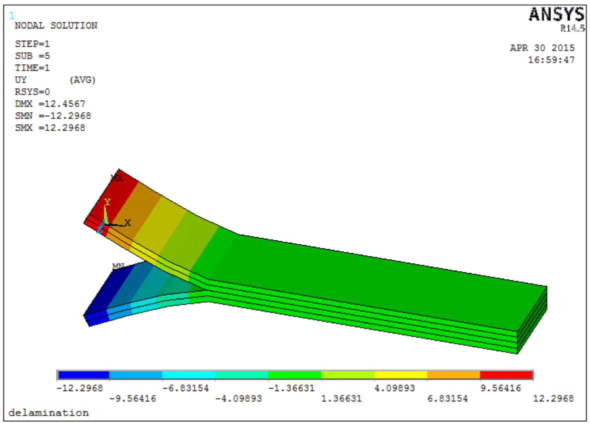

Fig-5: Y-component of displacement (12mm)

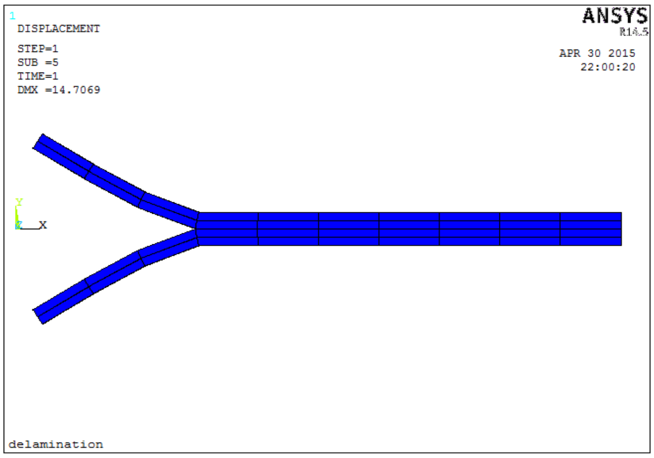

Fig-6: Y-component of displacement (14mm)

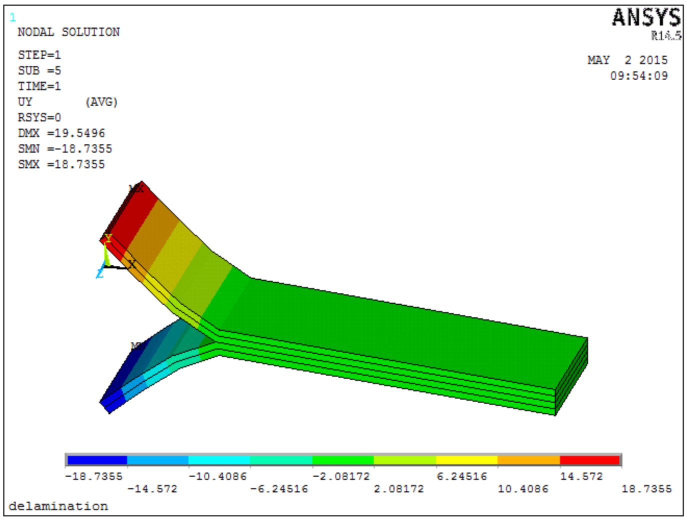

Fig-7: Y-component of displacement (18mm)

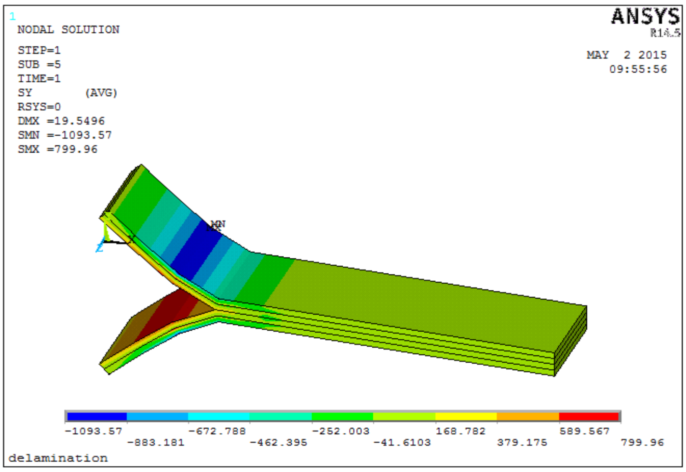

Fig-8: Y-component of stress with displacement 18mm

4.1.2 Normal stresses and strains:

The variation of normal stresses in Y–direction with increase in displacement is as shown in Figures-9. It is found that the stresses are increasing uniformly with displacement and is maximum at 18mm. Similarly, the variation of Von mises stresses with displacement is shown in figure-10. The variation of von mises stresses is considerably more between 14 to 18 mm.

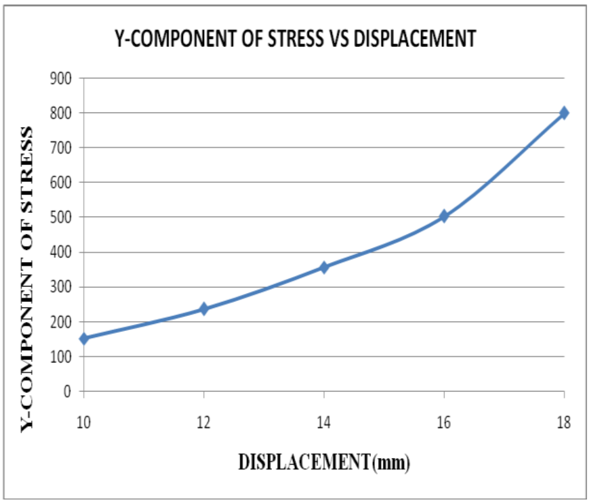

Fig-9: Variation of Y-component of stress to Displacement

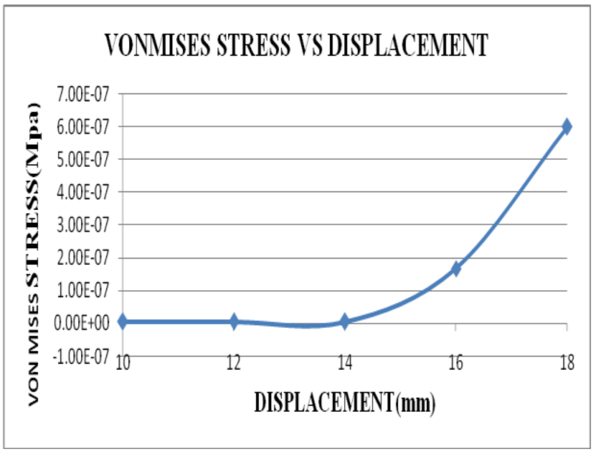

Fig-10: Variation of Von Mises stress to Displacement

The variation of normal strains with increase in displacement is as shown in Figures-11. As the Displacement increases, the y-component of total mechanical strain also increases.

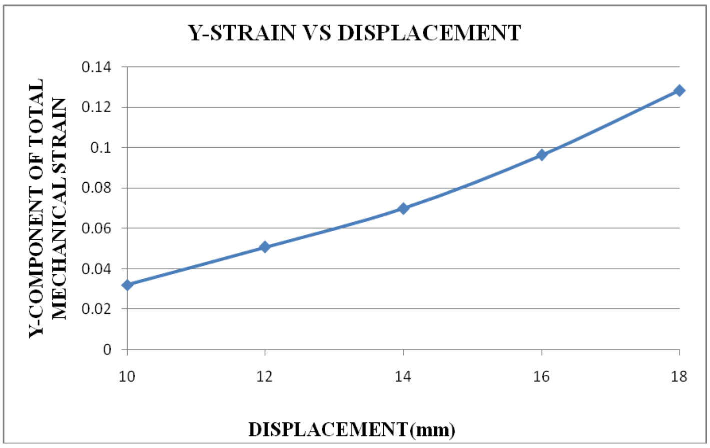

Fig-11: Variation of Y-component of total mechanical strain to Displacement

5. CONCLUSION

Study of Load-Displacement characteristics of a laminated composite is vital in the design and fabrication of structural and engineering components. Therefore, an attempt has been made to successfully simulate the delamination process of Carbon/Epoxy laminated composite. The load –displacement behavior is extracted by giving displacement as the input to the top two layers of the composite. The output reaction obtained by simulation process is the load characteristic and thus the load-displacement displacement characteristics are obtained successfully. The stresses and strains developed in four layered Carbon epoxy laminated composite are also studied in detail. The results are in good agreement with theoretical results.

REFERENCES

[1] Van der Zande HD, Grootenboer HJ. A finite element approach to interface cracks. J Appl Mech 1986;53:573–8.
[2] Comninou M. The interface crack. J Appl Mech 1977;44:631–6.
[3] Comninou M. The interface crack in a shear field. J Appl Mech 1978;45:287–90.
[4] Hwu C, Kao CJ, Chiang LE. Delamination fracture criteria for composite laminates. J Compos Mater 1995;29(15):1962–87.
[5] Davidson BD, Hu H, Schapery RA. An analytical crack-tip element for layered elastic structures. J Appl Mech 1995;62:243–53.
[6] Davidson BD. An analytical investigation of delamination front curvature in double cantilever beam specimens. J Compos Mater 1990;24:1124–37.
[7] Polaha JJ, Davidson BD, Hudson RC, Pieracci A. Effect of mode ratio, ply orientation and precracking on the delaminated toughness of a

laminated Composite. J Reinf Plast Compos 1996;15:141–72.

[8] Crisfield MA. Non-linear finite element analysis of solids and structures. 1997.

[9] Mi Y, Crisfield MA, Davies GAO, Hellweg HB. Progressive delamination using interface elements.*Journal of Composite Materials* 1998; 32: 1247-1272.

[10] Shen F, Lee KH, Tay TE. Modelling delamination growth in laminated composites. *Comp Sci Tech*2001; 61: 1239-1251.

[11] Borg R, Nilsson L, Simonsson K. Simulation of delamination in fibre composites with a discrete cohesive failure model. *Comp Sci Tech* 2001; 61: 667-677.

[12] ANSYS 14.5 User Manual