An Investigation of Failure and Damage in Unidirectional Composite under Static Loading

[1]Mahesh Korhale, [2]Dr. Sajal Roy, [3]Dr. Maruti Khot
[1]MTech student, [2]Scientist, [3]Professor
[1,3]Department of Mechanical Engineering, Walchand College of Engineering, Sangli, India
[2]R& DE Engineers (DRDO), Pune, India

Abstract: Composite materials shows different type of damages such as fiber breakage, matrix cracking, fiber matrix debond and delamination. Over the decades, mechanics of these damages are studying in two different levels, namely micro mechanics and macro mechanics. Fiber breakage, matrix cracking and fiber matrix debond are studying in micro mechanics whereas delamination is studying in macro levels. The paper presents micro mechanics modeling of damages and its evolution using Abaqus Finite Element Analysis (FEA) software. In doing so cohesive zone modeling approach is used. Three different kind of damages such as fiber breakage, matrix cracking and fiber matrix interface debond is studied in a representative volume element (RVE) which is also known as unit cell model. The present study gives a clear idea that fiber fails first, followed by fiber-matrix interface and finally matrix cracking.

Index Terms **-Composite damage, Finite element method, Unit cell model, Cohesive element, Damage evolution, Abaqus (FEA)**

I. INTRODUCTION

Composite material gives better overall performance because of its properties such as high stiffness and fatigue resistance, low weight and thermal expansion which single engineering material cannot provide. Composite materials used in various engineering fields such as aerospace, transportation, sports, infrastructure and many other sectors like biomedical industry, consumer goods, agricultural equipment, heavy machinery, computers, healthcare. The properties of composite materials are not same as common engineering materials which are homogeneous and isentropic, but they are transversely orthotropic materials. Composite materials shows different type of failures than common engineering materials. Study of composite failure has been done with damage mechanics which deals with analyses of microstructural events in solids responsible for changes in their response to external loading. Failure and damage in composite materials can be studied at microscopic and macroscopic level. Microscopic damage involves breaking of fibers, development of microcracks in matrix, debonding between fibers and matrix. Macroscopic damage involves delamination of laminate. Work has been restricted to the objective of obtaining an understanding of composite fractures at microscopic level only.

In order to model the damage and failure of fiber reinforced composites under mechanical load representative volume element (RVE) or unit cell model has been used. The unit cell is the smallest volume over which a measurement can be made that will yield a value representative of the whole. In this work, unit cell models to simulate the damage evolution, interaction between different damage mechanisms in unidirectional glass fiber reinforced composites under tension load. Mishnaevsky and Brondsted[1] studied damage evolution of glass fiber reinforced composite with unit cell models. They used special program code Meso3DFiber for the commercial software MSC/PATRAN which allows to automatic generation of 3D micromechanical finite element unit cell models of composites. Wang et al.[2] done the numerical simulations of damage evolution for composite unit cell models. Two numerical damage models cohesive elements and damageable layers are used for single fiber unit cell, multiple fiber unit cell with one and several damageable sections per fibers. They created number of 3D unit cell models using program code Meso3DFiber. Kottnera et al. [3] studied parameters of cohesive elements for modeling of adhesively bonded joints of epoxy composites. They evaluated the critical strain energy release rate and the critical opening displacement required for cohesive zone modelling. Mendes and Fretas [4] calculated fracture toughness in failure modes of tension, shear and scissoring modes. Kanit et al. [5] determined size of representative volume element (RVE) for random composites. Representative volume element (RVE) size is proposed for two-phase three-dimensional Voronoi-mosaic microstructure. They abandoned the idea that there exists one single possible minimal RVE size. Chari and Reddy [6] studied interface debonding and particle fracture for metal matrix composites. Damage modelling done using representative volume element (RVE) and testing has been done using Universal tensile test machine (UTM) and RVE justified with experimentation.

This work provides numerical modelling of damages in unit cell model studied by Mishnaevsky and Brondsted [1] for damage study. Cohesive zone model used by Wang et al. [2] used to model fiber, interface damage and matrix crack without using programming.

II. DAMAGE MODELING IN UNIT CELL

Composite damage modelled and simulated in Abaqus (FEA) software in the present work. The dimensions of the unit cell models used 10 ×10 ×10 mm^3 with definite fiber volume content in single-fiber model. The epoxy matrix composite reinforced with glass fiber was studied which shows elastic-brittle behavior. Cohesive elements used to model damageable layer in unit cell. Failure behavior of this cohesive elements tracked by traction separation response. Stress vs strain relation mostly linear-elastic in a first part and then tracked by degradation until the material fully losses its stiffness, There are three typical models to describe the post-damage process of a traction-separation, including exponential model, trapezoidal model and bilinear model. In this study, bilinear model was employed. Damage initiation refers to the beginnings of the degradation of the response of a material point. The process of degradations begins when the stresses and/or strains satisfy certain damage criterion. Here maximum nominal stress criterion was selected, which was assumed to initiate when the maximum nominal stress ration reaches a critical value. Once the damage initiation criterion reached, the damage evolution describes the rate of degradation of the material. A scalar damage variable D was introduced to represent the overall damage process. D monotonically evolves from 0 (no damage) to 1(overall damage) upon further loading after damage initiation. The stress tensor is shown in Equation (1)

$$t = (1 - D)t_0 \qquad \text{... ... ... [1]}$$

Where t_0 represents the stress tensor, if on damage occurs and t is the reduced actual stress tensor. In the cohesive elements model, cohesive elements were placed in the damageable layers of fibers, matrix or interfaces. Three-dimensional 6-node linear triangular prism cohesive elements COH3D6 used for fiber and matrix and three-dimensional 8-node linear brick cohesive elements COH3D8 were used in interface cohesive layers. C3D6 elements used for solid fiber and matrix sections. The cohesive elements connected to other elements by sharing nodes. Properties of phases used for modelling listed in Table 1

Table 1 Properties of materials used in modelling

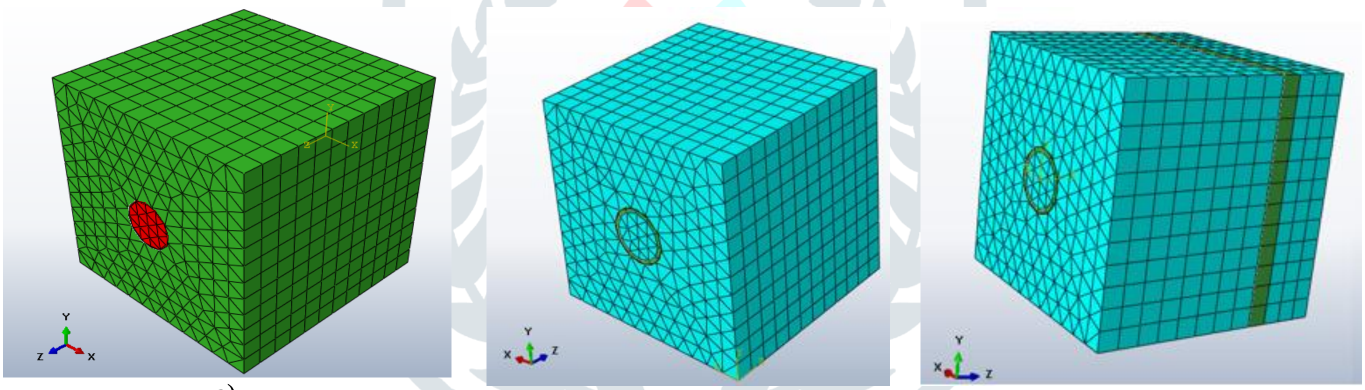
Phase	Material	Young' s modulus (GPa)	Poisson's ratio	Modulus of rigidity (GPa)
Fiber	Glass	72	0.26	28.57
Matrix	Epoxy	3.79	0.37	1.38
Interface	Averaged properties of fiber and matrix	37.9	0.37	13.83

The stiffness of the cohesive elements was calculated as Young's modulus divided by the thickness of cohesive element. The maximum nominal stress traction-interaction failure criterion was selected for the damage initiation in the cohesive elements, and the energy-based damage evolution law is selected for damage propagation. The relation between the damage dissipation energy and the strength of the cohesive layer is nonlinear. The relation between damage dissipation energy per unit area and damage initiation stress can be expressed by Equation (1),

$$E = 0.000042\sigma^2 - 0.000275\sigma + 0.2088 \qquad \text{... ... ... [2]}$$

III. UNIT CELL MODEL WITH FIBER DAMAGE

Glass fiber was modelled in Abaqus (FEA) software with diameter of 1.13 mm. fiber contains 4% volume of unit cell model which comes out 40 mm^3. Fiber is modelled in two parts so that damageable cohesive element layer can modelled in between the fiber parts. One part of fiber modeled with length of 5mm and second part 4.6mm. Overall length of fiber was 10mm including the damageable layer thickness of 0.4mm. Matrix contains 96% of total unit cell which comes out 960 mm3. Three damage model shown in figure 1, single fiber damage shown in (a), interface damage in (b) and interface with matrix crack shown in (c).



a) b) c)

Fig-1: single fiber unit cell model a) fiber damage b) fiber with damageable interface c) fiber with interface and matrix crack

Matrix made up of epoxy material and meshed with C3D6 elements, which are solid prism elements. Cohesive elements modelled as orphan mesh on existing solid meshed part. Elements used were COH3D6, which are prism elements, this cohesive elements share the nodes with the parent part on which they are modelled and tied on other side to solid part. Properties of these cohesive elements has been same as solid fiber element except they were made ready to fail assigning them cohesive element properties. Material response of cohesive elements is traced by traction separation response which will fail once value of applied load reached to predefined value.

Properties needed to be defined for unit cell model with only fiber damage are glass, epoxy and cohesive element properties for traction separation response. Glass properties are used for fiber solid part as mentioned in table 1. For fiber cohesive element strength of 100 MPa, energy for damage initiation can be calculated by using equation (2) as

$$E = 0.000042 \times 100^2 - .000275 \times 100 + 0.2088$$

$$E = 0.6013\ mJ.$$

Stiffness in normal direction for fiber cohesive elements calculated by dividing Young's modulus by thickness of cohesive element and stiffness in transverse direction is calculated by dividing rigidity modulus by cohesive element thickness. Normal stiffness for cohesive element for traction-separation response is given by equation,

$$Knn = E/t \qquad \text{... ... ... [3]}$$

Where, E is Young's modulus and t is thickness of cohesive element.

For this work, E= 72000 MPa and thickness of fiber cohesive elements is 0.4 mm.

$$Knn = 72000/0.4$$

$$Knn = 180000\ MPa$$

Similarly, transverse stiffness for cohesive element for traction-separation response is given by equation,

$$Ktt = G/t \qquad \text{... ... ... [4]}$$

Where, G is Rigidity modulus, for this work its value is 28570 MPa,

$$Ktt = 28570/0.4$$
$$Ktt = 71425\ MPa$$

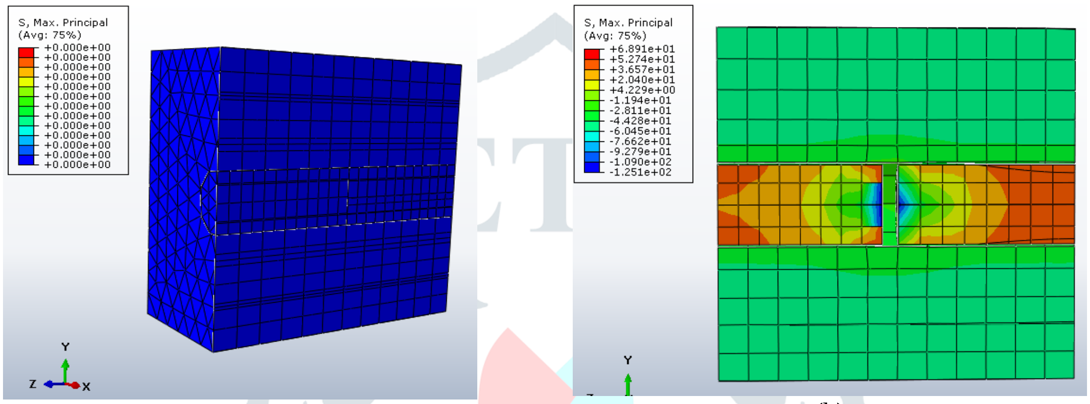
Table 2 Fiber cohesive element properties

Properties	Damageable layer thickness(mm)	Stiffness(N/mm^2)		Damage Strength(N/mm^2)	Damage dissipation energy (mJ)
		Knn	Ktt		
Fiber cohesive elements	0.4	80000	71425	100	0.6013

For all the simulations in this work boundary conditions remains the same. Boundary conditions for unit cell is as follows:

1. For only one corner node all the degrees of freedom are fixed. i.e. node with co-ordinates (0,0,0)
2. For bottom surface displacement in z-direction is zero.
3. For line with x=0 and z=0, first degree of freedom is zero.
4. For sides, other than bottom and top, first and second degrees of freedom are zero.
5. For top side uniform displacement load is applied in z- direction.

Maximum principal stresses distribution before and after damage shown in Figure 2.



(a) (b)

Fig-2: single fiber unit cell model with fiber damage a) before damage b) after damage

Stress vs strain graph for unit cell with fiber and interface damage is shown in Figure 3.

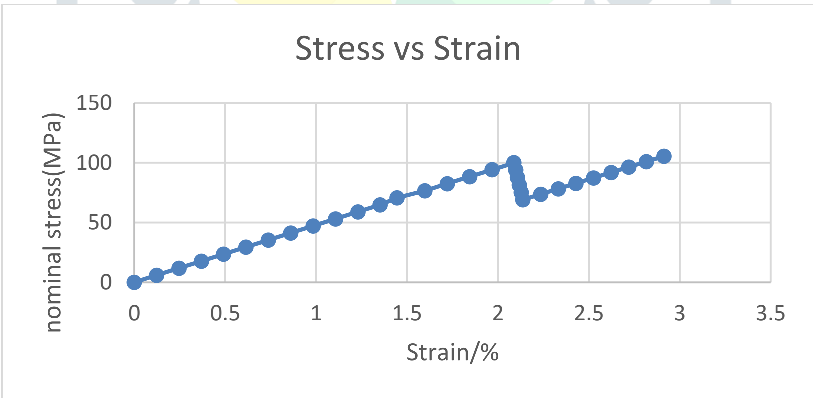


Fig-3: Stress vs strain graph for unit cell with fiber and interface damage

IV. Unit cell model with fiber and interface damage

Unit cell modelled for this damage is similar to previous model with addition of damageable interface. For fiber strength of 80 MPa energy for damage initiation can be calculated by using equation (2) as

$$E = 0.000042 \times 80^2 - 0.000275 \times 80 + 0.2088$$
$$E = 0.4556\ mJ$$

Similarly for interface strength of 10 MPa energy for damage initiation can be calculated as

$$E = 0.000042 \times 10^2 - 0.000275 \times 10 + 0.2088$$
$$E = 0.21025\ mJ$$

Normal stiffness for fiber cohesive elements kept same as for fiber damage and for interface is given by equation (3), for this work, E= 37900 MPa and thickness of interface cohesive elements is 0.2 mm.

$$Knn = 37900/0.2$$
$$Knn = 189500\ MPa$$

Similarly, stiffness for cohesive element for traction-separation response is given by equation (4), for this work its value for interface is 13.83 GPa,

$$Ktt = 13830/0.2$$

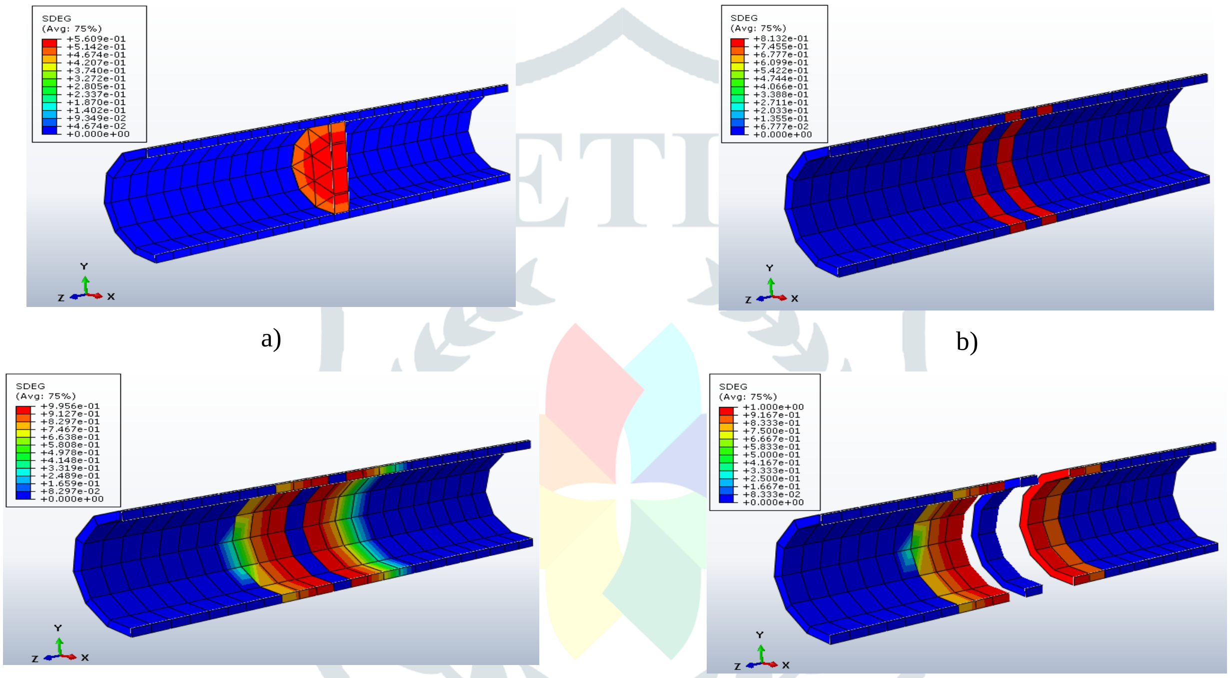
$Ktt = 69150\ MPa$

For fiber failure with damageable interface properties of cohesive elements are listed in Table 3

Table 3 cohesive element properties for fiber and interface damage

Properties	Damageable layer thickness (mm)	Stiffness (N/mm^2)		Damage Strength (MPa)	Damage dissipation energy (mJ)
		Knn	Ktt		
Fiber cohesive elements	0.4	180	71.425	80	0.4556
Interface	0.2	189.5	69.15	10	0.21025

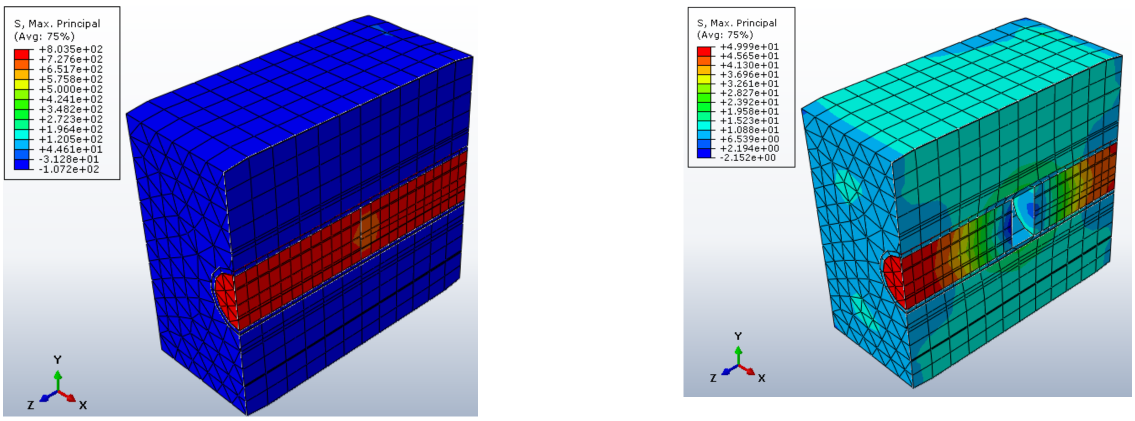
Damage evolution with scalar degradation variable for damageable cohesive element with fiber and interface damage shown in Figure 4



a) b) c) d)

Fig-4: single fiber unit cell model with fiber damage a) fiber damage b) Fiber damage complete c) interface damage d) interface damage complete

Maximum principal stress distribution for unit cell with fiber and interface damage shown in Figure 5.



a) b)

Fig-5: single fiber unit cell model with fiber and interface damage a) Before failure b) After failure

Stress vs strain graph for unit cell with fiber and interface damage shown in Figure 6.

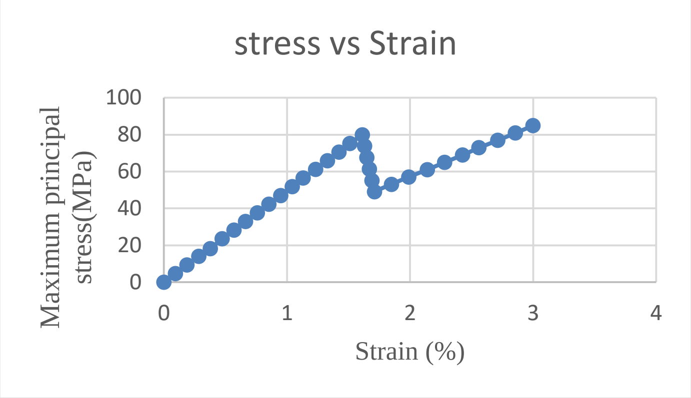


Fig-6: Stress vs strain graph for unit cell with fiber and interface damage

UNIT CELL DAMAGE WITH FIBER, INTERFACE AND MATRIX CRACK

Cohesive element properties for traction separation response calculated as for unit cell as following

For fiber strength of 50 MPa energy for damage initiation calculated by using equation (2) as

$E = 0.000042 \times 50^2 - 0.000275 \times 50 + 0.2088$

$E = 0.30005\ mJ$

Similarly for interface strength of 16 MPa energy for damage initiation can be calculated by using equation (2) as

$E = 0.000042 \times 16^2 - 0.000275 \times 16 + 0.2088$

$E = 0.215152\ mJ$

For matrix strength of 5 MPa energy for damage initiation can be calculated by using equation (2) as

$E = 0.000042 \times 5^2 - 0.000275 \times 5 + 0.2088$

$E = 0.208475\ mJ$

Normal stiffness for fiber cohesive elements and interface are kept same as for fiber damage and interface damage model, for matrix cohesive elements normal stiffness is given by equation (3), Young's modulus and t is thickness of cohesive element. For this work, E= 3790 MPa and thickness of matrix cohesive elements is 0.4 mm.

$Knn = 3790/0.4$

$Knn = 9475\ MPa$

Similarly, transverse stiffness for cohesive element for traction-separation response is given by equation (4), Rigidity modulus, for matrix material was 13.83 GPa,

$Ktt = 1380/0.4$

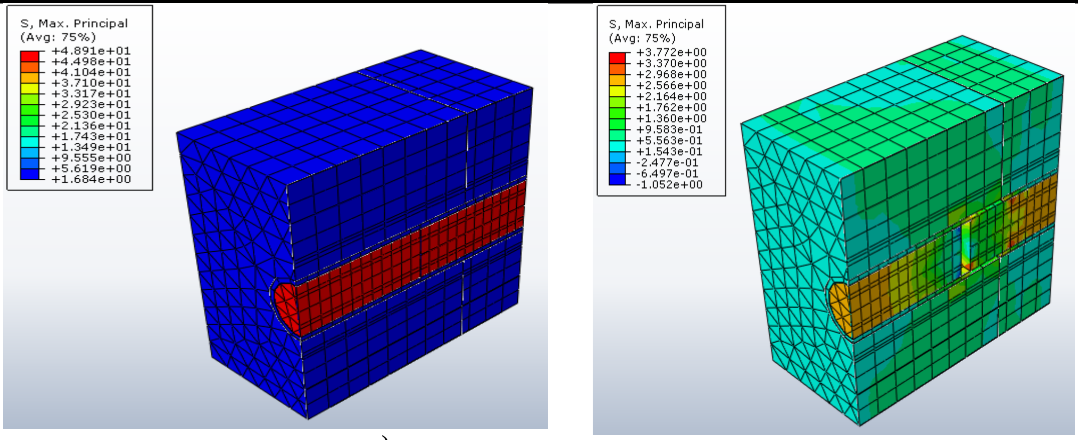
$Ktt = 3450\ MPa$

For unit cell model with all three damages of fiber, fiber/matrix interface and matrix damages properties of cohesive elements has been listed in table 4.

Table 4 cohesive element properties for fiber and interface and matrix crack

Properties	Damageable layer thickness(mm)	Stiffness(K)		Damage Strength(MPa)	Damage dissipation energy (mJ)
		Knn	Ktt		
Fiber cohesive elements	0.4	180	71.425	50	0.30005
Interface	0.2	189.5	69.15	16	0.215152
Matrix cohesive element	0.4	9.475	3.45	5	0.208475

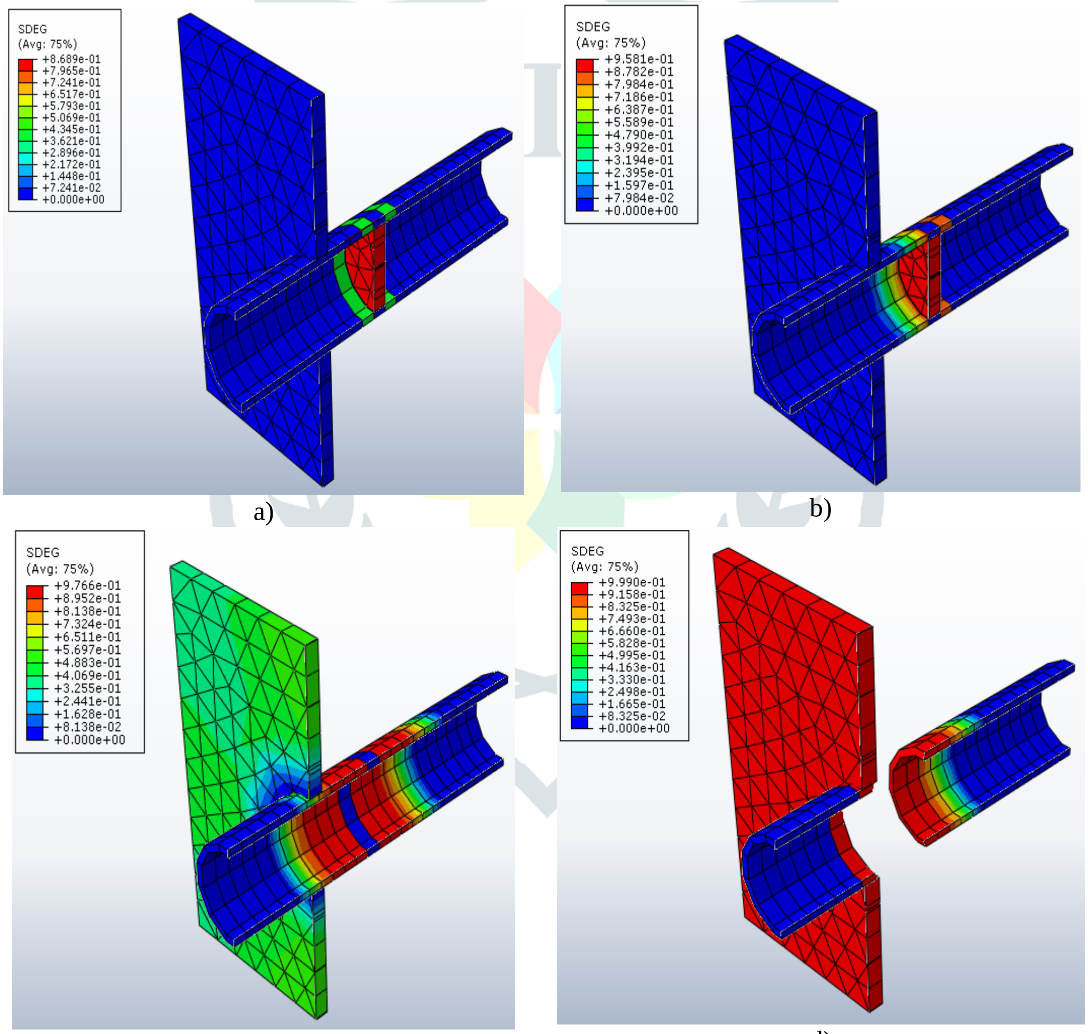
Maximum principal stress distribution for unit cell model with fiber, interface and matrix crack is as shown in figure 7.



a) b)

Fig-7: single fiber unit cell model with fiber damage a) before damage b) after damage

Damage evolution with scalar degradation variable for damageable cohesive element with fiber, interface and matrix damage shown in figure 8 it was concluded that fiber cohesive elements gets damaged first followed by interface and matrix cohesive elements.



c) d)

Fig-8: single fiber unit cell model with fiber damage a) fiber damage b) interface damage c) matrix damage d) total damages

Stress vs strain graph for unit cell with fiber, interface and matrix damage is shown in figure 9.

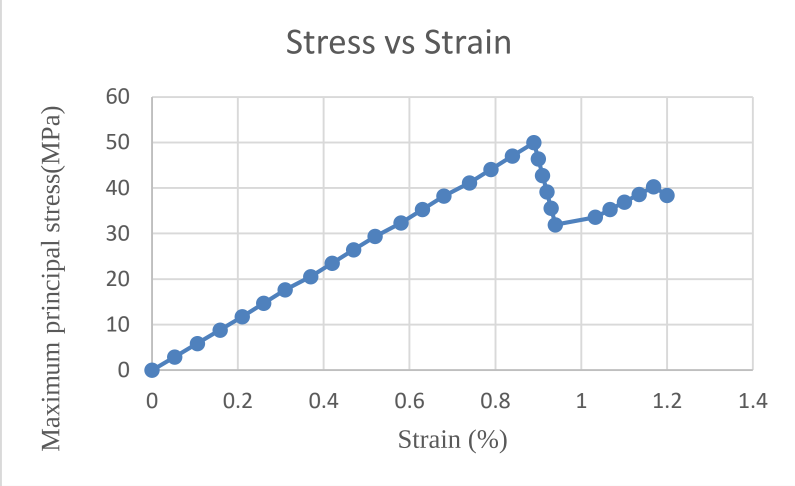


Fig-9: Stress vs strain graph for unit cell with fiber, interface and matrix damage

V. CONCLUSION

Three different kinds of damages such as fiber breakage, matrix cracking and fiber matrix interface debond are studied in a representative volume element (RVE) or unit cell model. For modelling these damages cohesive elements has been introduced in damageable layer. These cohesive elements introduce geometric non-linearity in unit cell and failure of these cohesive elements was defined by traction-separation response. Work performed on unit cell model for damage evolution in composites can be used to predict the damage behavior of composite material in lamina level. Damage evolution is studied in three types of unit cell models. The first model shows only fiber failure, second model shows fiber and matrix interface failure and the third model shows all the three damages (i.e. fiber breakage, matrix cracking and fiber matrix interface debond). Third model which includes all the three damages used to predict the actual behavior of composite lamina. The present study gives a clear idea that fiber fails first, followed by fiber-matrix interface and finally matrix cracking.

VI. REFERENCES

1. Leon Mishnaevsky and PovlBrøndsted, "Micromechanisms of damage in unidirectional fiber reinforced composites: 3D computational analysis", *Composites Science and Technology 69 (2009) 1036–1044*.
2. H.W. Wang, H.W. Zhou, L. Mishnaevsky Jr., P. Brøndsted and L.N. Wang, "Single fiber and multifiber unit cell analysis of strength and cracking of unidirectional composites", *Computational Materials Science 46 (2009) 810–820.*
3. R. Kottnera, R. Hynekb and T. Kroupa, "Identification of parameters of cohesive elements for modeling ofadhesively bonded joints of epoxy composites", *Applied and Computational Mechanics 7 (2013) 137–144.*
4. R. Mendes and M. Freitas, "DCB, ENF and ECT TESTS for interlaminar fracture of glass/epoxy unidirectional laminates",*10th Portuguese conference on fracture – 2006.*
5. T. Kanit, S. Forest, I. Galliet, "Determination of size of representative volume element for random composites: statistical and numerical approach", *International journal of solids and structures 40 (2003) 3647-3679*
6. B. Chari and A. Reddy, "finite element modelling and experimental validation of interphase debonding and particle fracture in Titanium carbide/AA1100 alloy composites", *3rd National conference on material and manufacturing processes.*
7. Krishan K. Chawla, a book on "Composite Materials", third Edition, pp 7-68
8. Ramesh Talreja and Janis Varna, a book on "Modeling Damage, Fatigue and Failure of Composite Materials", pp 1-24

9. Nachiketa Tiwari, lecture series on “Introduction to Composite Materials and Structures”, Indian Institute of Technology Kanpur.
10. A.W. Pryce and P. A. Smith, “Behaviour of unidirectional and crossply ceramic matrix composites under quasi-static tensile loading” Department of materials science and engineering, university of surrey, UK.
11. S. N. Nair, A. Dasari, Chee Yoon Yue and S. Narasimalu, “Failure Behavior of Unidirectional Composites under Compression Loading: Effect of Fiber Waviness”, Energy Research Institute at NTU/Interdisciplinary Graduate School, Singapore.
12. Miguel Herraez, Diego Mora, Fernando Naya, and Claudio Lopes, “Transverse cracking of cross-ply laminates: A computational micromechanics perspective”, IMDEA Materials Institute, C/Eric Kandel, Spain.
13. Ganesh Soni, Saurabh Gupta, Ramesh Singh, Mira Mitra and Wenyi, “Study of localized damage in composite laminates using micro–macro approach”, Department of Mechanical Engineering, IITB-Monash Research Academy, Mumbai, India.
14. R. W. Mines and A. Alias, “Numerical simulation of the progressive collapse of polymer composite sandwich beams under static loading”, Composites: Part A 33(2002), 11-26.
15. Huaiwen Wang, Qing-Hua Qin, Hongwei Ji and Ying Sun, “Comparison Among Different Modeling Techniques of 3D Micromechanical Modeling of Damage in Unidirectional Composites”, Advanced Science Letters Vol. 4, 400–407, 2011.
16. Peter Feraren and Henrik Jensen, “Cohesive zone modelling of interface fracture near flaws in adhesive joints”, Volume 71, Issue 15, October 2004, Pages 2125-2142.
17. Abaqus (6.14) analysis user manual, 29.5.3 modelling with cohesive elements.