International Journal of Engineering & Technology, 7 (4.26) (2018) 235-239

International Journal of Engineering & Technology

Website: www.sciencepubco.com/index.php/IJET

Research paper

Failure Analysis of Woven Kevlar/Epoxy under Uniaxial Tension

Norzihan Rahimi[1], Ahmad Kamil Hussain[2*], Mohd Sultan Ibrahim Shaik Dawood[3], Adib Zulfadhli Mohd Alias[4], Muhamed Hafiy Afiq Md Yusoff[5], Jamaluddin Mahmud[6]

[1]Faculty of Mechanical Engineering, Universiti Teknologi MARA, Shah Alam, Selangor, Malaysia
[2] Department of Mechanical Engineering, International Islamic University Malaysia
**Corresponding author E-mail: ahmadkamil@salam.uitm.edu.my*

Abstract

The increasing demand for newer materials with versatile properties such as high strength-to-weight ratio; has made fiber reinforced composite materials a favourable choice in various applications, particularly in the automotive, aerospace, marine, sports and defense industries. Moreover, the properties of a composite material could be tailored for specific functions or applications. Despite its many attractive features, composite material fails in a complex manner involving matrix failure, fiber failure and delamination. This failure behaviour needs to be well established. The objective of this study is to perform failure analysis on a woven Kevlar/Epoxy composite laminate subjected to uniaxial tension and establish its failure trend. The lamination sequence is $(\theta_4/0_4/-\theta_4)_S$ where the angle, θ, ranges from 0^o to 90^o. The failure analysis was carried out using a commercial finite element software, Ansys and comparisons were made using analytical methods (Matlab). The values of stresses were computed and Maximum Stress Theory was employed to check for failure. The trend of failure, in terms of the failure curves (normalized first ply failure and last ply failure loads), for woven Kevlar/Epoxy was established. This study had produced new failure datas for woven Kevlar/Epoxy and thus, contributes significant knowledge about the failure behaviour of composite materials.

Keywords: *Kevlar/epoxy, composite laminate, failure analysis, Ansys, Matlab*.

1. Introduction

Applications of composites have been expanded widely in various fields such as aircraft, sporting goods, automotive and in biomedical industries. Kevlar is one of the many kinds of fibers which has been used extensively due to its excellent mechanical properties, light weight, toughness, tensile strength, unique flexibility and resistance to impact damage [1][2]. Kevlar fiber hybridized with other synthetic fibers can improve the compressive strength and delamination strength of Kevlar composites [2][3].
Presently, either physical test or any advanced computational method could be used to evaluate the performance of a composite to withstand crucial loading. The drawbacks of physical experiments are repetitive tests need to be carried out as well as adequate number of samples need to be prepared [4]. Thus, it is a time and cost consuming process. Computational methods such as finite element analysis, FEA, eliminate the limitations associated with physical experiments. This has led to many composite related investigations to be conducted using FEA tools such as Ansys, Ls-Dyna and Abacus. In 2016, Sukirmar has evaluated ballistic response of the Kevlar/epoxy laminates for different velocities, thickness, lay-up and orientations using finite element approach [5].
There are several failure criteria available to be used in failure analysis. In this study the Maximum Stress Theory was used. Failure in the composite material is considered when the stress in the principal axis exceeds the corresponding strength in that direction [4][6].
The objective of this study is to perform failure analysis on a woven Kevlar/Epoxy composite laminate, subjected to uniaxial tension and establish its failure trend, using a commercially available finite element software and analytical method; and to compare the results obtained from the two approaches. This is novel as no similar study adapting this approach, has been reported before.

2. Methodology

Fig. 1 provides a flowchart detailing the methodology used in carrying out the intended investigation. The investigation includes two stages, i.e., the validation stage and failure analysis stage.

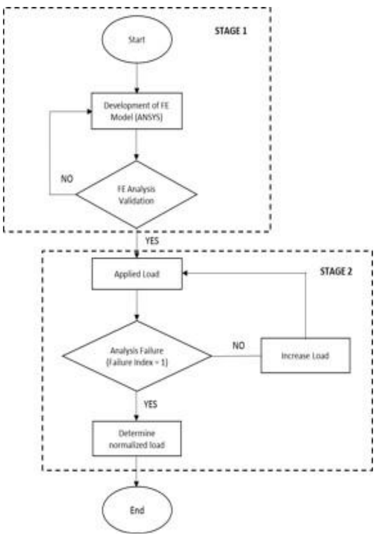

Fig. 1: Flowchart detailing the adopted methodology

3. Validation

To prove that the current FE model and FE implementation is acceptable, the numerical validation procedure was performed first. In FE simulation, this initial stage is crucial before proceeding to the actual case study (the main objective of the current study). For the numerical validation purposes, an established case study conducted by other researchers [7][8] related to the current study was selected, where the current FE were compared to exact solutions [8]. Composite laminates made of Graphite/ Epoxy subjected to uniform transverse loads with several lamination schemes were modelled and simulated using a commercially available software, Ansys (v16.0 2014 SAS IP, Inc). The composite laminate geometry is shown in Fig. 2 and the material properties are shown in Table 1. E_1, E_2, E_3 are referring to the Elastic Modulus in the principal directions 1, 2 and 3 respectively. ν_{12}, ν_{23}, ν_{13} refer to the Poisson's ratio in the principal planes 12, 23 and 13 respectively. G_{12}, G_{23}, G_{13} refer to the Shear Modulus in the principal planes 12, 23 and 13 respectively. The composite laminate was meshed into 18 elements using eight-noded shell elements (shell 281). The laminate was simply supported and 0.1 psi uniformly distributed load (UDL), q_o, was applied on top of the laminate in the transverse direction (negative z-direction). The maximum deflection of these FE models were determined, recorded and compared to the exact solutions. The results from the FE solution are presented in Table 2. The results clearly show that the current FE models and solutions are valid, where the errors are less than 1% for all lamination schemes.

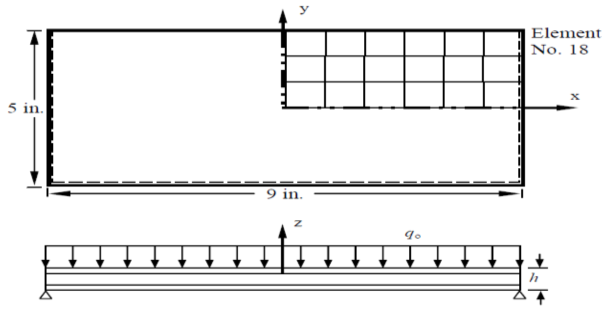

Fig. 2: Geometry of the composite laminate subjected to uniformly transverse load.

Table 1: Material properties of Graphite/Epoxy [8]

Elastic Parameters	Values
E_1	19.2×10^6 psi
$E_2 = E_3$	1.56×10^6 psi
$\nu_{12} = \nu_{23}$	0.24
ν_{13}	0.49
$G_{12} = G_{13}$	0.82×10^6 psi
G_{23}	0.49×10^6 psi
Ply thickness, h_i	0.005 in./ply

Table 2: Current FE solutions (maximum deflection, z-displacement) compared to exact solutions [8]

Lamination scheme	UDL (psi)	Exact Solution (in.)	ANSYS (in.)	Error (%)
[0/90/0/90]	0.1	0.134	0.135	0.75
[0/90/90/0]	0.1	0.229	0.23	0.44
[45/-45/45/-45]	0.1	0.1086	0.109	0.37
[15/-15/15/-15]	0.1	0.2515	0.2519	0.16
[45/-45]	0.1	1.6006	1.603	0.15
[15/-15]	0.1	2.6039	2.607	0.12

4. Failure Analysis

The objective of this study is to perform failure analysis on a woven Kevlar/Epoxy composite laminate subjected to uniaxial tension and establish its failure trend. Fig. 3 shows the typical loading and boundary conditions for a uniaxial tension model. Therefore, a square composite laminate with the dimensions of 20mm×20mm (*a* x *a*) made of 24 layers of woven Kevlar/epoxy was considered. The composite laminate was arranged in the sequence of $(\theta_4/0_4/-\theta_4)_S$ where the angle, θ, ranges from 0^o to 90^o. The material properties of a single lamina of woven Kevlar/epoxy are given in Table 3. As stated earlier in the previous section, E_1, E_2, E_3 are referring to the Elastic Modulus in the principal directions 1, 2 and 3 respectively. ν_{12}, ν_{23}, ν_{13} refer to the Poisson's ratio in the principal planes 12, 23 and 13 respectively. G_{12}, G_{23}, G_{13} refer to the Shear Modulus in the principal planes 12, 23 and 13 respectively. The strength of the Kevlar/epoxy lamina is denoted in Table 3 by X_T and X_C, the strength of fiber in tension and compression respectively; by Y_T and Y_C, the strength of matrix in tension and compression respectively; and *S*, its shear strength. An aspect ratio, S= a/h = 150 was considered, and therefore the plate thickness, h = 0.133mm with a cross sectional area of $2.666 mm^2$.

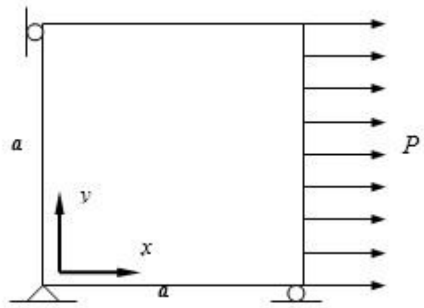

Fig. 3: Uniaxial tension model

Table 3: Material properties for woven Kevlar/epoxy [9]

Elastic & Strength parameters	Values
E_1	26.18 GPa
$E_2 = E_3$	26.18 GPa
$\nu_{12} = \nu_{23} = \nu_{13}$	0.11
$G_{12} = G_{23} = G_{13}$	1.53 GPa
X_T	420 MPa
X_C	150 MPa
Y_T	420 MPa
Y_C	150 MPa
S	106 MPa

For the element simulation, a commercially available software, ANSYS (v16.0 2014 SAS IP, Inc) was used. The Kevlar/epoxy composite laminate was meshed using eight-noded shell elements (shell 281). The boundary conditions for uniaxial tension model as shown in Fig. 3 was applied. When uniaxial tensile load, *P* was applied, stresses induced into the laminate. Failure occurred when the applied load induced principal stresses greater than the lamina strength (fiber, matrix or shear strength). Therefore, for the FE failure analysis using Ansys, the failure predicted was based on the available built in failure theories and failure criteria available in Ansys, which is Maximum Stress Failure Criterion as shown in Eq. (1). In the general FE failure procedure, Ansys indicates the laminate failure when the failure index is 1.

$$\sigma_1 \geq X_T \text{ or } \sigma_1 \leq X_C;\ \sigma_2 \geq Y_T \text{ or } \sigma_2 \leq Y_C;\ \tau_{12} \geq S \qquad (1)$$

In determining the first ply failure (FPF) load, the lowest load, *P* that induced any first failure on the Kevlar/epoxy composite laminate was actually representing the FPF load. This load was recorded as FPF load. The procedure was repeated for all lamination scheme of $(\theta_4/0_4/-\theta_4)_S$ as the angle θ, increased in a step size of 10^o from 0^o to 90^o**.** In determining the last ply failure (LPF) load, the lowest load, *P* that induced the total (final) failure of the Kevlar/epoxy composite laminate, where all the layers failed, was actually representing the LPF load. This load was recorded as LPF load. And again, the procedure was repeated for all lamination scheme of $(\theta_4/0_4/-\theta_4)_S$ as the angle θ, increased in a step size of 10^o from 0^o to 90^o**.**

For comparison, analytical method was also adopted to compute the strains and stresses (global and principal) for the Kevlar/epoxy composite laminate due to the uniaxial tensile load applied on it. The strains and stresses computation was based on the general mechanics of materials approach involving equations such as Eq 2 to Eq. 10 in Appendix 1. Due to the size of the matrix calculation, as well as complexity in solving the mathematical equations, a Matlab programme was written to aid the computation. For the failure analysis, when the load increased, the stresses value increased. Similar to the FE simulation, Maximum Stress Failure Criterion (Eq. 1) was employed to predict the failure of the Kevlar/epoxy composite laminate. The FPF load and LPF load were determined, recorded and used as reference for the comparison between FE simulation and analytical method.

5. Results and Discussion

The sample outputs from the Ansys simulation is shown in Fig. 4. The sample outputs from the Matlab programme is shown in Fig. 5. In general, both figures display the failure index computed by both methods, FE simulation and analytical method. It is obvious to see that the main advantage of using a FE software is that it provides visualisation. The laminate deformation could be viewed instantly. However, the outputs from the Matlab programme are basically a table of numbers. Nevertheless, it is much better than computing the stresses of a laminate using hand calculation (without the aid of a programme).

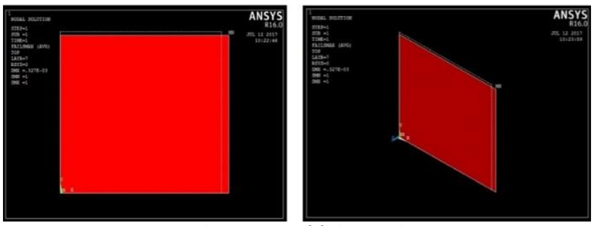

Fig. 4: Sample outputs of failure index (Ansys)

layer 1	2	3	4	5	6	7	8
45 degree	33126.48						
0.295773	0.295773	0.295773	0.295773	1.00E+00	1.00E+00	1.00E+00	1.00E+00
0.295773	0.295773	0.295773	0.295773	-4.08E-01	-4.08E-01	-4.08E-01	-4.08E-01
-0.36273	-0.36273	-0.36273	-0.36273	4.20E-33	4.20E-33	4.20E-33	4.20E-33

Fig. 5: Sample outputs of failure index (Matlab)

The results of the first ply failure and last ply failure are shown in Table 4 and Table 5 respectively. The tables compare the results obtained from Ansys and Matlab for the respective cases. From the tables it can be seen that the results from Ansys and Matlab conform well to each other.

Table 4: First ply failure

Angle	Ansys			Analytical Method		
	First Ply failure			First Ply failure		
	Line load	Load	Stress (MPa)	Line load	Load	Stress (MPa)
0°	55999.44	1119.989	419.991	55999.44	1119.989	419.991
10°	54033.05	1080.661	405.243	54033.44	1080.669	405.245
15°	51623.00	1032.460	387.168	51622.46	1032.449	387.160
20°	48376.17	967.523	362.817	48514.02	970.2803	363.850
30°	40434.10	808.682	303.252	40434.26	808.6852	303.253
40°	31109.03	622.180	233.315	34118.98	682.3796	255.889
45°	28964.80	579.296	217.233	33126.48	662.5296	248.445
50°	31109.00	622.180	233.315	34118.98	682.3796	255.889
60°	40433.85	808.677	303.250	40434.27	808.6853	303.253
70°	48376.20	967.524	362.817	48514.01	970.2802	363.850
75°	51622.08	1032.442	387.161	51622.46	1032.449	387.163
80°	54023.04	1080.461	405.168	54148.00	1082.960	406.104
90°	55999.46	1119.989	419.991	55999.83	1119.997	419.993

Table 5: Last ply failure

Angle	Ansys			Analytical Method		
	Last Ply failure			Last Ply failure		
	Line load	Load	Stress (MPa)	Line load	Load	Stress (MPa)
0°	55999.44	1119.989	419.991	55999.44	1119.989	419.990
10°	55773.85	1115.477	418.297	55772.44	1115.449	418.288
15°	55640.65	1112.813	417.300	55640.65	1112.813	417.299
20°	55790.78	1115.816	418.426	55791.12	1115.822	418.428
30°	59417.11	1188.342	445.623	59417.71	1188.354	445.627
40°	80175.44	1603.509	684.930	80179.60	1603.592	601.339
45°	91325.17	1826.503	684.930	112007.2	2240.144	840.043
50°	80175.37	1603.507	601.308	80179.60	1603.592	601.339
60°	59417.14	1188.343	445.623	59417.71	1188.354	445.627
70°	55790.82	1115.816	418.426	55791.12	1115.822	418.428
75°	55640.77	1112.816	417.301	55642.10	1112.842	417.310
80°	55773.87	1115.477	418.299	55772.44	1115.449	418.288
90°	55999.46	1119.989	419.991	55997.31	1119.946	419.974

A plot comparing the first and last ply failure is provided in Fig. 6 which compares the effect of ply angle on the failure stress. The pattern of failure is different for the first and last ply failure. In the case of first ply failure it can be seen that the failure stress reduces with respect to the ply angle up to 45^o after which there is an observed symmetry. Whereas, in the case of last ply failure, the failure stresses are higher in comparison to first ply failure and there is also an observed symmetry at 45^o. However, in the case of last ply failure, it is noticed that the for ply angles between 0^o to 15^o there is a slight drop in the failure stress values and beyond 15^o the values increase until ply angle of 45^o. From the plot it can be seen that the highest and lowest maximum stress that can be withstood by composite woven Kevlar/epoxy is 684.930 MPa at ply angle of 45^o and 417.3 MPa at ply angle of 15^o for the last ply failure. For the first ply failure, the highest and lowest maximum stress values that can be withstood by composite woven Kevlar/epoxy are 419.991 MPa at ply angle of 45^o and 217.233 MPa at ply angle of 15^o.

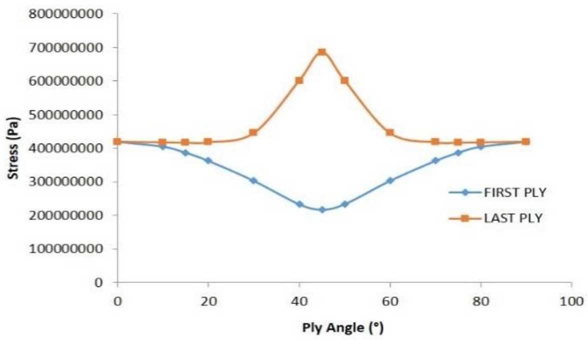

Fig. 6: Influence of ply angle on failure behaviour (FPF and LPF curves) of woven Kevlar/epoxy using Ansys

From the analysis conducted using Ansys, it was also observed that the composite plate stretched under the applied load. Figs. 7 and 8 show the observed elongation for 45^o ply angle for first and last ply failure respectively. The [A] [B] and [D] matrixes for 45^o and 0^o provided in Table 6 and 7 justify the observation made. In both the tables it can be seen that the coupling term [B] and bending term [D] are very small in comparison to the extensional term [A].

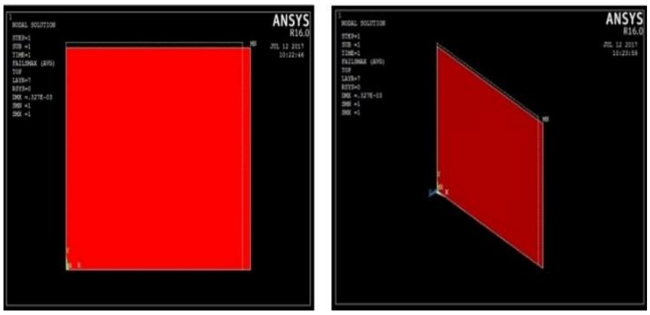

Fig. 7: (a) Front view (b) Isometric view. Predicted first ply failure on at ply angle of 45^o

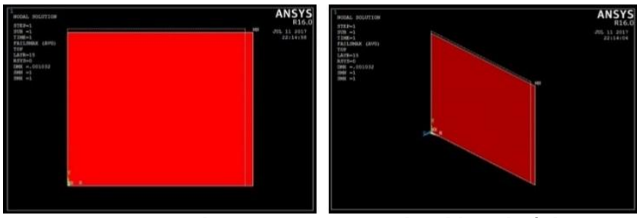

Fig. 8: (a) Front view (b) Isometric view. Predicted last ply failure on at ply angle of 45^o

Table 6: [A], [B], and [D] matrix obtained from Ansys for 45^o

First Ply Failure and Last Ply Failure			
	Angle 45°		
[A]=	0.262141E+07	0.130064E+07	-0.605845E-27
	0.130064E+07	0.262141E+07	0.605845E-27
	0.282728E-26	-0.282728E-26	0.111637E+07
[B]=	0.310862E-13	0.183187E-13	-0.175029E-29
	0.183187E-13	0.310862E-13	0.175029E-29
	0.825839E-30	-0.825839E-30	0.186517E-13
[D]=	0.373338E-02	0.207696E-02	0.206342E-18
	0.207696E-02	0.373338E-02	-0.206342E-18
	0.457508E-18	-0.457508E-18	0.180396E-02

Table 7: [A], [B], and [D] matrix obtained from Ansys for 0^o

First Ply Failure and Last Ply Failure			
	Angle 0°		
[A]=	0.353339E+07	388672.	0.00000
	388672.	0.353339E+07	0.00000
	0.00000	0.00000	204398.
[B]=	0.723865E-13	0.455191E-14	0.00000
	0.455191E-14	0.723865E-13	0.00000
	0.00000	0.00000	0.408007E-14
[D]=	0.523454E-02	0.575799E-03	0.00000
	0.575799E-03	0.523454E-02	0.00000
	0.00000	0.00000	0.302806E-03

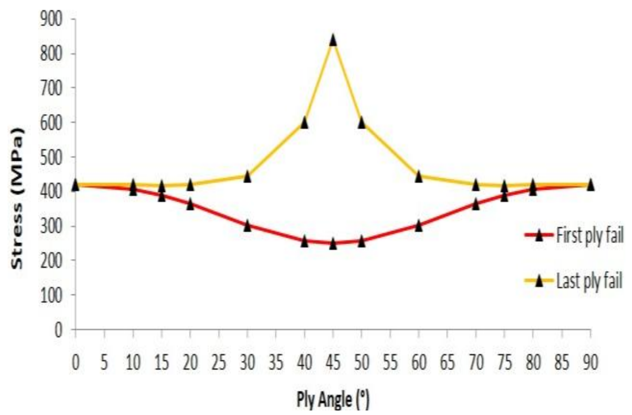

Fig. 9: Influence of ply angle on failure behaviour (FPF and LPF curves) of woven Kevlar/epoxy calculated using Matlab

Fig. 9 shows the results of calculated failure stress values against the ply angle on the first ply failure and last ply failure using Matlab. The trend of the results is the same as to that obtained using Ansys. For the first ply failure the stress values decrease with the increase in the ply angles and there is an observed symmetry at 45^o. In the case of last ply failure, the stress values increase with increasing ply angles. From the plots it can be seen that ply angle 45^o offers best resistance to complete failure as the failure stress required is about 840 MPa. This is due to the fact that woven composites will retain more stress to break at the last ply at ply angle of 45^o. The calculated reduced stiffness [Q], transformed reduced stiffness [$\bar{\boldsymbol{Q}}$] and [A], [B], [D] coefficients are shown in Table 8. From the table it can be seen that the calculated values of [A], [B], [D] coefficients are similar to that obtained from Ansys.

Table 8: [Q], [$\bar{\boldsymbol{Q}}$], [A], [B], and [D] matrix calculated using Matlab

Analytical result for First Ply fail and Last Ply fail

Angle	0°	45°
Q_{11}	2.65E+10	2.65E+10
Q_{12}	2.92E+09	2.92E+09
Q_{22}	2.65E+10	2.65E+10
Q_{16}	0	0
Q_{26}	0	0
Q_{66}	1.53E+09	1.53E+09
$\overline{Q_{11}}$	2.65E+10	1st layer =1.62E+10
		5th layer =2.65E+10
$\overline{Q_{12}}$	2.92E+09	1st layer =1.32E+10
		5th layer =2.92E+09
$\overline{Q_{16}}$	-	1st layer =1.91E-06
		5th layer =0
$\overline{Q_{22}}$	2.65E+10	1st layer =1.62E+10
		5th layer =2.6501E+10
$\overline{Q_{26}}$	-	1st layer =-1.91E-06
		5th layer =0
$\overline{Q_{66}}$	1.53E+09	1st layer =1.18E+10
		5th layer =1.53E+09

[A], [B], and [D] matrix

First Ply fail and Last Ply fail			
	Angle	0°	
[A]	3533411.17	388675.2	0
	388675.228	3533411	0
	0	0	204399.4
[B]	0	0	0
	0	0	0
	0	0	0
[D]	0.00523465	0.000576	0
	0.00057581	0.005235	0
	0	0	0.000303
	Angle	45°	
[A]	2621432.14	1300654	-1.29E-26
	1300654.26	2621432	1.29E-26
	-1.29E-26	1.29E-26	1116378
[B]	2.66E-15	-4.44E-15	0
	-4.44E-15	2.66E-15	0
	0	0	-8.88E-16
[D]	0.00373346	0.002077	2.51E-19
	0.002077	0.003733	-2.51E-19
	2.51E-19	-2.51E-19	0.001804

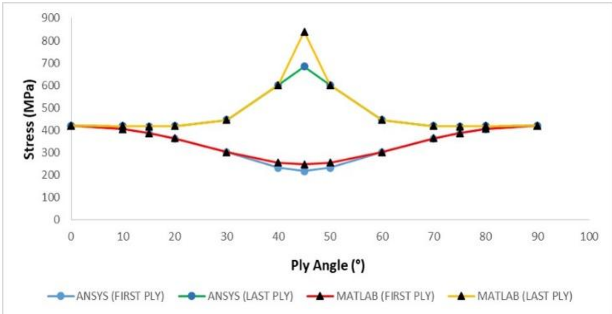

Fig. 10: Comparison of results obtained from Ansys and Matlab: influence of ply angle on FPF and LPF curves

Lastly, the results obtained by Ansys and Matlab are compared in Fig. 10. The failure stress values for first ply failure between that obtained from Ansys and Matlab are very close to each other. A maximum difference in results for the first ply failure between the two models occurs at ply angle of 45^o for which the difference is about 12.6%. In the case of last ply failure the difference is much higher which is at 18.5% for 45^o ply angle. The differences in the results can be attributed to mesh sensitivity and element type used for those obtained using Ansys.

6. Conclusion

This paper reports on the failure analysis of woven Kevlar/epoxy laminate subjected to uniaxial tension using Ansys and Matlab. The corresponding failure trend has also been highlighted. Results from both simulations were successfully obtained and discussed. The failure was predicted based on Maximum Stress Theory. It was found that the failure stress values obtained using Ansys were generally similar to that obtained using Matlab. Though a maximum difference of 18% was observed in the case of last ply failure at 45^o ply angle, this discrepancy could be due to mesh sensitivity for the results obtained using Ansys. As conclusion, the analytical model offers a simplistic approach to model and predict failure which can give significant insight to the understanding of the failure mechanism of simple composite models such as the one considered in the present study.

Acknowledgement

This work is supported by the Ministry of Education (MOE) Malaysia and Universiti Teknologi MARA (Fundamental Research Grant Scheme, grant no 600 - RMI/FRGS 5/3 (80/2014)).

References

Yang S, Chalivendra VB & Kim YK (2017), Fracture and impact characterization of novel auxetic Kevlar/Epoxy laminated composites. *Composite Structures*, vol. 168, pp. 120–129.
Singh TJ & Samanta S (2015), Characterization of Kevlar Fiber and Its Composites: A Review, *Mater. Today Proc.*, vol. 2, no. 4–5, pp. 1381–1387.
Bandaru AK, Patel S, Sachan Y, Ahmad S, Alagirusamy R & Bhatnagar N (2016), Mechanical behavior of Kevlar/basalt reinforced polypropylene composites, *Compos. Part A Appl. Sci. Manuf.*, vol. 90, pp. 642–652.
Noh NN, Samsudin AH & Mahmud J (2017), Failure Analysis of Glass/Epoxy and Graphite/Epoxy Laminates due to the Effect of Variation in Lamination Scheme and Angle of Fibre Orientation. *Materials Science Forum*, vol. 889, pp. 36–44.
Sikarwar RS, Rajput NS, Velmurugan R & Naik S (2017), FE Analysis of Impact on Kevlar/Epoxy Laminates with Different Orientations and Thicknesses. *Mater. Today Proc.*, vol. 4, no. 2, pp. 2599–2607.
Rahimi N, Hussain AK, Meon MS & Mahmud J (2012), Capability Assessment of Finite Element Software in Predicting the Last Ply Failure of Composite Laminates. *Procedia Engineering* 41, 1647-1653.
Rahimi N, Musa M, Hussain AK & Mahmud J (2012), Finite Element Implementations to Predict the Failure of Composite Laminates under Uniaxial Tension. *Advanced Materials Research*, Vol. 499, pp 20–24.
Mahmud J, AF Ismail AF & Pervez T (2005), Employing a failure criterion with interaction terms to simulate the progressive failure of carbon-epoxy laminates, *IEM Journal*, vol. 66(2), pp 6–14.
Icten BM, Karakuzu R & Toygar ME (2006), Failure analysis of woven kevlar fiber reinforced epoxy composites pinned joints, *Composite Structures*, vol. 73, pp 443–450.

Appendix 1

The reduced stiffness formula and transformed reduced stiffness (due to angle variation) were calculated using the equations below:

Reduced Stiffness Formula

$$Q_{11} = \frac{E_1}{1 - V_{12}V_{21}} \quad (2)$$

$$Q_{12} = \frac{V_{12}E_2}{1 - V_{12}V_{21}} \quad (3)$$

$$Q_{22} = \frac{E_2}{1 - V_{12}V_{21}} \quad (4)$$

$$Q_{66} = G_{12} \quad (5)$$

Transformed reduced stiffness formula

$$\overline{Q_{11}} = Q_{11}*m^4 + 2*(Q_{12}+2*Q_{66})*m^2*n^2 + Q_{22}*n^4 \quad (6)$$

$$\overline{Q_{12}} = (Q_{11}+Q_{22}-4*Q_{66})*m^2*n^2+Q_{12}*(m^4+n^4) \quad (7)$$

$$\overline{Q_{22}}= Q_{11}*n^4 + 2*(Q_{12}+2*Q_{66})*m^2*n^2+Q_{22}*m^4 \quad (8)$$

$$\overline{Q_{16}}= ((Q_{11}-Q_{12}-(2*Q_{66}))*n*m^3) + ((Q_{12}-Q_{22} + (2*Q_{66}))*n^3*m) \quad (9)$$

$$\overline{Q_{16}}= ((Q_{11}-Q_{12}-(2*Q_{66}))*n*m^3) + ((Q_{12}-Q_{22} + (2*Q_{66}))*n^3*m) \quad (10)$$

$$\overline{Q_{16}}= ((Q_{11}-Q_{12}-(2*Q_{66}))*n*m^3) + ((Q_{12}-Q_{22} + (2*Q_{66}))*n^3*m) \quad (11)$$

where:
m = cos θ and n = sin θ