14th International Conference on
AEROSPACE SCIENCES & AVIATION TECHNOLOGY,
ASAT - 14 – May 24 - 26, 2011, Email: asat@mtc.edu.eg
Military Technical College, Kobry Elkobbah, Cairo, Egypt
Tel: +(202) 24025292 –24036138, Fax: +(202) 22621908

Failure Analysis of Generally Loaded Composite Laminated Structures Using Finite Element Method

M.M. Hegaze*

Abstract: Composite materials are used in several industrial and military applications due to its weight to strength ratio. From the advantages of composite materials, it may pass through different modes of failure before reaching the complete failure. Many analyses have been developed to predict the failure of composite laminates. This paper presents the development of a progressive damage analysis methodology for stress analysis of composite laminated structures using high order finite element. It is an extension to the work developed by Moutaz [10] where the large effect force vector has been introduced separately to show the geometric nonlinear effect. A simple failure analysis technique has been developed to predict the mode of failure of composite laminated structures. This technique is performed based on the stress and strain values in longitudinal (fiber direction) and transverse (matrix direction) directions of the composite. The mode of failure has been detected and defined at each increment based on the value of stress and strain in fiber and matrix directions. The proposed technique and finite element derivation have been validated by comparing the obtained results with published results and with results obtained by ANSYS-12 commercial package for the same case studies.

Keywords: Composite material, Finite element method, Failure analysis

1. Introduction

Over the past years, Composite materials are used in several industrial and military applications due to its weight to strength ratio. From the advantages of composite materials, it may pass through different modes of failure before reaching the complete failure. A loaded structure goes through several stages to be completely failed; damage initiation, delamination, fiber breaking, damage growth, and fracture. Due to applying this load, energy is accumulated near flaws and defects that grow and unite, forming small cracks (damage initiation). In some applications, the damage initiation is considered a complete failure state of a structure, especially for those applications designed by fail-safe design criterion [1]. The next stage is the delamination, which is the debonding from one ply to another. It is caused by the discontinuities in the path of the load due to the matrix cracks and the small cracks through the interface. The critical stage of the damage development of composite materials, which leads to complete failure, is fiber breaking. The fiber may fail at the weakest point along its length or at a point of high stress concentration. Damage growth process is considered a better way to understand the failure phenomena; especially in the applications which have stress concentration effects.

* Egyptian Armed Forces, Egypt

The fracture is the final stage, which represents a complete failure of a component due to rapid progression of the damage modes. The fracture is the final stage of damage development, which represents a complete failure of a component due to rapid progression of the damage modes. Due to the complexities and restrictions of the existing failure analyses methods of composite materials and the existing computational tools, there is a need to develop a reliable method to predict the behavior and strength of composite laminates which can be used as a design guideline for structures made of composite materials.

Many researches on the progressive failure analysis of composite laminated structures have been successfully carried out using the finite element method in order to simulate the failure modes in composite materials. T. Y. Kam & T. B. Jan [2] studied the first-ply failure of moderately thick laminated composite plates using the layerwise linear displacement theory. The accuracy of the finite element in predicting displacements and first-ply failure loads of laminated structures was verified by comparing results with experimental data and previously obtained analytical results. Since then, there have been numerous literatures regarding the progressive failure analysis of composite laminated plates and shells [3–7]. Liu and Zheng [8] introduced damage model to predict the progressive failure properties of the3D composite cylindrical laminates. 3D finite element technique was developed to investigate the non-linear stress–strain relationships and the final failure strengths of composite structures. The results were compared with those obtained from experiments and other existing models. Recently, Zahari, A. El-Zafrany [9] developed a progressive damage analysis methodology for stress analysis of composite laminated plates using new derivations of finite strip methods based on Mindlin's plate-bending theory.

From the literature review, the following points can be concluded.

- The finite element formulation was considered based on infinitesimal strain.
- Numerical case studies was reported and validated with commercial software only which concerns the analysis.
- There is a need for a flexible numerical tool, which can be used in assessing the composite failure analysis accurately and efficiently.
- Although, there is very little published work on the use of the finite element method for failure analysis of composite materials, most of that work is based on the current versions of commercial packages (ANSYS, ABAQUS, …etc.), which means that the methodology followed has to be changed corresponding to any new version. Furthermore, it is restricted to a certain accuracy of results, while in the failure analysis one may need results of six or more digits.

This paper presents the development of a progressive damage analysis methodology for stress analysis of composite laminated structures using high order finite element. It is an extension to the work developed by Moutaz [10 & 11] where the large effect force vector has been introduced separately to show the geometric nonlinear effect. A simple failure analysis technique has been to predict the mode of failure of composite laminated structures. This technique is performed based on the stress and strain values in longitudinal (fiber direction) and transverse (matrix direction) directions of the composite. The mode of failure has been detected and defined at each increment based on the value of stress and strain in fiber and matrix directions. The proposed technique and finite element derivation have been validated by comparing the obtained results with published results and with results obtained by ANSYS-12 commercial package for the same case studies.

2. Progressive Failure Analysis

As mentioned in the literature, many analyses have been developed to predict the failure of unidirectional composite laminates. Due to some complexities and restrictions of those types of analyses, a simple technique is used in this work to predict the failure of composite laminates. This technique is based on ultimate strain values of the laminate in fiber and matrix direction. The ultimate strain values of the fiber ε_{uf} and matrix ε_{um} can be obtained experimentally by applying the maximum possible load on a unidirectional ply with [0^o], on-axes, and [90^o], off-axes, respectively, and measuring the corresponding strain values [1]. A progressive failure methodology is developed for predicting the failure of laminate composite structures under incremental static load taking into consideration the effect of geometrical nonlinearity. Procedures and methods for the progressive failure analysis have been developed and are illustrated in Fig. 1 The main objects in that assessment are the material degradation method, which will be discussed in the next section.

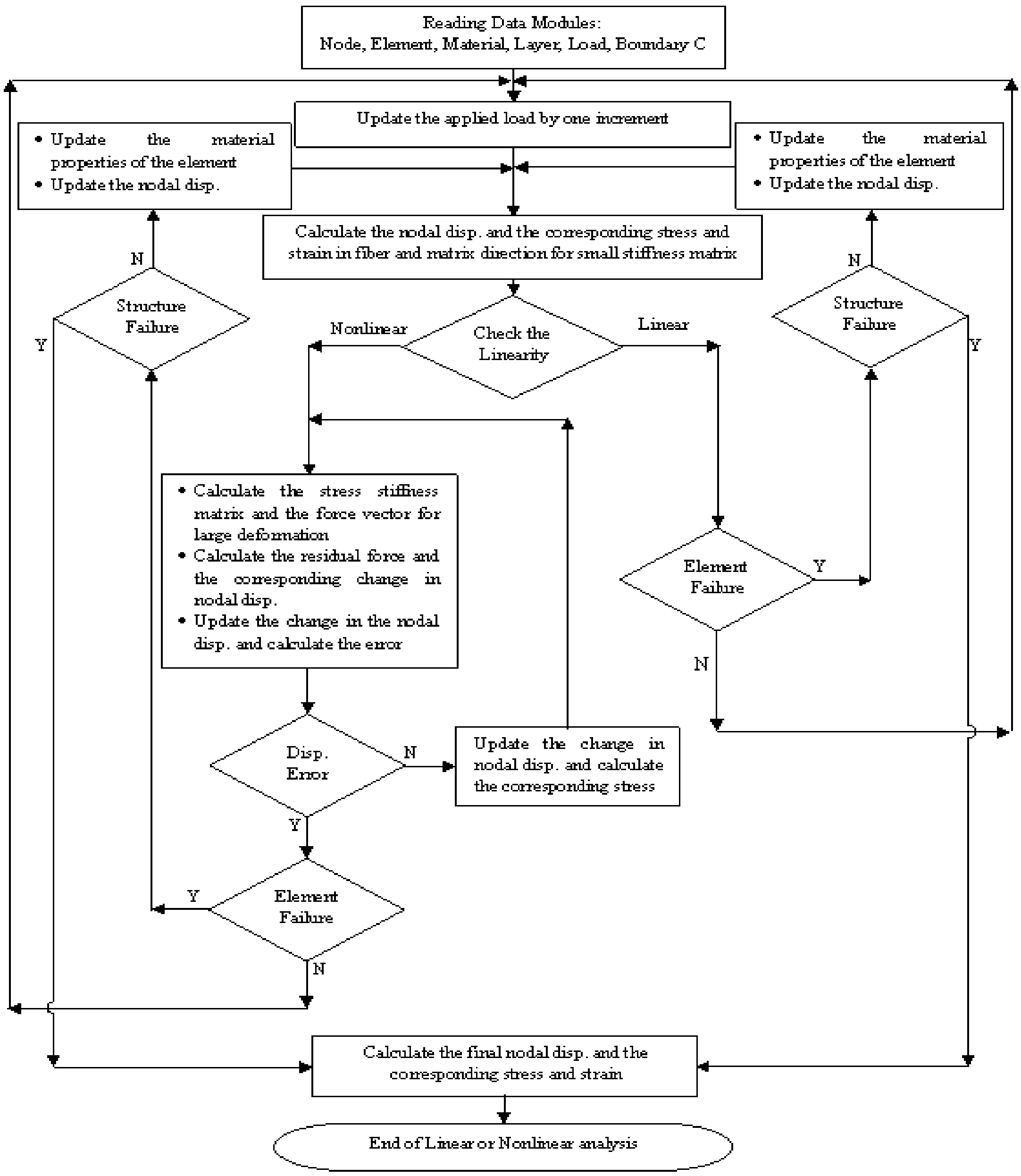


Fig. 1 Progressive failure algorithm

The failure analysis is performed based on some assumptions:

- A complete failure will occur when the strain value in the fiber direction at any element ε_f reaches the ultimate strain value of the fiber ε_{uf}.

  $\varepsilon_f \geq \varepsilon_{uf}$ (Complete failure)

- While the strain value in the matrix direction ε_m reaches its ultimate value (ultimate strain of the matrix ε_{um}), this implies that matrix cracking has occurred and crack growth will develop. At this stage the material degradation process will take place.

  $\varepsilon_m \geq \varepsilon_{um}$ (Failure or crack initiation)

where ε_{um} represents the ultimate strain of the matrix beyond which the degradation process reaches its ultimate in matrix.

3. Material Degradation

Material degradation is the reduction of the material properties of the structure due to certain type of applied load. The material degradation rules of composite materials are mainly based on experimental data. Due to the difficulty of measurement process, a theoretical solution has been introduced based on some assumptions and on the available literatures. For a laminated composite under incremental static loading conditions, in the first increments, the strength of the plies can be higher than the applied stress. Therefore, during the first increments, there is no failure mode that can be detected. Once the number of increments increases and the failure takes place in the material, the material properties at the damaged area have to be degraded by a set of degradation rules that based on some factors, such as the mode of failure, the original properties, the stress-strain state, the strength of the material, …etc. That type of degradation is called gradual material properties degradation. As the density of damaged area increases, the stiffness of the laminate decreases. In the matrix mode of failure, the degradation process of the mechanical properties is based on several reduction factors [1].

As soon as the failure condition of the matrix has been satisfied with in the laminate, the degradation process of the material properties have to take place. Consequently, the damaged area will be increased up to a critical value which is the damaged area of the laminate corresponding to the failure condition of the fiber.

Based on the literature, some assumptions are taken into consideration during the material degradation process, [2]:

- The material properties are gradually degraded.
- The material degradation will take place based on the state of strain of the matrix and fiber.

Hence, the new material properties will take place through the damaged area (element) which will be replaced with a one having the following properties

$$(E_1)_{new} = C_f (E_1)_{old} \qquad (G_{12})_{new} = C_m (G_{12})_{old}$$

$$(E_2)_{new} = C_m (E_2)_{old} \qquad (\nu_{12})_{new} = C_f (\nu_{12})_{old}$$

where C_f, C_m represent the material degradation factors. Based on the previous assumptions and some experimental data, they can be approximated as follows:

$$C_f = \left(1 - \frac{\varepsilon_f}{\varepsilon_{uf}}\right) \qquad C_m = \left(1 - \frac{\varepsilon_m}{\varepsilon_{um}}\right)$$

where the maximum values of the modulus reduction factor at the start of the applied stress will be unity as there is no damaged area (element) at this stage.

4. Finite Element Theory and Incremental Static Analysis

The proposed finite element theory, used in these analyses, has been discussed earlier by the author [10 & 11]. The finite element is based on high order shear deformation theory. The theory includes the formulation of the displacement equations, strain equation, stress equation and the strain energy variation which can be divided into small, large and coupled strain energies corresponding to which the element stiffness matrices can be obtained. The change of strain energy density can be written as follows:

$$dU = dU_S + dU_L + dU_{SL}$$

where dU_S contains infinitesimal strain effect and dU_L, dU_{SL} contain finite strain effect

This can be rewritten in terms of nodal displacements as follows;

$$dU = d\underline{\delta}^T \underline{K}\underline{\delta} + d\underline{\delta}^T \underline{K}^\sigma \underline{\delta} + d\underline{\delta}^T \underline{F}_L$$

where

$\underline{K}$ represents the infinitesimal stiffness matrix

$\underline{K}^\sigma$ represents the stress stiffness matrix

$\underline{F}_L$ represents the element force vector due to coupling effect

An equivalent nodal force $\underline{F}$ can be defined such that the change of work done by it due to a virtual displacement field is equivalent to the change in the strain energy, i.e.

$$dU = dW = d\underline{\delta}^T \underline{F}$$

Hence, $\underline{F} = \underline{K}\underline{\delta} + \underline{K}^\sigma \underline{\delta} + \underline{F}_L$

The principle of virtual work has been applied to obtain the generalized equations of equilibrium. The nodal force $\underline{F}$ can be divided into “N” increments of nodal force $\Delta\underline{F}$. The analysis can be carried out at each increment to obtain the global nodal displacement for linear or nonlinear case.

The generalized equations of equilibrium in the linear analysis can be represented based on the infinitesimal stiffness matrix as: $\underline{F} = \underline{K} \; \underline{\delta}$

At each nodal incremental force, $\Delta\underline{F}_i$, the change of nodal displacement can be calculated as:

$$\Delta\underline{F}_i = \underline{K} \; \Delta\underline{\delta}_i$$

The nodal displacement vector after “i” increment can be calculated as follows:

$$\underline{\delta}_i = \underline{\delta}_{i-1} + \Delta\underline{\delta}_i$$

While, equation of equilibrium in the nonlinear analysis can be represented as;

$$\left(\underline{K}+\underline{K}^{\sigma}\right)\underline{\delta}+\ \underline{F}_{L}\ -\ \underline{F}=0$$

Let $\underline{\delta}+\Delta\underline{\delta}$ represent the exact solution of the equilibrium equation, and then the residual force can be derived as; $\left(\underline{K}+\underline{K}^{\sigma}\right)\ \Delta\underline{\delta}\ =\underline{R}$

where the residual nodal force vector $\underline{R}$ can be defined as $\underline{R}\ =\ \underline{F}\ -\ \left(\underline{K}+\underline{K}^{\sigma}\right)\ \underline{\delta}\ -\ \underline{F}_{L}$

At each nodal incremental force $\Delta\underline{F}_i$ the nodal displacement can be solved by means of iterative algorithm until acceptable of error as follows:

1- Incremental force $\Delta\underline{F}_i$ the initial change of nodal displacement can be calculated according to the linear relation $\Delta\underline{F}_i\ =\ \underline{K}\ \ \Delta\underline{\delta}_i$

2- Update the nodal displacement after " i " increment, $\underline{\delta}_i\ =\ \underline{\delta}_{i-1}\ +\Delta\,\underline{\delta}_i$

3- Calculate the corresponding stress stiffness matrix and load force vector $\underline{K}_i^{\sigma}=\underline{\mathrm{K}}^{\sigma}(\underline{\sigma}_i)$ and $\underline{F}_{\mathrm{L}}^{\mathrm{i}}=\underline{\mathrm{F}}_L(\underline{\sigma}_i)$

4- Calculate the change of residual force vector, $\Delta\underline{R}\ =\ \underline{F}_i\ -\ \left(\underline{K}+\underline{K}_i^{\sigma}\right)\ \underline{\delta}_i\ -\ \underline{F}_L^{i}$

5- Obtain the change of incremental displacement, $\left(\underline{K}+\underline{K}_i^{\sigma}\right)\ \Delta\Delta\underline{\delta}_i\ =\ \Delta\underline{R}_i$

6- Update the change of nodal displacement $\Delta\underline{\delta}_i\ =\ \Delta\,\underline{\delta}_{i-1}\ +\Delta\Delta\,\underline{\delta}_i$

7- Calculate the value of the error: $error=\dfrac{\left|\Delta\underline{\delta}_i-\Delta\underline{\delta}_{i-1}\right|}{\left|\Delta\underline{\delta}_{i-1}\right|}$

8- If the error is greater than a certain acceptable value then repeat the procedure from step 2 using $\Delta\underline{\delta}_i$ instead of $\Delta\,\underline{\delta}_{i-1}$ until an acceptable value of error.

9- Update the global nodal displacement $\underline{\delta}_i\ =\ \underline{\delta}_{i-1}\ +\Delta\,\underline{\delta}_i$

5. Validation and Numerical Case Study

The validation process has been carried out to illustrate the accuracy of the proposed failure algorithm and the finite element package with different case studies. The validation has been performed by comparing the results of the proposed failure algorithm with corresponding results from a commercial finite element package ANSYS-12 and with theoretical and experimental published results while the failure analysis using ANSYS has been achieved using the Tsai–Wu failure theory [12].

5.1 Clamped Square Plate

A four-layer clamped square plate with 100 span-to-thickness ratio was employed to carry out the incremental static analysis. Two different types of mesh have been applied with this study, a coarse mesh (100 4-node elements) and a fine mesh (400 4-node elements) to study the mesh convergence. The plate is clamped at all four edges and the history of maximum deflection at the centre with the load is represented by a non-dimensional deflection parameter $\bar{w}$ and a load parameter $\bar{q}$ as follows [13]:

$$\overline{W} = \frac{w_c}{t} \qquad \overline{q} = q_o \left(\frac{L}{t}\right)^4 10^{-1} / E_2$$

where w_c is the central deflection and t is the thickness of the plate. A quarter-plate mesh has been used due to the symmetry of the plate. Different types of stacking sequence of the fiber; cross-ply [$0^o/90^o/0^o/90^o$] and angle-ply [$45^o/-45^o/45^o/-45^o$] have been used. The plate is subjected to uniform distributed load. Table 1 demonstrates the central non-dimensional deflection parameter history of the angle-ply and the cross-ply plates with the transverse load parameter.

It is clear that the proposed element gives good results compared with ANSYS and published results by Reddy [13] for the two different types of stacking sequences.

It can also be seen from the shown results that the proposed element provides very close results from the published and ANSYS results. It can also be seen from the shown results that the proposed element provides very close results to each other with the two types of meshes.

Table 1. Central non-dimensional deflection parameter with the load parameter

	Load Para.	4-node (coarse)	4-node (fine)	Reddy [13]	ANSYS (coarse)	ANSYS (fine)
angle-ply [$45^o/-45^o/45^o/-45^o$]	50	0.4659	0.4677	0.46	0.455	0.456
	100	0.7664	0.7693	0.75	0.746	0.746
	150	0.9803	0.9841	0.95	0.955	0.955
	200	1.148	1.152	1.14	1.121	1.12
	250	1.286	1.291	1.27	1.259	1.258
	300	1.406	1.411	1.39	1.378	1.377
	350	1.511	1.517	1.49	1.484	1.482
	400	1.605	1.612	1.59	1.58	1.577
	450	1.692	1.698	1.68	1.667	1.665
cross-ply [$0^o/90^o/0^o/90^o$]	50	0.4949	0.4949	0.5	0.497	0.497
	100	0.797	0.7981	0.82	0.803	0.802
	150	1.004	1.006	1.03	1.013	1.011
	200	1.163	1.165	1.2	1.175	1.171
	250	1.293	1.296	1.34	1.307	1.303
	300	1.404	1.408	1.46	1.42	1.416
	350	1.501	1.505	1.56	1.52	1.515
	400	1.588	1.593	1.64	1.609	1.603
	450	1.667	1.673	1.72	1.689	1.684

5.2 Failure Analysis of a Clamped Square Plate

Numerical and experimental validation of the proposed failure analysis was carried out with the same case study published by Kam & Jan [2] to ensure the accuracy of the finite element program with the proposed failure algorithm. Clamped square laminated composite plates with two different stacking sequences have been analyzed using the proposed finite element package. The plates were made of Graphite/Epoxy with the following material and geometric properties [2]; E_1 = 142.5 Gpa, E_2 = 9.79 Gpa, $G_{12} = G_{13}$ = 4.72 Gpa, G_{23} = 1.192 Gpa, ν_{12} = 0.27, plate length = 100 mm, layer thickness = 0.155 mm, number of layers = 16 and 32 with stacking sequence $[0_4/90_4]_s$ and $[0_8/90_8]_s$ and with strength properties as; X_t =2193.5 Mpa, X_c

= 2457.0 Mpa, Y_t = 41.3 Mpa, Y_c = 206.8 Mpa. The plates were subjected to centrally applied load to the first ply failure.

Table 2 shows a comparison between the results obtained by the proposed finite element program, with and without applying the proposed failure algorithm and material degradation, and the published experimental and analytical results by Kam & Jan [2]. The results show the central deflection due to 1000 N applied central load and the first-ply failure load for the 32-layer plate and the first-ply failure load for the 16-layer plate using the same 4x4 mesh for a quarter of the plate due to its symmetry. The load has been applied gradually on the laminated plates until the first-ply failure in the fiber appear. In general, the results obtained by the proposed finite element program are in good agreement with those obtained by Kam & Jan [2] experimentally and analytically. The main observation from the results is that the applying of the material degradation process increases the deflection and the failure load due to decrease of stiffness and applying the degradation process on the strength properties as well, respectively.

Table 2. Central deflection and first-ply failure loads for clamped square plates

		Without Mat. Deg.		With Mat. Deg.		Kam & Jan [2]	
		Lin.	Nonlin.	Lin.	Nonlin.	Analy.	Exper.
C. deflection [mm]	$[0_8/90_8]_s$	0.2365	0.2362	0.2381	0.2379	0.212	0.24
FPF Load [N]	$[0_4/90_4]_s$	672.96	683.34	685.31	689.67	696.2	647.0
	$[0_8/90_8]_s$	2220.72	2239.10	2182.52	2189.23	2207.07	2136.0

5.3 Progressive failure analysis of a plate with hole under in-plane tensile load

A 20 kN tensile load has been applied incrementally on a square plate with hole at different diameter to width ratio. The plate is fabricated from Graphite/Epoxy composite material with the same material and strength properties used with the clamped plate [2]. The effects of stacking sequence and the plate thickness have been investigated using different stacking sequences as follows; $[0^o/90^o]$, $[0^o/90^o]_s$, $[45^o/-45^o]$ and $[45^o/-45^o]_s$. The geometry of the plate and the applied load are shown in Fig. 2. A 4-node quadrilateral element has been used with a suitable mesh as shown in Fig. 2, where a fine mesh is used near the hole as it is the stress concentration zone. Due to the symmetric property of the plate, one quarter of the plate has been selected for the finite element analysis.

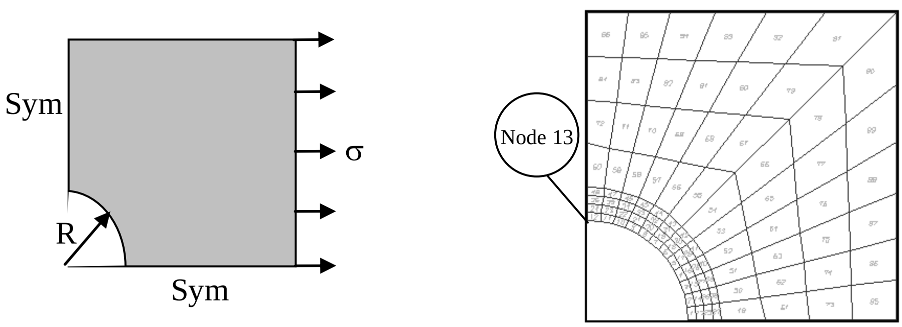


Fig. 2 Mesh and geometry of a square plate with hole under tensile load

Twenty increments load have been applied during the stress analysis. Each increment has been carried out with new material and strength properties for the damaged area as mentioned in the material degradation process.

Table 3 shows the deflection in y-direction at node 13 and the expected failure loads for matrix and fiber of plates with hole at different diameter to width ratios and at different stacking sequences. The results illustrate that a good prediction of the proposed failure algorithm for the start load of failure compared with ANSYS results for different stacking sequences and width to diameter ratios. Slight variations in the results between the ANSYS results and the Finite element program results may due to the slight difference in the damage modeling.

Figure 3 (a-b) shows the load increment versus damaged area in matrix and fiber for the plate of 10 width to diameter ratio for different stacking sequences and different number of layers. The main observations of these analyses are the damage occurred firstly in the 90^o plies at the elements within the stress concentration zone. The damaged area in fiber and matrix increase by decrease the number of layers. The plates with [45^o/-45^o] stacking sequence have a good response to failure than the [0^o/90^o] in the in-plane tension load.

Figure 4 shows the progress and the behavior of the failure in matrix and fiber for the [0^o/90^o] plate of 10 width to diameter ratio. The figures illustrate the damage in matrix and fiber with in 0^o and 90^o layers. The main observation is that, the failure start firstly in the matrix within the 90^o layers then in 0^o layers. The failure starts in the fiber within the 0^o layers firstly then the failure start at the last three increments in 90^o layers for one element only.

6. Conclusions

The present work contributes to the development of progressive failure algorithm of composite laminated structures using high order finite element. The proposed technique and finite element derivation have been validated by comparing the obtained results with published results and with results obtained by ANSYS commercial package for the same case studies. Good comparison with the finite element results ANSYS were observed from previous test cases, confirming the accuracy and reliability of the new derivations, damage algorithm and the programming package.

Table 3. Displacement in y-direction at node number 13 and the start load of failure

width/Radius	Stacking sequence	PFE		ANSYS-12 (SHELL99)	
		Disp. in y-direction	Start load of Failure	Disp. In y-direction	Failure Load
100/10 = 10	[45/-45]	-0.6887	7 kN	-0.6321	7.45
	$[45/-45]_s$	-0.3106	No Failure	-0.3008	No Failure
	[0/90]	-0.09248	7 kN	-0.08959	7.12
	$[0/90]_s$	-0.0492	No Failure	-0.0422	No Failure
100/30 = 3.3	[45/-45]	-2.579	7 kN	-2.387	7.03
	$[45/-45]_s$	-1.187	No Failure	-1.087	No Failure
	[0/90]	-0.6678	5 kN	-0.6194	5.67
	$[0/90]_s$	-0.3207	11 kN	-0.2906	11.06
100/50 = 2	[45/-45]	-7.702	6 kN	-7.528	6.87
	$[45/-45]_s$	-3.479	15 kN	-3.278	15.13
	[0/90]	-2.854	2 kN	-2.635	2.11
	$[0/90]_s$	-1.368	7 kN	-1.226	7.39

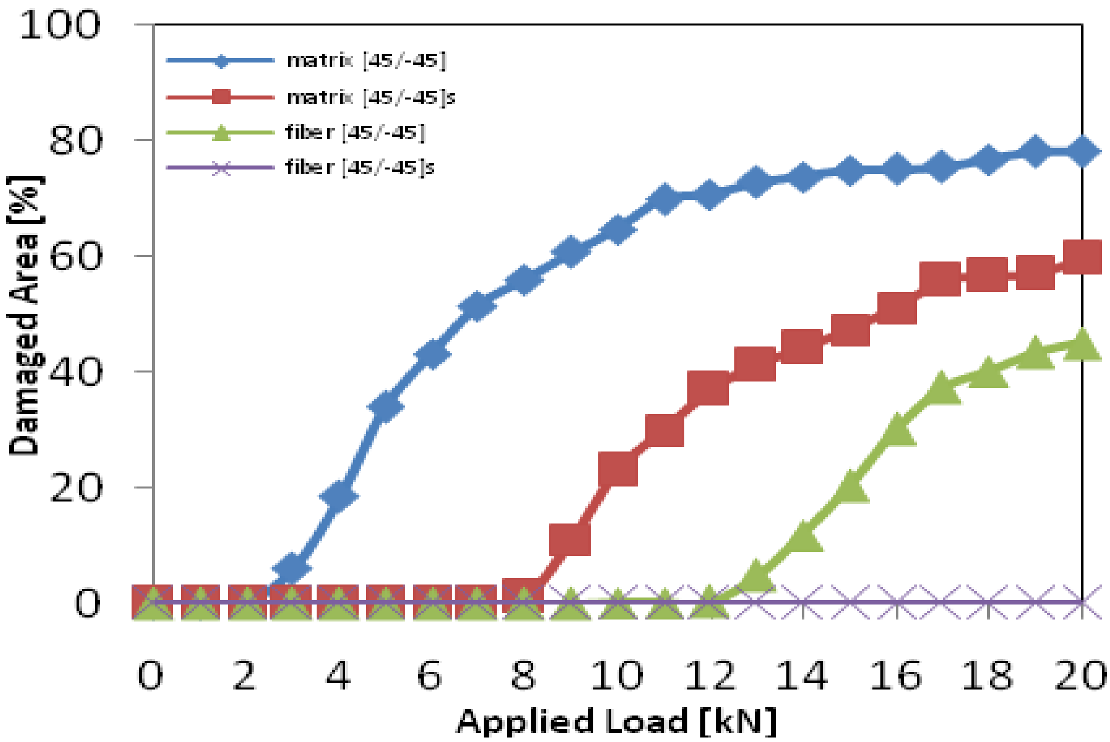


Fig. 3-a the damaged area of [45 ^{o} /-45^{o}] fiber and matrix vs. load increments

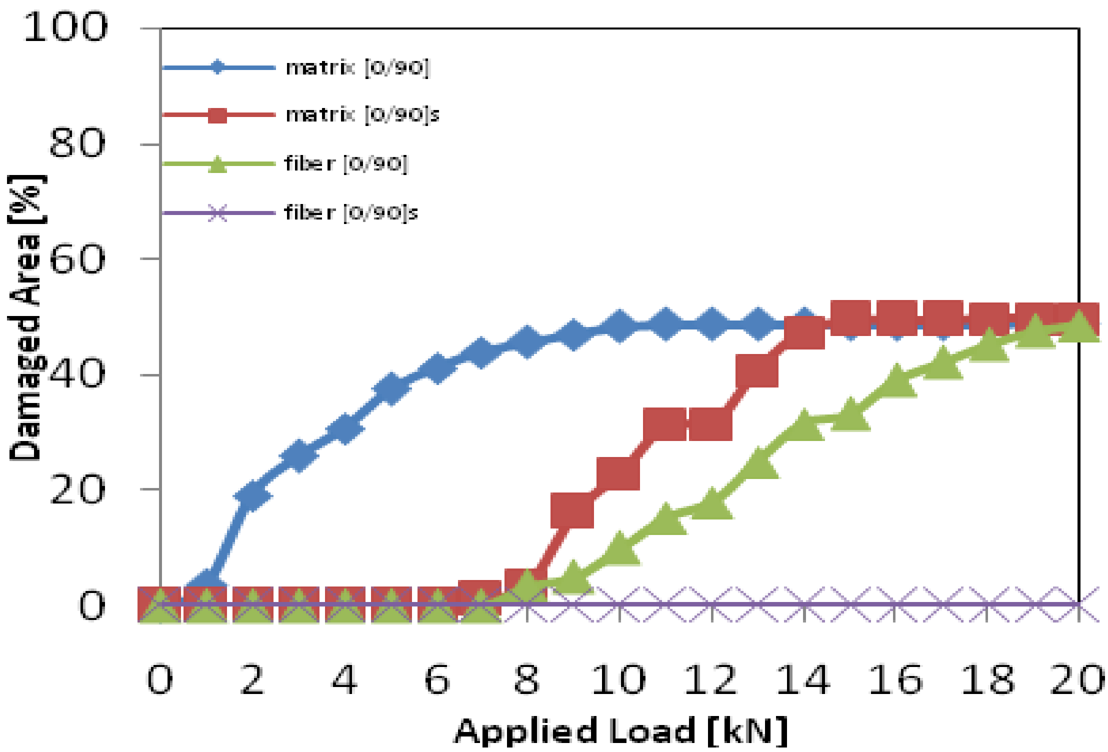
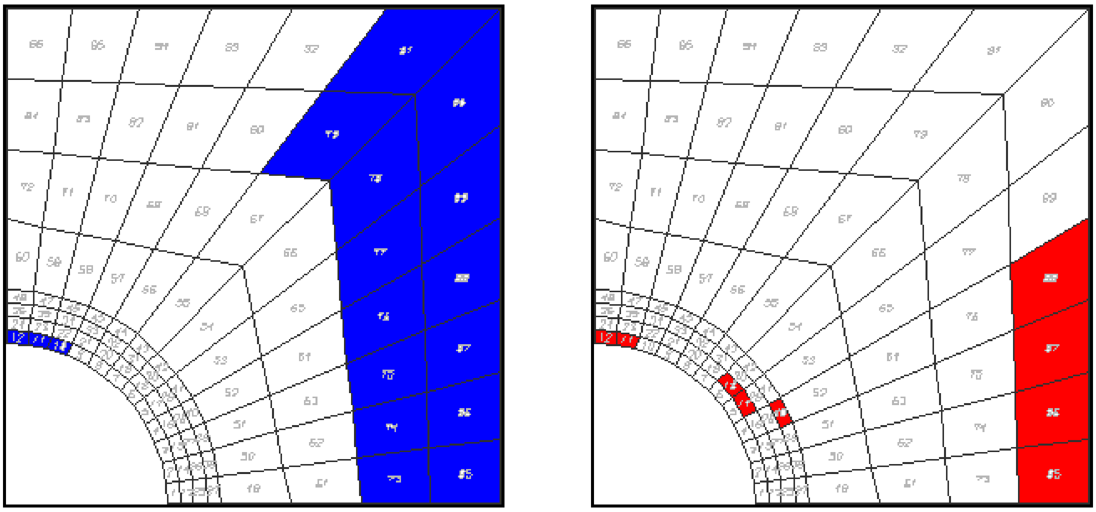


Fig. 3-b the damaged area of [0^{o}/-90^{o}] fiber and matrix vs. load increments



Damaged area in matrix within 90° layer start at increment number 1

Damaged area in fiber within 0° layer start at increment number 8

Figs. 4 Failure progress in matrix and fiber of the [0o/90o] plate.

7. References

[1] Carl O. (1982) “Fatigue design” A. Wheaton & Co. Ltd., Exeter, UK.

[2] T. Kam & T. Jan (1995) “First-ply failure analysis of laminated composite plates based on the layerwise linear displacement theory” Comp. Structures (32); 583-591

[3] Reddy YSN, Moorthy CMD & Reddy JN. (1995) “Non-linear progressive failure analysis of laminated composite plates” Int. J. Non-Linear Mech. (30):629–649.

[4] Marco D. S., Ugo I. & Michele V. (1998) “Failure analysis of composite laminates under large deflection” Composite Structures (40)- 3-4, 239-255.

[5] S. Michael & Anthony N. P. (2001) “progressive failure analysis of a composite shell” comsite structures (53); 117-131

[6] Yingshun Z., S. Wang & B. Petersson (2003) “Large deflection analysis of composite laminates” Journal of Materials Processing Technology (138); 34–40

[7] G. Akhras, W.C. Li (2007) “Progressive failure analysis of thick composite plates using the spline finite strip method” Composite Structures (79); 34–43

[8] P.F. Liu, J.Y. Zheng (2008) “Progressive failure analysis of carbon fiber/epoxy composite laminates using continuum damage mechanics” Materials Science and Engineering A 485; 711–717

[9] R. Zahar & A. El-Zafrany (2009) “Progressive failure analysis of composite laminated stiffened plates using the finite strip method” Comp. Structures 87; 63–70

[10] Moutaz M. Hegaze (2010). Nonlinear dynamic analysis of stiffened and unstiffened laminated composite plates using a high-order element, J. of Composite Materials, 44-(3):327-346.

[11] Moutaz M. Hegaze (2011). Geometrical Nonlinearity and Stability Analyses of Stiffened and Unstiffened Laminated Composite Plates Using High Order Element, ASAT 2011.

[12] Stephen W. Tsai. (1987) “Composites Design” 3rd Edition, Dayton, Ohio.

[13] Reddy J. N. (1982). Analysis of layered composite plates accounting for large deflections and transverse shear strains, Recent Advances in Non-Linear computational Mechanics By Hinton et al., University College of Swansea, U.K.