ISSN 2395-1621

Progressive Failure analysis of composite plate under mechanical load using FEA

[#1]Digvijay Vilas Ghadage, [#2]Prof.N.K.Chappkhane

[#1]Department of Mechanical Engineering
RIT Rajaramnagar
Islampur India
digvijayghadage@gmail.com

[#2]Department of Mechanical Engineering
RIT Rajaramnagar
Islampur India
narendra.chhapkhane@rit.edu.in

ABSTRACT

Applications of composite materials is dominant due to various favourable physical properties and high strength to weight ratio. The exact determination of modes of failure composite materials has been an important quest for reasearchers. A plane stress arterial model for composite material is presented. The material properties of composites are generalised as transversally isotropic material wherein only the plane stress conditions studied. The material behaviour is formulated in user defined sub routine UMAT. The subroutine is implemented a FEM software Abaqus/CAE for validation of UMAT. The obtained results are compared with FSDT and CPT theory for validation. A good agreement between numerical results and the theoretical results approves UMAT for further implementation in progressive failure analysis of composite plate.

Keywords-ABAQUS, FEA, UMAT, Composite Constitutive Material Model. Progressive Failure Analysis ,UMAT.

ARTICLE INFO

Article History

Received: 25th March 2017

Received in revised form :

25th March 2017

Accepted: 25th March 2017

Published online :

4th May 2017

I. INTRODUCTION

Among the other structural materials, composites have become popular because of their high strength and stiffness-to-weight ratio. As the enthusiasm for using composite materials in marine, aerospace and automobile structures is increasing, designers are trying to understand the damage mechanisms under compressive, tensile and combined loading i.e Thermal and Mechanical conditions and damage propagation and modes. For predicting the failure modes, damage propagation and failure loads in fibre-reinforced composite structures ,progressive failure analysis methodology has been implemented in finite element analysis codes.Numerous finite element programs such as ABAQUS ,NASTRAN and ANSYS are currently available for carrying out progressive failure analysis. Damage modes such as fiber breakage in tension and compression, matrix cracking and delamination are generally observed in composite structures. Fiber breakage and matrix cracking are called as intra-laminar damage and delamination damage mode is called as called inter-laminar damage. These intra-laminar and inter-laminar damages can lead to major strength reduction in the post-damage performance of the structure. In general, A laminate failure may not be catastrophic under thermal, environmental and mechanical loads. The layer which fails first is called as the first ply failure (FPF) and that the composite continues to take more loads until all the plies fail, the layer which fails at last is called as the Last ply failure (LPF). Failed plies may still contribute to the stiffness and strength of the laminate. The degradation of the stiffness and strength properties of each failed lamina depends on failure criteria followed by the designer. To predict the failure of the laminated composites, there are two categories of failure criteria

(i)Failure criteria associated with failure modes such as Tsai-Wu, Tsai-Hill, Modified Tsai- Wu and Hoffman
(ii) Failure criteria not associated with failure modes such as Maximum stress, Maximum strain, Hashin and Puck criteria.
After the stress distribution in the laminate has been determined, a failure criterion is used to determine if the laminate has failed at a certain point. To predict the failure load and failure propagation, failure criteria should be used in conjunction with progressive failure analysis. New

strength properties will be assigned failed elements according to the degradation rule and the failure mode until the final failure.

.

II. CONSTITUTIVE MODEL FOR COMPOSITES

The composite materials are generalized as transversely isotropic material. The constitutive models in a quasi-static stress analysis relate the state of strain to the state of stress. In performing point strain analyses, the state of stress at a point is desired and is readily computed once the point strains are determined through the use of the kinematics relations. For two-dimensional models, C^0 plate and shell kinematics models have five non-zero components which include transverse shear stresses through thickness and that for a C^1 plate and shell kinematics models have only three stress components and neglect transverse stresses.

For the C^0 plate model, the stress-strain relationship is given as below:

$$\{\sigma\} = \begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{12} \\ \sigma_{13} \\ \sigma_{23} \end{Bmatrix} = \begin{bmatrix} C_{11}^0 & C_{12}^0 & 0 & 0 & 0 \\ C_{21}^0 & C_{22}^0 & 0 & 0 & 0 \\ 0 & 0 & C_{44}^0 & 0 & 0 \\ 0 & 0 & 0 & C_{55}^0 & 0 \\ 0 & 0 & 0 & 0 & C_{66}^0 \end{bmatrix} \begin{Bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \gamma_{12} \\ \gamma_{13} \\ \gamma_{23} \end{Bmatrix} = [C^0]\{\varepsilon\}$$

Note that the order of the stress and strain terms is the convention used by ABAQUS.

The stiffness coefficients C_{ij}^0 in this matrix are defined in terms of the material elastic constants as given below:

$$C_{11}^0 = \frac{E_{11}}{1-\nu_{12}\nu_{21}};\; C_{22}^0 = \frac{E_{22}}{1-\nu_{12}\nu_{21}}$$

$$C_{12}^0 = C_{21}^0 = \frac{\nu_{12}E_{22}}{1-\nu_{12}\nu_{21}} = \frac{\nu_{21}E_{11}}{1-\nu_{12}\nu_{21}}$$

$$C_{44}^0 = G_{12};;C_{55}^0 = G_{13};C_{66}^0 = G_{23}$$

The relationships in above equation are manipulated in order to satisfy reciprocity equation for plane stress elasticity given

By

$$\nu_{21}E_{11} = \nu_{12}E_{22}$$

In the presented approach, the material is modeled implicitly through the thickness by using the kinematics assumptions of the two-dimensional shell elements (*i.e.*, in-plane strains vary linearly through the shell thickness and the transverse normal strain is zero). Hence the above equation can be further deduced for a C^1 plate model as given below:

$$\{\sigma\} = \begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{12} \end{Bmatrix} = \begin{bmatrix} C_{11}^0 & C_{12}^0 & 0 \\ C_{21}^0 & C_{22}^0 & 0 \\ 0 & 0 & C_{44}^0 \end{bmatrix} \begin{Bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \gamma_{12} \end{Bmatrix} = [C^0]\{\varepsilon\}$$

The above equation is used to simulate the behaviour of composite laminates as only two dimensional plane stress condition is implemented for the current work.

III. TRANSVERSE SHEAR STIFFNESS FOR SHELLS

Now that we are done with the constitutive material model, we need to deal with the transverse shear stress/stiffness for shell elements. In Abaqus/CAE, every node is further associated with material integration points. During derivation of solution, at each planar Gaussian integration point within each shell element, the material points through the entire thickness of laminate are evaluated to determine the through-the-thickness state of strain and to estimate the trial stresses (i.e., point stresses and strains). The transverse shear stiffness terms for shell elements are defined only once and only outside of the UMAT subroutine. For a homogeneous transversely anisotropic material, these values are defined as:

$$K_{11}^{ts} = \frac{5}{6}t \cdot G_{13};\; K_{22}^{ts} = \frac{5}{6}t \cdot G_{23};\; K_{12}^{ts} = 0$$

IV. FAILURE INITIATION CRITERIA

Most experimental determinations of the material strength are based on uni-axial stress states. However, in real composite structures, multi-axial stress states are present and failure under these multi-axial stress states should be predicted. To that purpose, many failure criteria have been developed, amongst which the Tsai-Wu quadratic failure criterion is widely used. Tsai and Wu developed an operationally simple failure criterion from strength tensors. The polynomial takes into account internal stresses which describe the difference between positive and negative stress-induced failures **Error! Reference source not found.**. He defined a scalar form of the strength criterion as a failure index defined as:

$$f(\sigma_k) = F_i\sigma_i + F_{ij}\sigma_i\sigma_j = 1$$

where *i*, *j*=1,2 and 6 for two dimensional plane stress case. Also F_i and F_{ij} are constant coefficients that depend on ultimate stress values of material. The quadratic terms $\sigma_i\sigma_j$ define an ellipsoid in the stress space. It is assumed that failure occurs when a stress vector reaches the failure surface. Inside the surface, no failure occurs and the material is elastic. The Tsai-Wu failure criteria for a plane-stress state is given by,

$$f(\sigma_{11},\sigma_{22},\tau_{12}) = F_1\sigma_{11} + F_2\sigma_{22} + F_{11}\sigma_{11}^2 + F_{22}\sigma_{22}^2 + F_{66}\tau_{12}^2 + 2F_{12}\sigma_{11}\sigma_{22} \le 1$$

where

$$F_1 = \frac{1}{X_T} - \frac{1}{X_C};\; F_{11} = \frac{1}{X_T X_C};\; F_{66} = \frac{1}{S_{12}^2}$$

$$F_2 = \frac{1}{Y_T} - \frac{1}{Y_C};F_{22} = \frac{1}{Y_T Y_C};F_{12} = -\frac{1}{2}\sqrt{F_{11}F_{22}}$$

In above equation (3.10), X_T, X_C are ultimate strengths in fiber-direction, Y_T, Y_C are ultimate strengths in transverse-fiber direction and S_{12} is the ultimate shear strength for the lamina. The value of F_{12} in above equation is as per the literature, but it actually is a curve-fitting factor. Hence the value of F_{12} is manipulated between -1 and 1 so that the stress ellipsoid matches with that obtained from experiment. For the current UMAT, the value of F_{12} is taken -1 because we obtain more accurate results with this value. This value is based on the strength values measured for unidirectional composite specimens and it is generally accepted that the influence of the F_{12} term is often negligible.

Tsai and Wu defined the strength ratio R as the scaling factor of the loading vector, and the failure index f as the inverse value of R ($f = 1/R$). The strength ratio is the positive root of the equation . Hence multiplying R on both sides we get,

$$R(F_1\sigma_{11} + F_2\sigma_{22}) + R^2(F_{11}\sigma_{11}^2 + F_{22}\sigma_{22}^2 + F_{66}\tau_{12}^2 + 2F_{12}\sigma_{11}\sigma_{22}) \geq 1$$

Converting above equation to equality, we get;

$$R^2 \cdot a + R \cdot b - 1 = 0$$

where

$$a = F_{11}\sigma_{11}^2 + F_{22}\sigma_{22}^2 + F_{66}\tau_{12}^2 + 2F_{12}\sigma_{11}\sigma_{22}$$
$$b = F_1\sigma_{11} + F_2\sigma_{22}$$

The solution to above quadratic equation is two roots out of which we need only the positive root. The positive root can be obtained as;

$$x = \left[\frac{-b+\sqrt{b^2-4ac}}{2a}\right]$$

substituting $c = -1$ we get;

$$R = \left[\frac{-b}{2a} + \sqrt{\left(\frac{b}{2a}\right)^2 + \left(\frac{1}{a}\right)}\right]$$

The strength ratio can be further used to obtain the failure index $f = 1/R$. This failure index is then compared to one (1) to define failure initiation *i.e.*, failure occurs when $f \geq 1$ .The values of Tsai-Wu failure are implemented to indicate that the stresses have needed the threshold strength of the laminate.It does not take any action of material degradation or material point elimination.It can only implemented to start or initiate msterial degradation or element deletion,Hence Tsai-Wu Failure criterion is called failure initiation criterion.

USER SUBROUTINE UMAT:

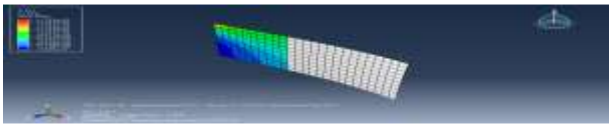
The ABAQUS/CAE is a nonlinear finite element analysis tool. Abaqus/CAE provides a complete interactive environment for creating Abaqus models, submitting and monitoring analysis jobs and viewing and manipulating simulation results. But the more distinguishing feature of ABAQUS is that it not only provides a library for materials, loading conditions, friction contacts, elements, flow and all other features, but also it has a provision to define a user defined entity if none of those given in ABAQUS are able to simulate the behaviors as expected by the user.

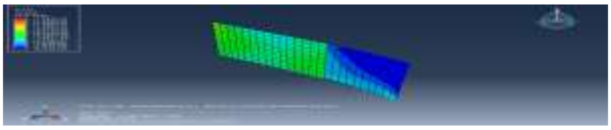
The analyst defines various steps to perform. Each step includes a specific load and boundary condition applied to the model.at each load step a non linear finite element analysis is perf0ormed. the strain increments are passed from ABAQUS to the UMAT i.e.to a fortran compiler, where in the behavior of material is encoded .As per the UMAT subroutine program, the strain increments are used to calculate stresses and other corresponding values for the model. The UMAT passes two entities at each material point to ABAQUS; one the stress at material point ,and two ,the Jacobian Matrix. These calculated stresses and Jacobian matrix are returned to abaqus at each material point. The failure indices are passed back to Abaqus at each material point.

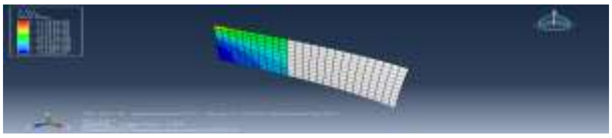
Numerical Result:

Model the laminate shown in figure 5.6.The laminate in section A is a [+45/-45/0/90/0].The thickness of each lmina is 1.2 mmA ply drop ratio 1:10 is used The strip is 120 mm long and 100 mm wide It is loaded by tension Nx=10N/mm applied to bottom edge on the strip Use symmetry to the model1/2 of the tape the material is AS4D/9310.visualise the report the maximum value of transverse deflection U3.Also visualize and tabulate the maximum value of principal stress at the top surface of every lamina(k1,k2,k3,k4,k5)Note the absence of results for the laminas dropped in the region where those lamina have been dropped.

Solution of Ply 1,2,3 are given below.



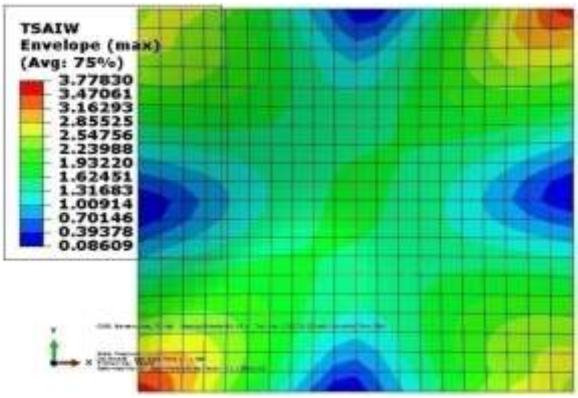




Example2: simply supported symmetric $[0/90/\pm45]_s$ square laminate plate of sides $a = b = 2000$mm and thickness $t = 10$ mm is subjected to a constant pressure of 0.1 MPa. The properties of the laminate are given in Table 6.2. The Tsai-Wu failure index is calculated using Abaqus in-built function and with UMAT. The results of the same are compared to correlate them. The transverse shear stiffness values are constant throughout the analysis with UMAT.

Material properties of unidirectional carbon/epoxy components:

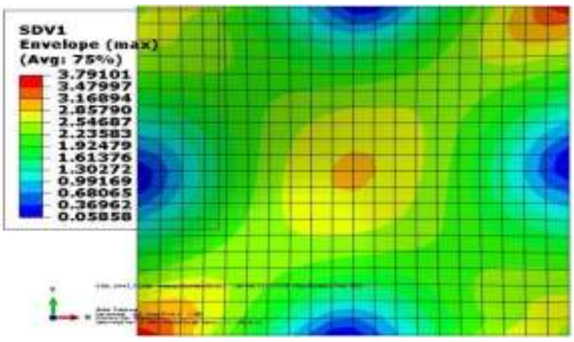
Property	Unit	AS4D/9310
E_1	[GPa]	133.86
$E_2=E_3$	[GPa]	7.706
$G_{12}=G_{13}$	[GPa]	4.306
G_{23}	[GPa]	2.76
$\bar{\upsilon}_{12}=\bar{\upsilon}_{13}$	------	0.301
$\bar{\upsilon}_{23}$	-------	0.396
X_T	[Mpa]	1830
X_C	[MPa]	1096
$Y_T=Z_T$	[MPa]	57
$Y_C=Z_C$	[MPa]	228
S_{12}	[MPa]	71
F_{12}	[MPa]	-1



Tsai-Wu Failure Index Plot using Abaqus in Built Funcion

The spectrum plots of the failure indices of the two plates, one calculated with Abaqus in-built function and other with the UMAT are shown in above. The maximum values of the failure indices differ with a very small error of 0.3363%. These results of UMAT concords to those obtained from Abaqus. A small difference or error between the two results can be attributed to various factors such as the transverse shear stiffness of the shell element.also the round offling numerical values contributes to the error.With the above two validated problem solutions,we advance the conclusion of the paperawards.

Conclusion:



As the values of the failure indices using UMAT and in-built feature concur with a very small and negligible error, the UMAT can be regarded good for implementation in progressive failure analysis. Hence the UMAT can now be implemented as a failure initiation criterion for PFA.

References

[1] Ambur D.R., Jaunky N., Hilburger M. W., Progressive failure studies of stiffened panels subjected to shear loading .Composite Structures 65 (2004)

[2]Goyal V.K., Jaunky N., Johnson E.R., Ambur D.R., Intralaminar and interlaminar progressive failure analyses of composite panels with circular cutouts. Composite Structures 64 (2004)

[3]Basu S., Waas A.M., Ambur D.R., Prediction of progressive failure in multidirectional composite laminated panels. International Journal of Solids and Structures 44 (2007)

[4]Fana X.L., Wang T.J., Sun Q., Damage evolution of sandwich composite structure using a progressive failure analysis methodology Procedia Engineering 10 (2011)

[5]Wang T.J., Callus P.J., Bannister M.K., Experimental and numerical investigation of the tension and compression strength of un-notched and notched quasi-isotropic laminates. Composite Structures 64 (2004)

[6] ABAQUS Analysis Users Manual 6.11 volume III, Materials. ABAQUS User subroutines reference Manual 6.10.