Technical University of Munich
Computational Mechanics Program
Software Lab 18/19

Software Lab Project Documentation

Group # 11

Project Title:

Development of The Failure Criteria for Composites

Group Members:

Ammar Khallouf

Gavallas Panagiotis

Yasuyuki Shimizu

Supervisors:

Ani Khaloian

Michael Richter

Prof. Fabian Duddeck

Date Submitted:

11-12-2019

Abstract:

Composite materials are a challenge to analyze. This challenge is characterized by complex anisotropic behavior of the material and its heterogenous microstructure. This in turn leads to complex failure and damage modes which are of primary concern for the safety and performance of composite structures design. In context of computational mechanics, the available material models in commercial nonlinear finite element analysis tools for modeling these new material forms are often lagging behind the material science developments. In this regard, this project aims to develop custom material subroutines for composite materials, the developed models predict the onset of damage and damage progression in composite structures according to wide range of failure theories and damage models that can be customized by users accordingly to fit their requirements. The developed material models are verified using number of benchmark problems from literature. Finally, realizing the challenges associated with writing user defined material models, a standalone code "PDALAC" is developed to help researchers to visualize the progressive damage analysis process without the need to conduct a FEA, the developed code is written in vectorized form which can be easily translated to work with multiple standard FEM packages (e.g. ABAQUS, ANSYS, LS-DYNA)

Table of Contents

1.Introduction

1.1 Overview and motivation:

Composite materials have proven their potential for use in high performance engineering applications over the last fifty years. High mass, specific stiffness, strength and energy absorption, high functionality (e.g. through tailored anisotropy), and optimized structural concepts (e.g. due to high levels of design integration) are the main reasons for specifying composites. Hence, composites offer product manufacturers several advantages in terms of weight and performance. Today, the benefits of components designed and produced in composite materials instead of metals are well recognized by many industries. However, composite materials also come with several challenges during product design when compared to normal materials such as metals. Delamination, micro-cracking leading to eventual failure, and other mechanisms, that are not factors in designing with metals, are very important for composites. In this context having comprehensive failure criteria for composites become an important point of concern, because these criteria should establish when the failure takes place and predict the post-failure behavior. The analysis and simulation of the failure of composite laminated structures are quite cumbersome tasks, due to the complex anisotropic behavior and differences in tensile and compressive strengths, the failure mechanisms are very different from those of traditional metallic structures where brittle and ductile failure modes are observed under different loading conditions. The current major finite elements software packages such as (ABAQUS, ANSYS, LS-DYNA) include some basic failure models for composites, such models are based on stress quadratic functionals (e.g. Tsai & Wu,1971) and imply failure once the maximum stress/strain limits of the material are exceeded. These models however are only capable of covering simple aspects of the complex materials in 3D, more advanced material models are therefore required to extend the basic models for more complex scenarios. In finite element analysis context, this can be achieved by utilizing the user programmable subroutines supported by software packages such as ABAQUS, LS-DYNA. The user material routine (UMAT) for example allows for the creation of user-defined materials with their own constitutive equations and failure criteria. As a result, this programming feature makes it an ideal tool for the development of composite materials failure models.

1.2. Project aim and objectives:

The aim of this project is to develop user defined material models within FEM framework that can capture more complex aspects of composite materials behavior and failure modes, this is achieved through the following suggested steps:

- Develop an orthotropic material model for composite laminate.
- Incorporate multiple failure theories to predict onset of damage.
- Incorporate multiple damage models to describe the post failure behavior.
- Integrate all developed failure theories and damage models in one material subroutine.

2.Theory Reference Manual

2.1 Review of composite materials mechanics:

Generally, composite materials can be derived by combining two or more materials to use the benefit of the characteristics of each material and especially their interaction. Thus, the properties of a composite are normally better than those of its individual constituents. Considering fiber reinforced plastics, the following distribution of the individual tasks can be given: the fiber provides the strength and stiffness, whereas the matrix gives rigidity and environmental resistance.

Separating the main mechanical functions of the fibers and the matrix in a composite structure provides the following functionalities. The fibers carry the main load while the matrix binds the fibers and transfers the load between the fibers. In addition, the matrix isolates the fibers, so that individual fibers can act separately which slows or even stops crack propagation Furthermore, the failure is strongly affected by the type of the used material, the fiber distribution and the applied load. Since a composite material can consist of multiple layers, failure may occur within a single ply (intralaminar fracture) or between two plies (interlaminar fracture). The latter is also known as delamination of the bonded plies. Furthermore, a fracture oriented transversely to a ply is called translaminar fracture

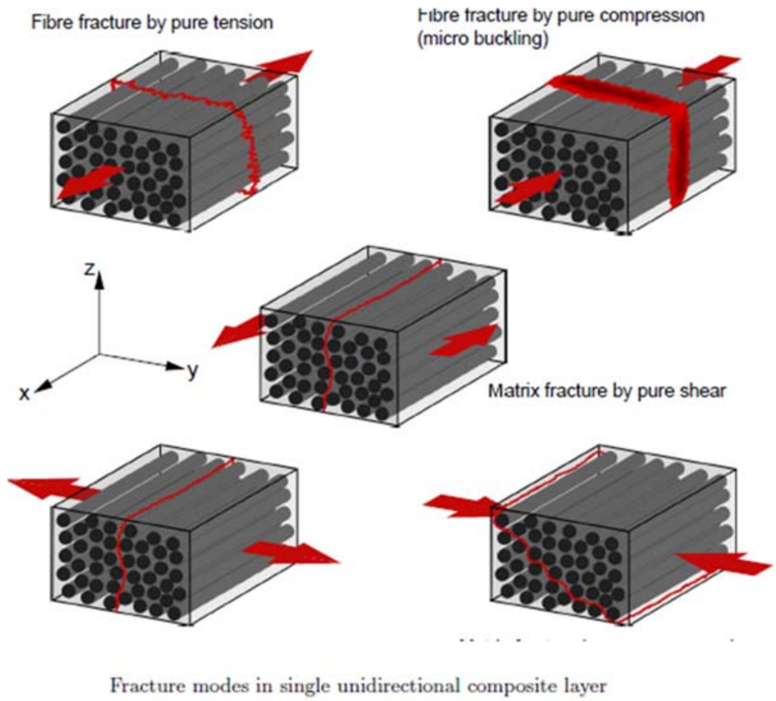


Fig.1 Fracture Modes UD composites (A, Berger 2014)

The main failure modes of laminated fiber-reinforced composites are the following:

- Delamination: the process through which composite materials made of different plies stacked together tend to delaminate. The bending stiffness of delaminated panels can be significantly reduced, even when no visual defect is visible on the surface or the free edges. The physics of delamination is quite understood, and one of the best numerical tools to predict the propagation of delamination consists in the use of Decohesion Elements. These elements have been developed and implemented in commercial Finite Element (FE) codes, like ABAQUS.
- Matrix compression failure: what is commonly considered as matrix compression failure is shear matrix failure. Indeed, the failure occurs at an angle with the loading direction, which is the evidence of the shear nature of the failure process.
- Matrix tensile failure: the fracture on the surface resulting from this failure mode is typically normal to the loading direction. Some fiber splitting at the fracture surface usually can be observed.
- Fiber compression failure: this failure mode is largely affected by the resin shear behavior and imperfections such as the initial fiber misalignment of the angle and voids. Typically,

kinking bands can be observed on a smaller scale, and they are the result of the fiber micro-buckling, matrix shear failure or fiber failure.

- Fiber tensile failure: this failure mode is explosive. It releases large amounts of energy. In structures that cannot redistribute the load, it typically causes catastrophic failure.

2.2 Failure Criteria:

Many different failure criteria have been formulated in order to predict failure loads for general stress states. In this text is proposed the following classification, in which they could be grouped in two main groups firstly:

1. Failure criteria neglecting interactions between different stress components.

2. Failure criteria considering interactions between different stress components.

Criteria belonging to the first group are the simplest ones and they usually propose one inequality for each one of the three in-plane stresses (or strain) components.

In the remaining criteria, the failure in one direction may be sensitive to loads along other directions (including shear).

This last group can be divided into the following two subgroups.

a. Criteria proposing one single inequality to define the failure envelope.

b. Criteria proposing a combination of interactive and non-interactive conditions.

In general, one more Failure Indexes (FI) corresponds to each failure criteria. A FI exceeding the unitary value means that failure occurs, according to the applied criterion.

Some useful definitions are reported to a better understanding of the following concepts:

- Failure indices: represent a phenomenological failure criterion in that only an occurrence of a failure is indicated and not the mode of failure.
- Strength ratio: is a more direct indicator of failure than the failure index since it demonstrates the percentage of applied load to the failure criteria. Strength ratio is defined as: Strength Ratio (SR) = Allowable Stress / Calculated Stress.

The following failure theories are briefly described below:

➢ Maximum Stress Criterion:

Related to the maximum normal stress theory by Rankine and the maximum shearing stress theory by Tresca, this theory is similar to those applied to isotropic materials. The stresses acting on a lamina are resolved into the normal and shear stresses in the local axes. Failure is predicted in a lamina, if any of the normal or shear stresses in the local axes of a lamina is equal to or exceeds the corresponding ultimate strengths of the unidirectional lamina.

This approach considers stress components separately and assumes that the failure would occur when any of the stress components reach the allowable strength in a corresponding direction. This criterion proposes following inequalities to examine failure.

$$\sigma_{11} \geq X_1^T \ , \qquad \sigma_{11} \leq -X_1^C \ , \qquad \sigma_{22} \geq X_2^T$$
$$\sigma_{22} \leq -X_2^C \ , \qquad \sigma_{12} \geq S \ , \qquad \sigma_{12} \leq -S$$

Where X is the allowable stress and subscripts 1, 2 corresponds to fiber and transverse directions, while superscripts T, C stands for tension and compression, respectively; and S is the shear allowable. According to the criteria, if any of these inequalities is satisfied, the failure would occur. This failure envelop is graphically represented below

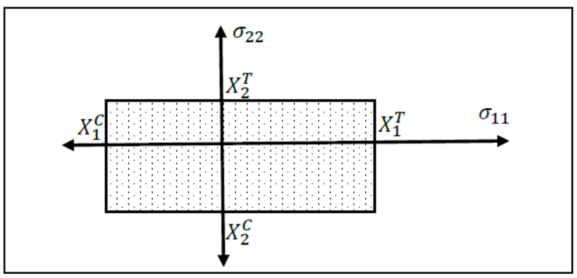


Fig.2 Failure envelope Maximum Stress Criterion (ATAR, M. 2016)

- Maximum Strain Criterion

This theory is based on the maximum normal strain theory by St. Venant and the maximum shear stress theory by Tresca as applied to isotropic materials. The strains applied to a lamina are resolved to strains in the local axes. Failure is predicted in a lamina, if any of the normal or shearing strains in the local axes of a lamina equal or exceed the corresponding ultimate strains of the unidirectional lamina. Similar to the maximum stress approach, allowable strains are utilized in this criterion. Failure criteria is given in terms of inequalities as follows:

$$\epsilon_{11} \geq e_1^T \ , \qquad \epsilon_{11} \leq -e_1^C \ , \qquad \epsilon_{22} \geq e_2^T$$
$$\epsilon_{22} \leq -e_2^C \ , \qquad \epsilon_{12} \geq e_{12} \ , \qquad \epsilon_{12} \leq -e_{12}$$

Where e is the allowable strain in the corresponding direction.

- Tsai-Hill Criterion:

This theory is based on the distortion energy failure theory of Von-Mises distortional energy yield criterion for isotropic materials as applied to anisotropic materials by Hill. Distortion energy is a part of the total strain energy in a body. The strain energy in a body consists of two parts; one due to a change in volume and is called the dilation energy and the second is due to a change in shape and is called the distortion energy. It is assumed that failure in the material takes place only when the distortion energy is greater than the failure distortion energy of the material. Then, Tsai adapted it to a unidirectional lamina. The criterion was simplified by assuming plane stress conditions (e.g. out of plane stresses are neglected). Tsai also claimed that material properties are equal in the direction transverse to the fibers, with these simplifications, failure criterion takes the following form:

$$\frac{1}{X^2}\sigma_1^2 - (\frac{1}{X^2})\sigma_1\sigma_2 + \frac{1}{Y^2}\sigma_2^2 + \frac{1}{S^2}\tau_{12}^2 = 1$$

The Tsai–Hill failure theory is a unified theory and thus does not give the mode of failure like the maximum stress and maximum strain failure theories do. The Tsai–Hill failure theory does not also distinguish between the compressive and tensile strengths in its equations. However, unlike the maximum strain and maximum stress failure theories, the Tsai–Hill failure theory considers the interaction among the three unidirectional lamina strength parameters.

- Tsai-Wu Criterion:

This failure theory is based on the total strain energy failure theory of Beltrami. Tsai-Wu applied the failure theory to a lamina in plane stress. This failure theory is more general than the Tsai–Hill failure theory because it distinguishes between the compressive and tensile strengths of a lamina. A lamina is considered to be failed if:

$$F_1\sigma_1 + F_2\sigma_2 + F_{11}\sigma_1^2 + F_{22}\sigma_2^2 + F_{66}\sigma_6^2 + 2F_{12}\sigma_1\sigma_2 = 1$$

$F1$ and $F11$ are obtained by tensile and compressive tests in longitudinal direction; on the other hand, $F2$ and $F22$ are obtained from tests in transverse direction. $F66$ can be obtained by calculating shear strength in shear test. As a result, the constants are:

$$F_1 = \frac{1}{X_1^T} - \frac{1}{X_1^C} \qquad F_{11} = \frac{1}{X_1^T X_1^C}$$

$$F_2 = \frac{1}{X_2^T} - \frac{1}{X_2^C} \qquad F_{22} = \frac{1}{X_2^T X_2^C}$$

$$F_{66} = \frac{1}{S^2}$$

The remaining constant, $F12$, must be evaluated by the biaxial test; however, it is a complicated test to conduct. As an alternative, Tsai and Hahn proposed that, $F12 = -0.5\sqrt{F11F22}$, can be used.

- Hoffman Criterion:

The Hoffman criterion is an extension of Tsai-Hill theory, this theory takes into account the difference in tensile and compressive allowable stresses by using linear terms in the equation. The resulting failure index in Hoffman's theory for an orthotropic lamina in a general state of plane stress (2D) with unequal tensile and compressive strengths is given by:

$$FI_{Hoffaman2D} = \left(\frac{1}{X_t} - \frac{1}{X_c}\right)\sigma_1 + \left(\frac{1}{Y_t} - \frac{1}{Y_c}\right)\sigma_2 + \frac{\sigma_1^2}{X_t X_c} + \frac{\sigma_2^2}{Y_t Y_c} + \frac{\sigma_{12}^2}{S^2} - \frac{\sigma_1 \sigma_2}{X_t X_c},$$

To calculate the strength ratio and then the margin of safety, the following terms are defined:

Hoffman's failure index (2D) coefficients	
$\boldsymbol{F_1 = \frac{1}{X_t} - \frac{1}{X_c}}$	$F_{22} = \frac{1}{Y_t Y_c}$
$\boldsymbol{F_2 = \frac{1}{Y_t} - \frac{1}{Y_c}}$	$F_{66} = \frac{1}{S^2}$
$\boldsymbol{F_{11} = \frac{1}{X_t X_c}}$	

Fig.3 Hoffman 2D failure coefficients (ATAR, M. 2016)

In case of composites modelled using solid elements, so for a 3D mesh and stress state, the relation of failure index becomes:

$$FI_{Hoff3D} = C_1(\sigma_2 - \sigma_3)^2 + C_2(\sigma_3 - \sigma_1)^2 + C_3(\sigma_1 - \sigma_2)^2 + C_4\sigma_1 + C_5\sigma_2 + C_6\sigma_3 + C_7\tau_{23}^2 + C_8\tau_{13}^2 + C_9\tau_{12}^2,$$

With the following failure index coefficients in 3D:

Hoffman's failure index (3D) coefficients	
$C_1 = \frac{1}{2}\left(\frac{1}{Z_t Z_c} + \frac{1}{Y_t Y_c} - \frac{1}{X_t X_c}\right)$	$C_6 = \left(\frac{1}{Z_t} - \frac{1}{Z_c}\right)$
$C_2 = \frac{1}{2}\left(\frac{1}{X_t X_c} + \frac{1}{Z_t Z_c} - \frac{1}{Y_t Y_c}\right)$	$C_7 = \frac{1}{S_{23}^2}$
$C_3 = \frac{1}{2}\left(\frac{1}{X_t X_c} + \frac{1}{Y_t Y_c} - \frac{1}{Z_t Z_c}\right)$	$C_8 = \frac{1}{S_{13}^2}$
$C_4 = \left(\frac{1}{X_t} - \frac{1}{X_c}\right)$	$C_9 = \frac{1}{S_{12}^2}$
$C_5 = \left(\frac{1}{Y_t} - \frac{1}{Y_c}\right)$	

Fig.4 Hoffman 3D failure coefficients (ATAR, M. 2016)

In each case, the following material data are required:

-Xt,Xc are the maximum allowable stresses in the 1-direction in tension and compression;

- Yt,Yc are the maximum allowable stresses in the 2-direction in tension and compression;

-Zt,Zc are the maximum allowable stresses in the 3-direction in tension and compression;

-$S12$ is the maximum allowable in-plane shear stress;

-$S23$ is the maximum allowable 23 shear stress;

-$S13$ is the maximum allowable 13 shear stress.

- Hashin Criterion:

  In 1980, Hashin made further studies on Tsai-Wu Criterion. He implies that Tsai-Wu Criterion had improvements over previous ones, and it provided good fit with test data. However, Hashin also proposed that different failure modes can occur, and a single quadratic function may not predict all of failure modes. On the other hand, proposing a failure criterion with higher than a quadratic degree would make things more complicated. As different from previous studies, Hashin examined fiber and matrix failure separately. By that way, instead of continuous failure surface a piece-wise continuous failure one is obtained. The Hashin criterion proposes a combination of four interactive and non-interactive conditions in order to distinguish between matrix or fiber failure caused by tension or compression. The conditions for failure being given by the following inequalities:

a) Matrix Failure for compression ($\sigma 22 \geq 0$)

$$\left(\frac{\sigma_{22}}{Y^T}\right)^2 + \left(\frac{\tau_{12}}{S^L}\right)^2 \geq 1$$

b) Matrix failure for tension ($\sigma 22<0$)

$$\left(\frac{\sigma_{22}}{2S^T}\right)^2 + \frac{\left[\left(\frac{Y^C}{2S^T}\right)^2 - 1\right]\sigma_{22}}{Y^C} + \left(\frac{\tau_{12}}{S^L}\right)^2 \geq 1$$

c) Fiber failure for tension ($\sigma 11 \geq 0$)

$$\left(\frac{\sigma_{11}}{X^T}\right)^2 + \left(\frac{\tau_{12}}{S^L}\right)^2 \geq 1$$

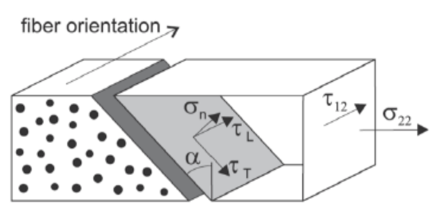
d) Fiber failure for compression ($\sigma 11<0$)

$$-\frac{\sigma_{11}}{X^C} \geq 1$$

In formula *b)* the term *ST* is the transverse shear strengths which is very difficult to measure experimentally. An analytic relation is suggested:

$$S^T = Y^C \cos(\alpha)\,(\sin(\alpha) + \frac{\cos(\alpha)}{\tan(2\alpha)}$$

where α is the angle of fracture plane as graphically depicted in the figure below:



Angle of fracture plane of an unidirectional lamina subjected to transverse compression and in-plane shear

Fig.5 Fracture plane angle for UD composites (ATAR, M. 2016)

- Puck Failure Theory:

Puck Failure Criterion is based on Mohr 's fracture hypothesis which is appropriate for brittle fracture behavior of composite materials. It can distinguish between fiber fracture and different inter-fiber fracture. Both 2D and 3D formulations are implemented Available fracture modes for 2D Puck Criterion are:

- Fiber Fracture (FF)
- Inter Fiber Fracture Mode A (IFF A)
- Inter Fiber Fracture Mode B (IFF B)
- Inter Fiber Fracture Mode C (IFF C)

Available fracture modes for 3D Puck Criterion are:

- Fiber Fracture (FF)
- Inter Fiber Fracture (IFF)

The stresses on the fracture plane for the puck criterion are calculated according to the following relations [1]:

$$\sigma_n(\theta) = \sigma_2 \cdot \cos^2\theta + \sigma_3 \cdot \sin^2\theta + 2 \cdot \tau_{23} \cdot \sin\theta \cdot \cos\theta$$

$$\tau_{nt}(\theta) = -\sigma_2 \cdot \sin\theta\cos\theta + \tau_{23} \cdot \left(\cos^2\theta - \sin^2\theta\right)$$

$$\tau_{n1}(\theta) = \tau_{31} \cdot \sin(\theta) + \tau_{21} \cdot \cos(\theta)$$

$$\tau_{n\psi}(\theta) = \sqrt{\tau_{nt}^2 + \tau_{n1}^2}$$

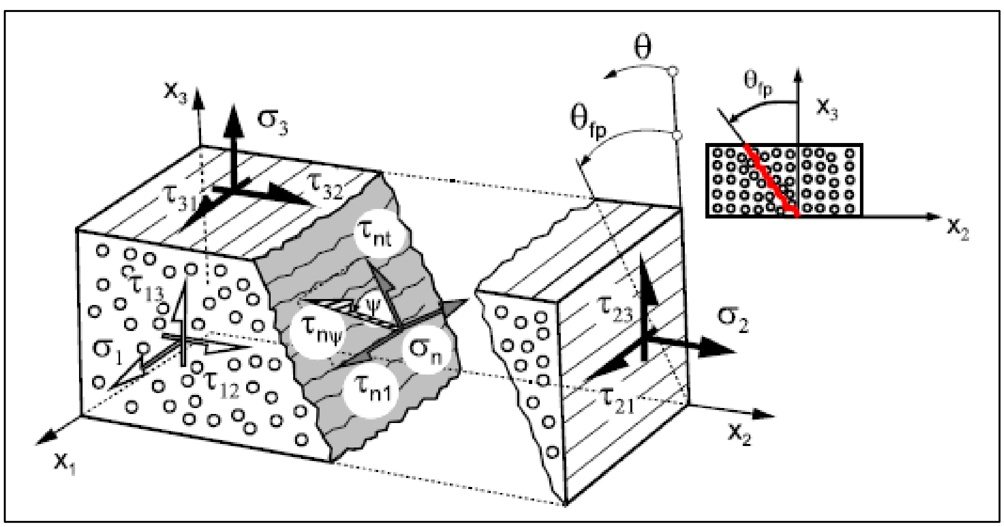


Fig.6 Stresses of the action plane (Puck & Shurmann,1998)

In order to judge if a stress vector on the stress space is leading to damage, a mathematical expression is needed. This expression is called fracture condition and is written in the following general form:

$$\boxed{F(\sigma_i, R_i)_{i=1\ to\ n} = 1}$$

Where:

σ_i : Components of stress vector

R_i.: Strengths under corresponding stresses

F: Fracture function

There are 6 main strengths that should be related to the occurring stress state:

$R_{\parallel}^{t}$: tensile strength parallel to fibers under uni-axial $\sigma_{\parallel}^{t}$
$R_{\parallel}^{c}$: compressive strength parallel to fibers under uniaxial $\sigma_{\parallel}^{c}$
$R_{\perp}^{t}$: tensile strength transverse to fibers under unaxial $\sigma_{\perp}^{t}$
$R_{\perp}^{c}$: compressive strength transverse to fibers under unaxial $\sigma_{\perp}^{c}$
$R_{\perp\perp}$: transverse shear strength under pure $\tau_{\perp\perp}$
$R_{\perp\parallel}$: longitudinal shear strength under pure $\tau_{\perp\parallel}$

The general form of fracture condition can also be rewritten as following:

$$\boxed{F(\sigma_1, \sigma_2, \sigma_3, \tau_{21}, \tau_{31}, \tau_{23}, R_{\parallel}^{t}, R_{\parallel}^{c}, R_{\perp}^{t}, R_{\perp}^{c}, R_{\perp\perp}, R_{\perp\parallel}) \lesseqgtr 1}$$

With:

F < 1: no fracture

F = 1: fracture limit reached, and fracture occurs

F > 1: fracture limit exceeded

with F $\in[0,\infty)$

The fiber failure criterion is expressed according to the following relation [1]:

FF criterion for 2D and 3D failure analyses	$f_{E,FF} = \frac{1}{\pm R_{\parallel}^{t,c}} \underbrace{\left[\sigma_1 - \left(\upsilon_{\perp\parallel} - \upsilon_{\perp\parallel f} \cdot m_{\sigma,f} \cdot \frac{E_{\parallel}}{E_{\parallel f}} \right) \cdot (\sigma_2 + \sigma_3) \right]}_{A}$ with; $\begin{cases} R_{\parallel}^{t} & for \quad A \geq 0 \\ -R_{\parallel}^{c} & for \quad A < 0 \end{cases}$

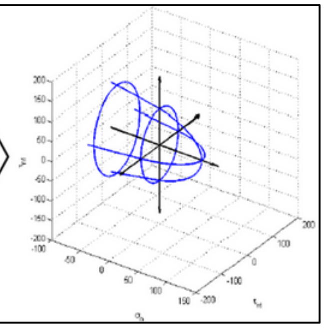
The Inter fiber failure criterion is expressed according to the following relation [1]:

IFF criterion for 3D failure analysis

$$f_{E,IFF}(\theta)=\sqrt{\left[\left(\frac{1}{R_{\perp}^{t}}-\frac{p_{\perp\psi}^{t}}{R_{\perp\psi}^{A}}\right)\cdot\sigma_n(\theta)\right]^2+\left(\frac{\tau_{nt}(\theta)}{R_{\perp\perp}^{A}}\right)^2+\left(\frac{\tau_{n1}(\theta)}{R_{\perp\parallel}}\right)^2}+\frac{p_{\perp\psi}^{t}}{R_{\perp\psi}^{A}}\cdot\sigma_n(\theta)\quad for\ \ \sigma_n(\theta)\geq 0$$

$$f_{E,IFF}(\theta)=\sqrt{\left(\frac{\tau_{nt}(\theta)}{R_{\perp\perp}^{A}}\right)^2+\left(\frac{\tau_{n1}(\theta)}{R_{\perp\parallel}}\right)^2+\left(\frac{p_{\perp\psi}^{c}}{R_{\perp\psi}^{A}}\cdot\sigma_n(\theta)\right)^2}+\frac{p_{\perp\psi}^{c}}{R_{\perp\psi}^{A}}\cdot\sigma_n(\theta)\quad for\ \ \sigma_n(\theta)<0$$

Fracture plane search with Stepwise or Golden Section search algorithms



- Comparison between failure theories:

TABLE 6.1 Comparison of Failure Theories

Type	Theory	Physical Basis	Operational Convenience	Required Experimental Characterization
Limit or noninteractive	Maximum stress	Tensile behavior of brittle material $\sigma_1>0, \sigma_2>0$ No stress interaction	Inconvenient	Few parameters by simple testing
	Maximum strain	Tensile behavior of brittle material $\sigma_1>0, \sigma_2>0$ Some stress interaction	Inconvenient	Few parameters by simple testing
Interactive	Strain-energy-based (Tsai-Hill)	Ductile behavior of anisotropic materials $\sigma_{1,2}<0, \tau_6\neq 0$ "Curve fitting" for heterogeneous brittle composites	Can be programmed Different functions required for tensile and compressive strengths (for each quadrant)	Biaxial testing is needed in addition to uniaxial testing
	Interactive tensor polynomial (Tsai-Wu)	Mathematically consistent Reliable "curve fitting"	General and comprehensive; operationally simple	Numerous parameters Comprehensive experimental program needed
Mixed	Failure mode separation (Hashin-Rotem)	Distinct separation between fiber and interfiber failures	Somewhat inconvenient	Few parameters by simple testing

Fig.7 Comparison of failure theories (Daniel, I, 1994)

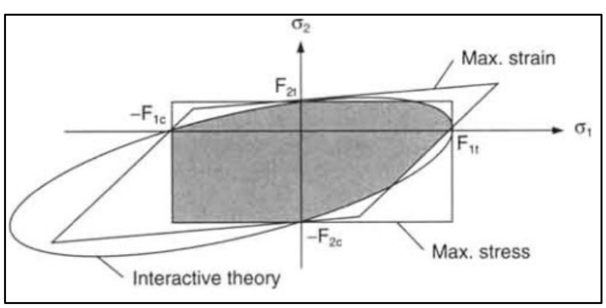


Fig.8 Graphical illustration of failure theories (Daniel, I, 1994)

2.3 Damage progression models:

After the failure criterion has been defined, a damage progression model has to be chosen. Various such models for laminated composites exist and can be categorized into two main groups (Knight, 2006) [1]: heuristic models based on a ply-discounting material degradation approach and models based on a continuum-damage-mechanics (CDM) approach using internal state variables.

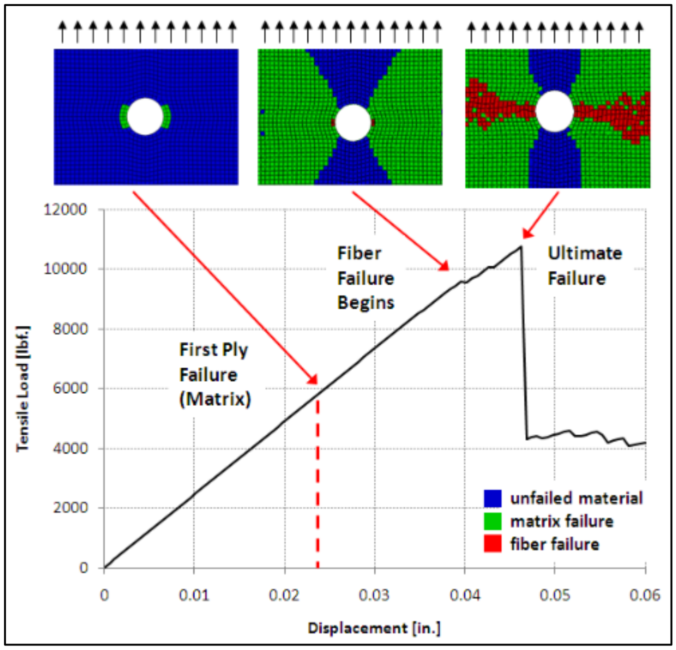


Fig.9 Illustration of progressive damage analysis (Autodesk, 2017)

➢ Ply-Discounting Approach

According to this approach, damage is modeled by degrading the elastic material stiffness coefficients by a value β_i, which essentially generates a diminishing stiffness value (i.e., approaches zero as the number of solution increments from failure initiation increases) for the i th stress component. In this strategy, the i th diagonal entry of the elastic constitutive matrix C_{ii} is set equal to β_i multiplied by C_{ii} , and the other row and column entries of the elastic constitutive matrix $[C]$ are also degraded in a similar manner. Another strategy to ply discounting involves degrading the elastic mechanical properties of the material directly (i.e., assumption of vanishing elastic moduli after failure initiation is detected) 20 and then re-computing the local material stiffness coefficients using the degraded mechanical properties. In this strategy, special care needs to be given to maintain symmetry in the degraded constitutive matrix.

The stiffness degradation can be applied either only once when failure initiation is detected (instant damage), or it can be done recursively on each solution increment after failure initiation is detected (recursive damage) [1]. Recursive material degradation typically provides a "gentler" process of degrading material stiffness data and potentially can improve convergence characteristics of the solution procedure compared to an "abrupt" single-step degradation approach using near zero values for the degradation factors. In addition, once a failure mode is detected that failure mode is not checked at that material point again; however, recursive degradation of the material stiffness coefficients will continue to be applied. Material degradation continues until the degradation factor reaches a specified minimum value and then is held constant at that minimum value (e.g. 10^{-30}). At subsequent solution increments, other failure modes within a given failure criteria are evaluated at that material point and potentially could lead to a subsequent failure in a different mode. All the possible stiffness degradation schemes can be seen in figure 10.

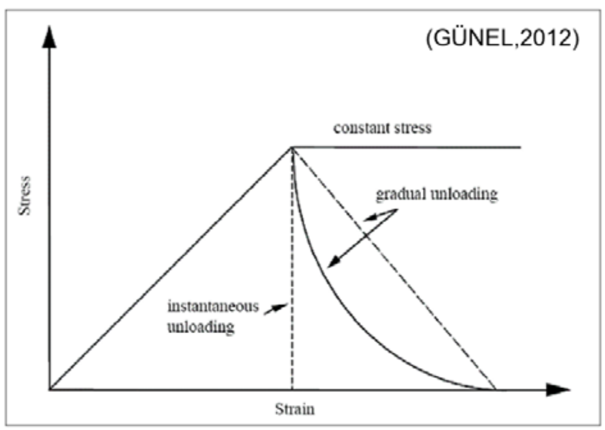


Fig.10 Stress-strain relationship for different stiffness degradation methods (Gunel, 2012)

➢ Internal State Variable Approach

Continuum-damage-mechanics (or CDM) models generally describe the internal damage in the material by defining one or more internal state variables. Regardless of the damage state, these CDM models still represent the material as continuum having smooth, continuous field equations. CDM models express the constitutive relations in a manner similar to the elastic constitutive relations given previously, except that the coefficients (i.e., either compliance coefficients, stiffness coefficients, or the mechanical properties themselves) are functions of one or more internal state variables.

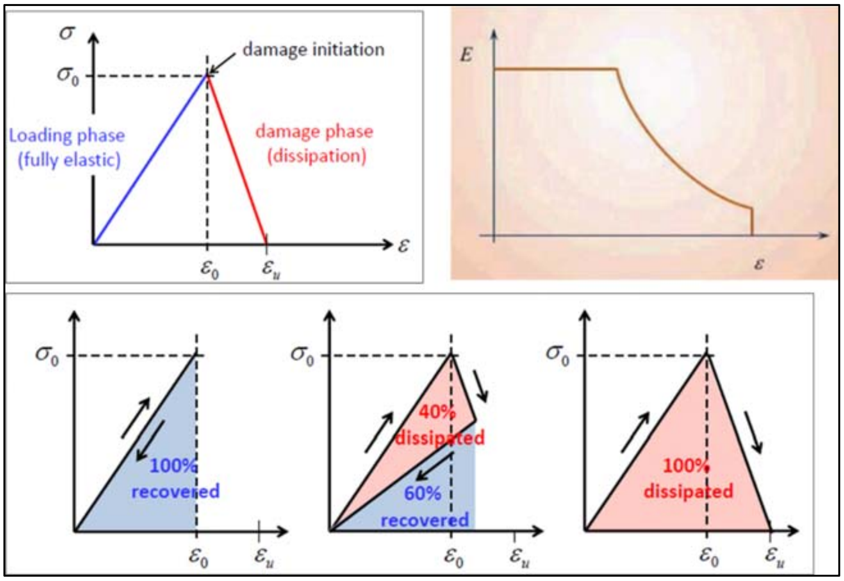


Fig.11 Damage propagation as function of strain energy
(Mohr, D 2015)

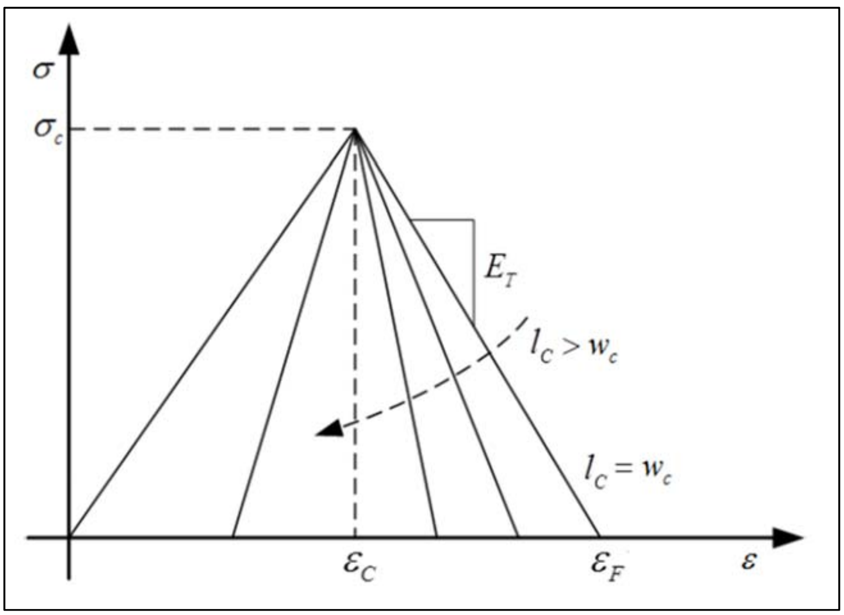


Fig.12 Illustration of effect of characteristic element length on softening
behavior (Mohr, D 2015)

3. Analysis and Development Reference Manual

3.1 User defined subroutines in ABAQUS:

User subroutines are special features that are provided to increase the functionality of several Abaqus capabilities for which the usual data input methods alone may be too restrictive. The available subroutines offer a variety of options, from specifying user-defined loading or initial conditions, to the creation of user-defined elements. Thus, providing an extremely powerful and flexible tool for analysis. The subroutines are typically written as FORTRAN code and must be included in a model when executing the analysis [6].

User subroutines should be written with great care. To ensure their successful implementation, the rules and guidelines below should be followed. For a detailed discussion of the individual subroutines, including coding interfaces and requirements, refer to the Abaqus User Subroutines Reference Manual [6].

3.2 UMAT Material subroutine for implicit analysis:

UMAT is the user subroutine for the definition of user based constitutive models in ABAQUS/Standard solver. In context of progressive damage analysis of laminated composites, the user material subroutines are written according to the following algorithm:

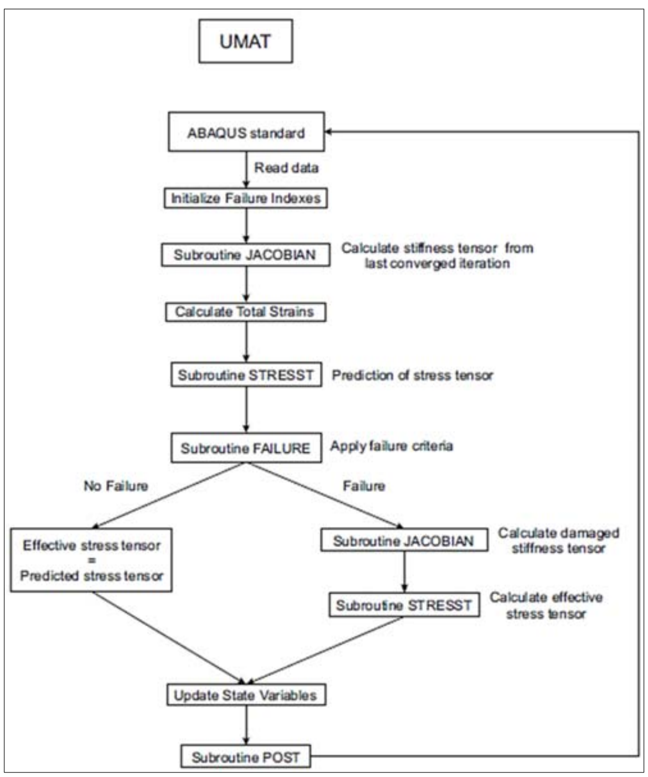


Fig.13 UMAT Subroutine flowchart (Gunel, 2012)

According to the ABAQUS documentation the UMAT subroutine:

- Can be used to define the mechanical constitutive behavior of the material.
- Can be used with any procedure that includes mechanical behavior.
- Can use solution-dependent variables.
- Must update at each solution increment, the stresses and the solution dependent variables.
- Must provide the material Jacobian matrix (incremental stresses and strains relationship).

3.3 VUMAT Material subroutine for explicit analysis:

In ABAQUS/Explicit the user-defined material model is implemented in user subroutine VUMAT. In the VUMAT subroutine. The Stress and SDVs at the end of each solution increment must be defined. With comparison to the UMAT subroutine, the VUMAT subroutine has vectorized interface. In VUMAT the data are passed in and out in large blocks (dimension nblock), where each entry in an array of length nblock corresponds to a single material point. All material points in the same block have the same material name and belong to the same element type.

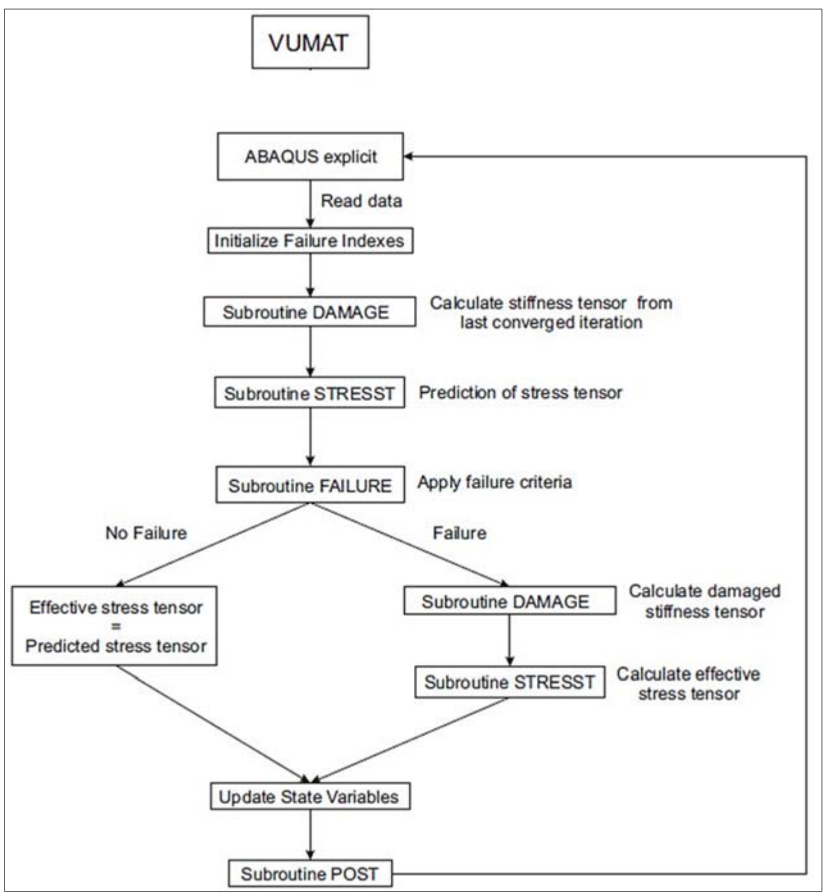


Fig.14 VUMAT Subroutine flowchart (Gunel, 2012)

3.4 Implementation of 3D orthotropic material law:

The adopted material law for calculating stresses of the composite laminate is the Generalized Hooke's law for orthotropic elasticity the orthotropic law allows for explicit consideration of 3D stress states and consider the laminate properties along three planes of symmetry. The stress-strain relations for the adopted constitutive law is illustrated in matrix format below, it should be noted that for such material models, the user has to specify the local coordinate system of the material (e.g. Fiber Direction) with respect to global coordinates of the FE model. In the implemented subroutine code, the utility subroutine (Ortho3D) is used to form the effective constitutive material tensor and calculate the stresses. It should be noted that composite material needs to be defined on macroscale for using the subroutines, the process of reaching this material modelling scale is summarized in Fig 16.

$$
\begin{Bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ \gamma_{12} \\ \gamma_{13} \\ \gamma_{23} \end{Bmatrix} = \begin{bmatrix} 1/E_1 & -\nu_{21}/E_2 & -\nu_{31}/E_3 & 0 & 0 & 0 \\ -\nu_{12}/E_1 & 1/E_2 & -\nu_{32}/E_3 & 0 & 0 & 0 \\ -\nu_{13}/E_1 & -\nu_{23}/E_2 & 1/E_3 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1/G_{12} & 0 & 0 \\ 0 & 0 & 0 & 0 & 1/G_{13} & 0 \\ 0 & 0 & 0 & 0 & 0 & 1/G_{23} \end{bmatrix} \begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{12} \\ \sigma_{13} \\ \sigma_{23} \end{Bmatrix}
$$

Extension   Extension-Extension Coupling   Shear

Fig.15 Stress-Strain relations for orthotropic material (ABAQUS Manual 2003)

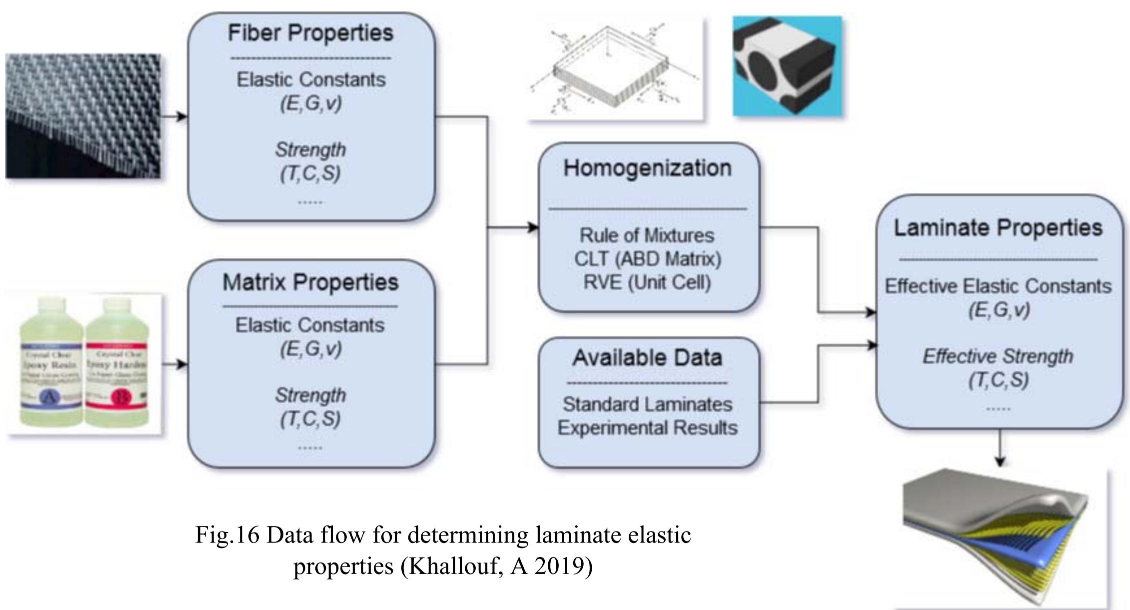


Fig.16 Data flow for determining laminate elastic properties (Khallouf, A 2019)

3.5 Implementation of 3D failure criteria:

Failure evaluation of material integration points at different stress levels is accomplished in the subroutine code by the (failure_calc) subroutine which calculates the material exposure coefficients labelled in vector form e(i), where i can range from 1 to 6 depending on the failure criteria selected. The implemented failure criteria in the subroutine code is illustrated in the following figure and more details can be obtained in the subroutine code script or in [11]

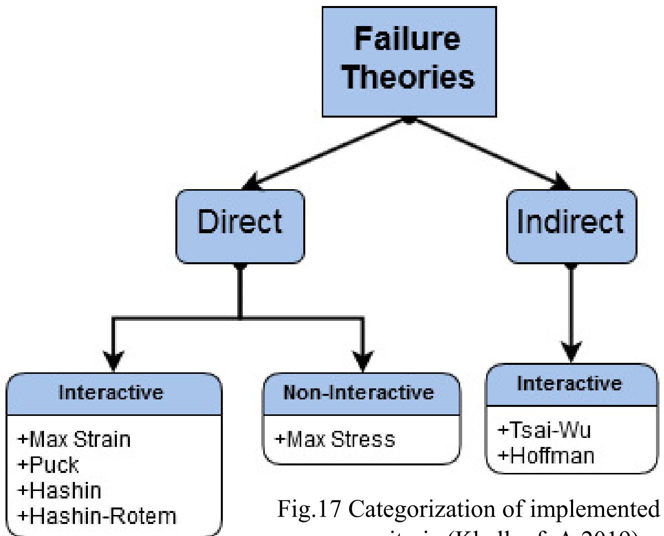


Fig.17 Categorization of implemented failure criteria (Khallouf, A 2019)

3.6 Application of damage and stiffness degradation:

Two different philosophies for damage models are implemented in the developed material subroutines as shown in figure 18. In the Ply-Discount approach, the user has to explicitly specify the degradation factors for the laminate properties upon failure detection. The degradation factors have to be specified for fiber and matrix failure modes in tension, compression and shear. The selection of these parameters depends on the desired laminate failure response which can be derived based on available experimental results, the parameters can also be tweaked to solve some convergence problems associated with rapid drop of elements stiffness in FE simulations. In the Continuum damage mechanics approach, the user is not required to specify any degradation factors. Instead the laminate fracture energies are required which have to be specified for fiber and matrix modes in tension, compression and shear. These values can be also obtained from experimental data and published standard laminate properties.

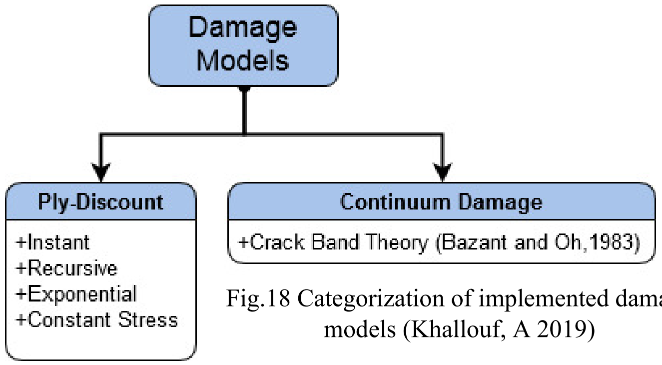


Fig.18 Categorization of implemented damage models (Khallouf, A 2019)

The following describes briefly the application algorithm for each damage model and more details can be obtained from the cited references:

- **Instantaneous damage:**

In this damage model, the material constitutive matrix elements shown in figure 15, are degraded instantly to certain percentage of their original value (i.e. usually 1%) based on the detected failure mode (Fiber/Matrix/Interlaminar/Shear). The degradation occurs in one analysis increment and therefore this approach can lead to numerical problems in large models at zones where high stress/strain gradients are expected. The user can eliminate such difficulties by tweaking the specified degradation factors until convergence is achieved.

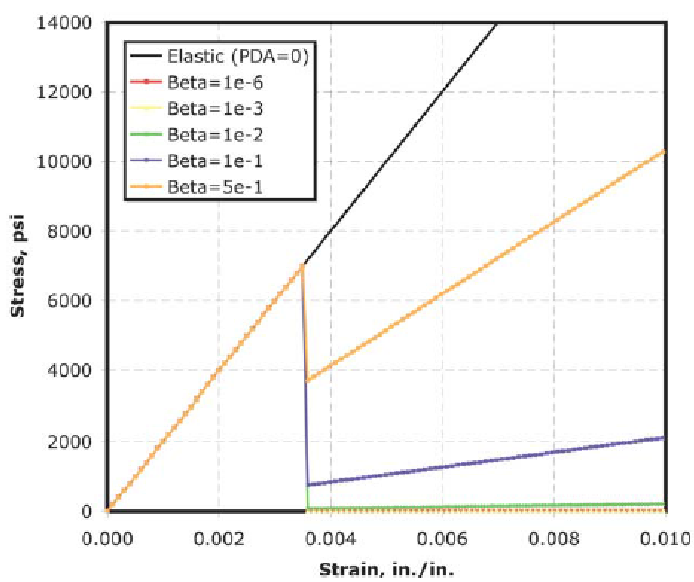


Fig.19 Effect of different degradation factors combined with instantaneous degradation (Norman,2006)

➢ Recursive damage:

Realizing the limitations of instantaneous damage model, the recursive model aims at reducing the material properties over multiple number of solution steps in recursive fashion, therefore the degradation factors which range from 0 to 1 are applied to the constitutive matrix for every step after damage has initiated until stiffness diminishes to a small value (i.e. 1% of its original value). A typical value suggested in literature for degradation factors is 0.5 which have been found to result in a good convergence behavior.

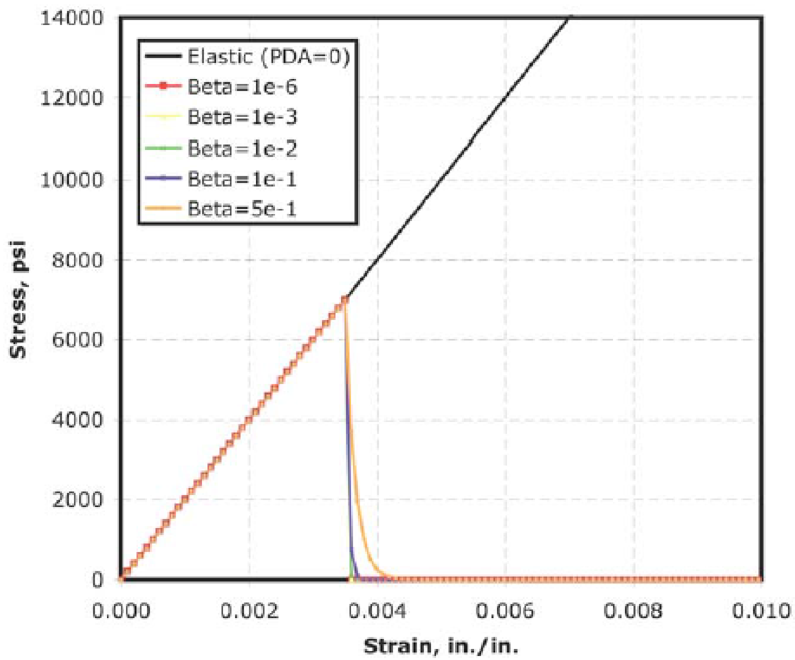


Fig.20 Effect of different degradation factors combined with recursive degradation (Norman,2006)

➢ Exponential damage:

The exponential damage models aim to describe the softening behavior of composite after failure has been detected using an exponential decay function as suggested by (Nahas,1986) and depicted in figure 21, similar to the previous damage models different degradation factors can be specified for different failure modes of the laminate which leads to different decay curves for fiber and matrix within the laminate, the degradation factors in this damage model control the rate at which the stiffness decays and the ultimate strain at which the material fully unloads, this in turn provides greater control over material post-failure behavior and allow for gradual degradation of properties.

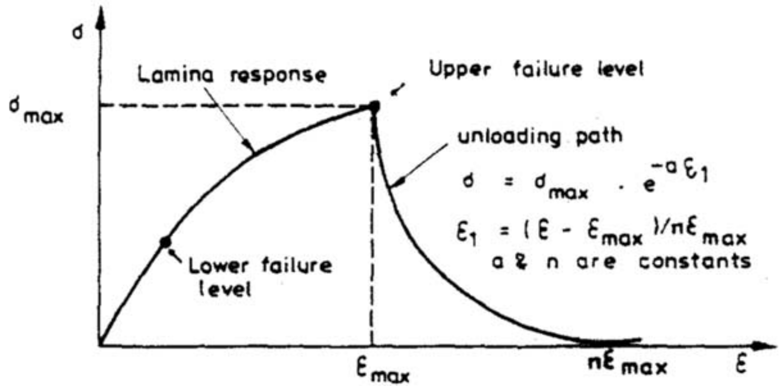


Fig.21 Post-failure model with exponential degradation

(Nahas,1986)

➢ **Constant Stress damage:**

The philosophy of this damage model is that failed laminates will support its initial failure load until total failure of lamina occurs, therefore no stress states greater than failure stress states are allowed for the material and the material preserve a constant stress level for the rest of the analysis as shown in figure 10. With regards to stiffness degradation, the material properties need to be reduced sufficiently such that the largest failure index equals one, this translates to a constrained optimization problem, where the degradation factors need to be optimized such that all stress states remains within the failure envelope of the material, in the current implementation a simplified version is assumed where the degermation factors are assumed to be the inverse of the material exposure coefficients e(i) for each failure mode. This simplification works well for simple failure criteria like Max Stress. However, further development and investigation is required for more complex failure criteria.

3.7 Numerical difficulties and possible solutions:

As with many non-linear problems in FEA software, convergence issues can be encountered when applying UMAT, therefore some possible solutions need to be proposed. In the case of highly non-linear problems, the initial increment size should be set sufficiently small. The maximum increment size should also be set to a small value in case of a sudden stiffness change and accordingly a large number of increments should be allowed. Knight N. [9] also recommends disabling the extrapolation feature for the next solution increment in order to enhance numerical convergence. This is accomplished using the ABAQUS keyword command *STEP EXTRAPOLATION=NO. Another potential remedy if a numerical instability is present is to add damping to the model (automatic stabilization). Damping can help convergence, but care should be taken that it does not affect the solution, which can be done by checking if the dissipation energy ALLSD is much smaller than the strain energy ALLIE (Obbink-Huizer), 2016) [14] Lastly, one should consider conducting a mesh sensitivity study, as the mesh quality might be affecting the solution stability. A very fine mesh leads to a large stiffness matrix which in return might lead to divergence due to unbalanced forces introduced to the system during analysis. On the other hand, a coarse mesh can lead to a singular stiffness matrix resulting in divergence of the solution and inability to capture sudden stiffness changes.

4. Code Testing in ABAQUS/PDALAC:

4.1 Rectangular Panel demonstration problem

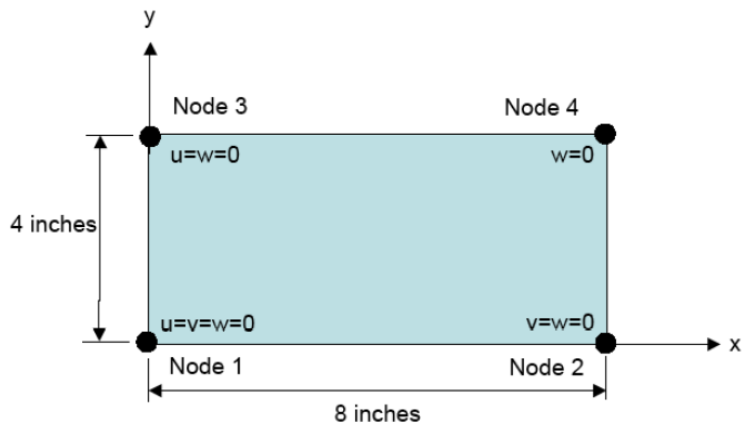


Fig.22 Geometry and Boundary conditions of the demonstration test problem

(Norman,2006)

```
** UMAT Property Data Definitions
** props(1-8):E11t,E22t,E33t,E11c,E22c,E33c,G12,G13,
** props(9-16):G23,nu12,nu13,nu23, Xt, Yt, Zt, Xc,
** props(17-24):Yc,Zc,S12,S13,S23,Eps11T,Eps22T,Eps33T,
** props(25-32):Eps11C,Eps22C,Eps33C,Gam12,Gam13,Gam23,Eps11Tmx,Eps22Tmx,
** props(33-40):Eps33Tmx,Eps11Cmx,Eps22Cmx,Eps33Cmx,Gam12mx,Gam13mx,Gam23mx,GIc,
** props(41-48):FPZ,SlimT,SlimC,SlimS,weibull(1),weibull(2),weibull(3),weibull(4),
** props(49-55):weibull(5),weibull(6),Dgrd(1),Dgrd(2),Dgrd(3),RECURS,PDA
** ===================================================================================
** Demo problem material definition #1, total thickness =  0.20 inches
**   - Linear elastic brittle
*MATERIAL, NAME=DEMOMAT1
*USER MATERIAL,  CONSTANTS=55
 2.000E+06, 2.000E+06, 2.000E+06, 4.500E+06, 4.500E+06, 4.500E+06, 0.800E+06, 0.800E+06,
 0.800E+06, 2.500E-01, 2.500E-01, 2.500E-01, 7.000E+03, 7.000E+03, 7.000E+03, 1.800E+04,
 1.800E+04, 1.800E+04, 6.200E+03, 6.200E+03, 6.200E+03, 3.500E-03, 3.500E-03, 3.500E-03,
 4.000E-03, 4.000E-03, 4.000E-03, 7.750E-03, 7.750E-03, 7.750E-03, 0.200E-01, 0.200E-01,
 0.200E-01, 0.200E-01, 0.200E-01, 1.000E-01, 1.000E-01, 1.000E-01, 1.000E-01, 3.000E+01,
 2.000E-01, 0.000E+00, 0.000E+00, 0.000E+00, 0.000E+00, 0.000E+00, 0.000E+00, 0.000E+00,
 0.000E+00, 0.000E+00, 1.000E-06, 1.000E-06, 1.000E-06, 1.000E+00, 1.000E+00
```

Fig.23 Prescribed material properties for the demonstration problem

(Norman,2006)

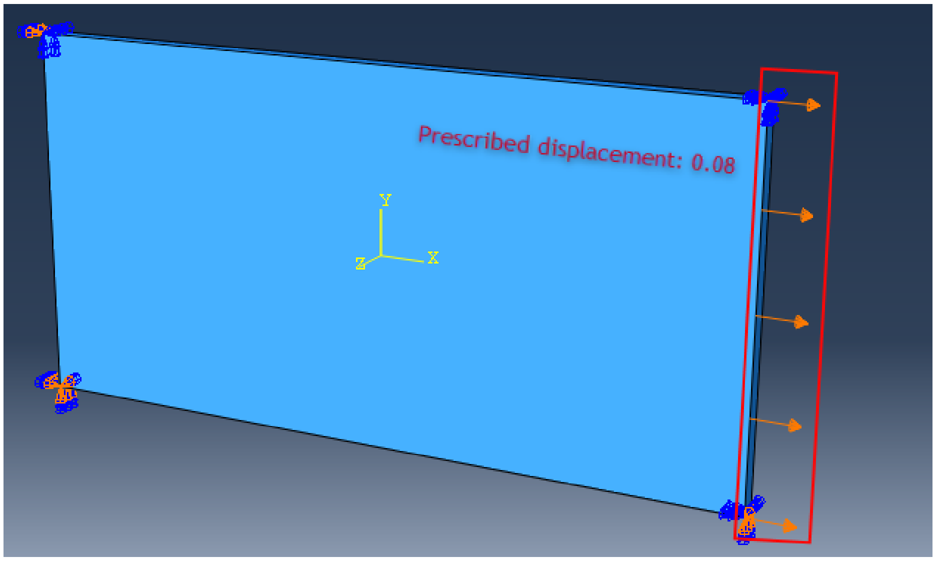


Fig.24 3D Finite element model in ABAQUS

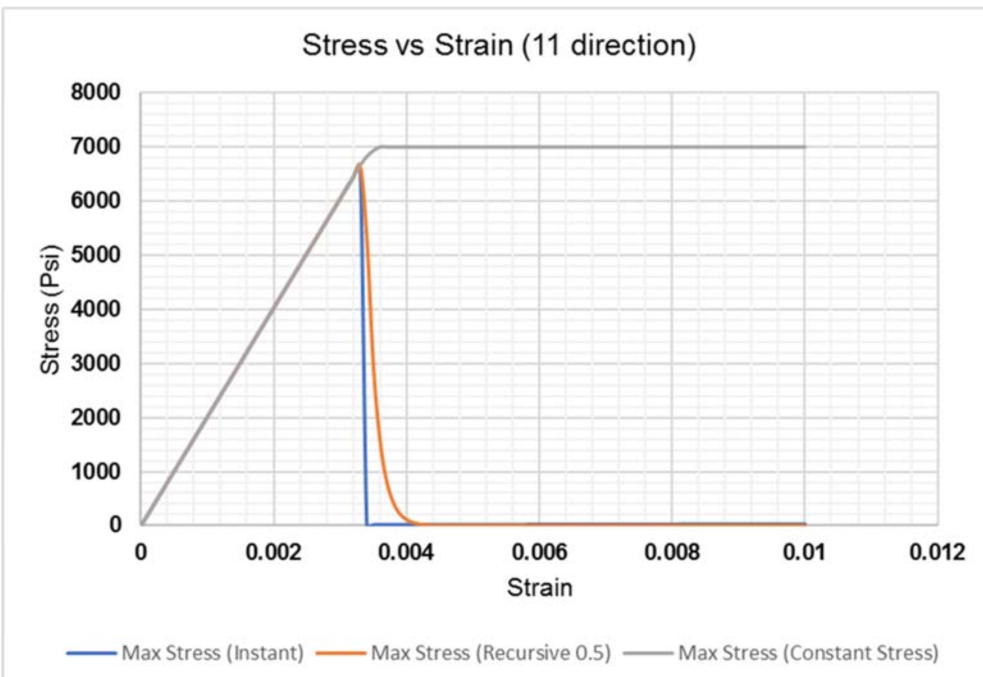


Fig.25 Stress vs Strain curve at plate centroid for Max stress criteria with different damage models

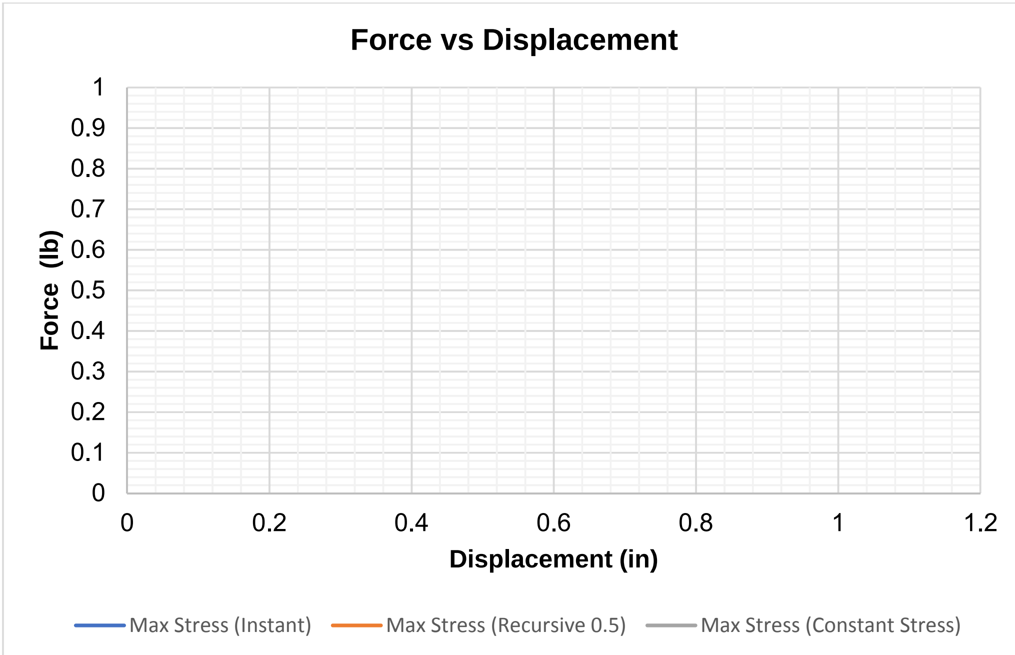


Fig.26 Horizontal displacement vs Reaction force for Max Stress criteria with different damage models

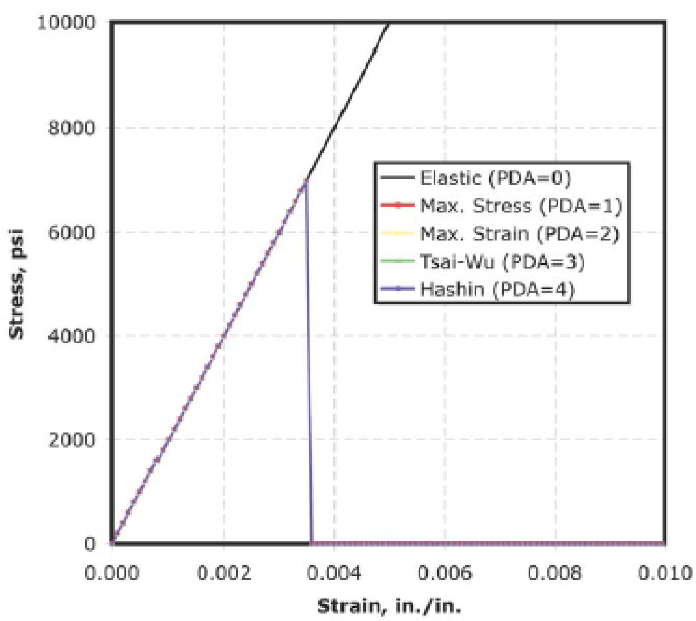
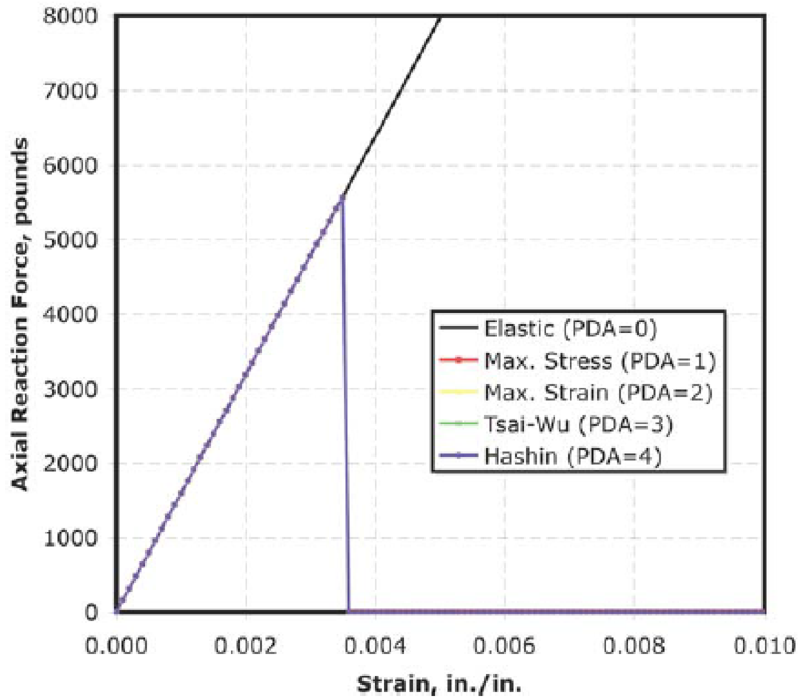


Figure 5. Stress-strain behavior for different progressive failure analysis criteria implemented within the UMAT subroutine obtained using instantaneous degradation, a degradation factor of 10^{-6}, and a maximum solution increment size factor of 0.01.

Fig.27 Stress vs Strain curve for benchmark problem solution

(Norman,2006)



(a) Axial reaction force as a function of applied strain.

Fig.28 Force vs Strain curve for benchmark problem solution

(Norman,2006)

4.2 Open Hole Tensile specimen problem

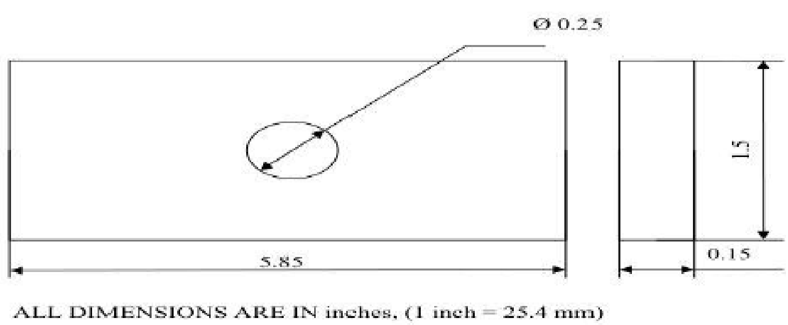


Fig.1. Geometric details of panel with circular cutout

Fig.29 Benchmark problem plate geometry

(Bammankatti,2014)

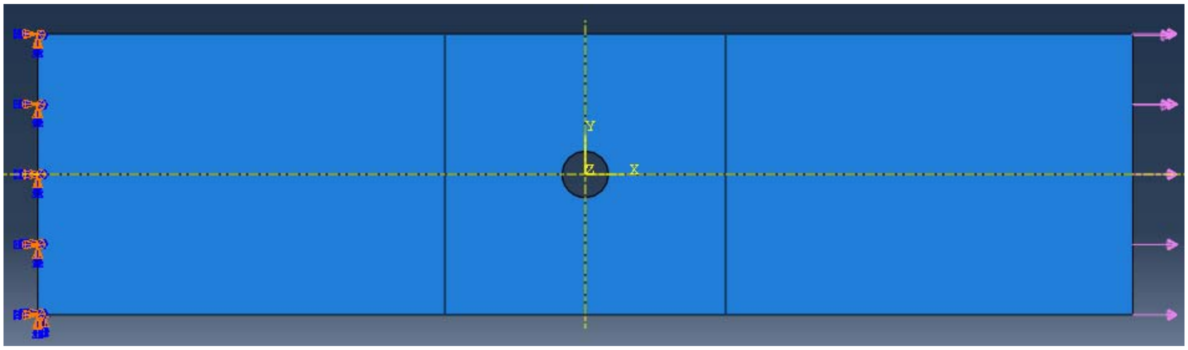


Fig.30 3D Finite element model in ABAQUS

Elastic Constants					
E11 (GPa)	117.2	v12	0.34	G12 (Gpa)	3.1
E22 (GPa)	9.653	v13	0.34	G13 (GPa)	3.1
E33 (GPa)	9.653	v23	0.34	G23 (GPa)	3.1
Failure Data					
Max Fiber Tension Xt (MPa)	1378.9	Max Matrix Tension Yt (MPa)		103.4	
Max Fiber Compression Xc (MPa)	689.5	Max Matrix Compression Yc (MPa)		206.8	
Layer Shear Strength S12 (MPa)	103.422	(Zt=Yt, Zc=Yc), (S12=S13=S23)			

Fig.31 T300/5208 Graphite/epoxy Laminate Material and Strength Properties

(Bammankatti,2014)

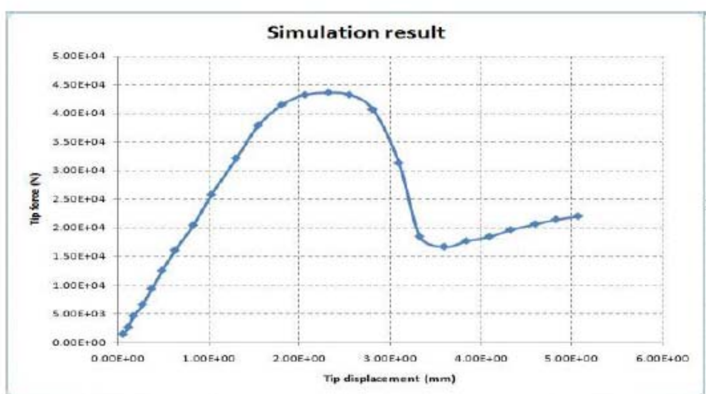


Fig 6, Force v/s displacement plot

Table 2, Simulated failure load result

First ply failure load	25800 N
Ultimate ply failure load	43700 N

Fig.32 Benchmark problem provided solution

(Bammankatti,2014)

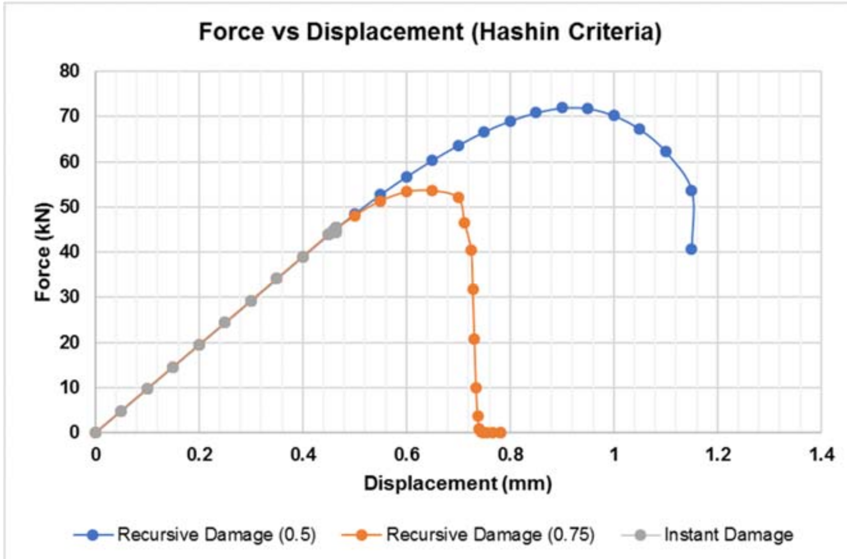


Fig.33 Horizontal displacement vs Reaction force for Hashin criteria with different damage models

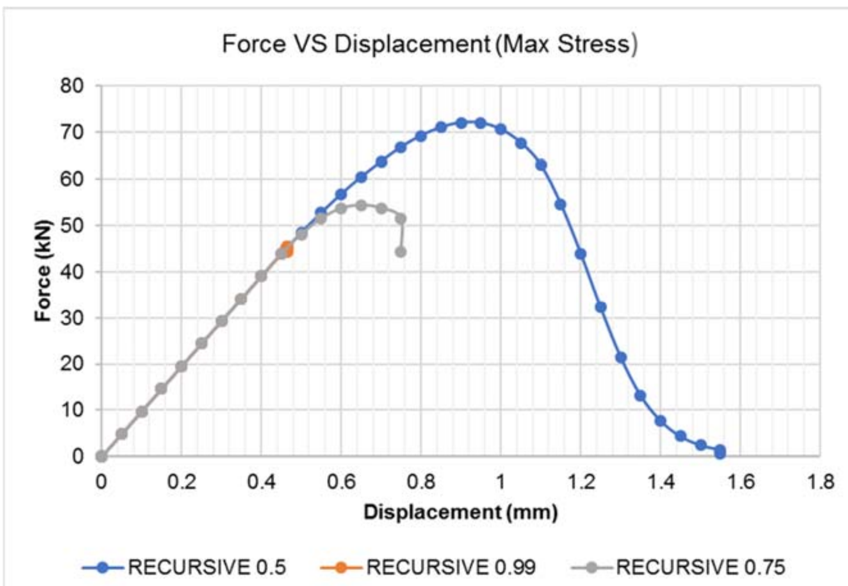


Fig.34 Horizontal displacement vs Reaction force for Max Stress criteria with different damage models

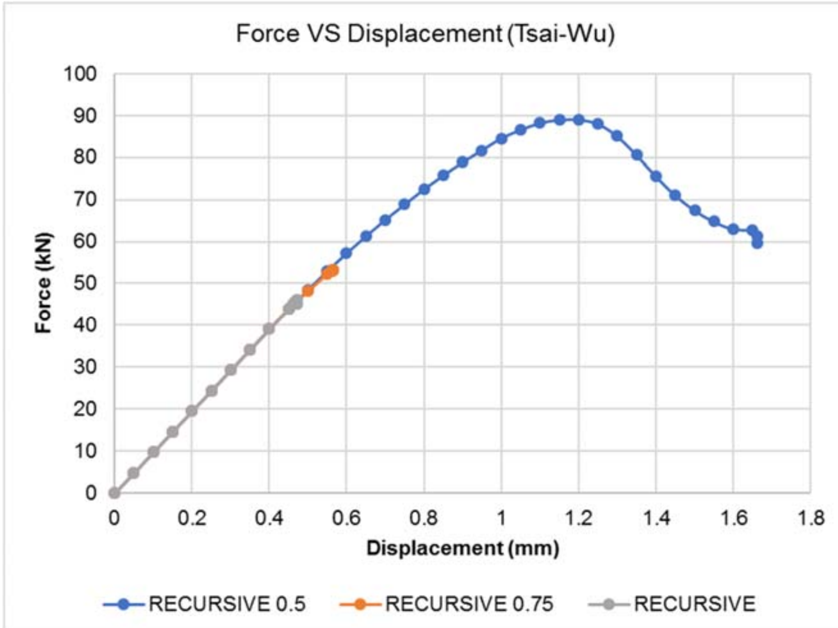


Fig.35 Horizontal displacement vs Reaction force for Tsai-Wu criteria with different damage models

4.3 Verification against WWFE test data

Table 1 Typical properties for a Carbon Epoxy composite [3]		
tensile strength 2 direction	Y_t	59.1 MPa
compressive strength 2 direction	Y_c	231.2 MPa
in plane shear strength	S_{12}	98.4 MPa

Fig.36 Typical properties for a carbon epoxy composite
Cuntze ,RG et al ,1997

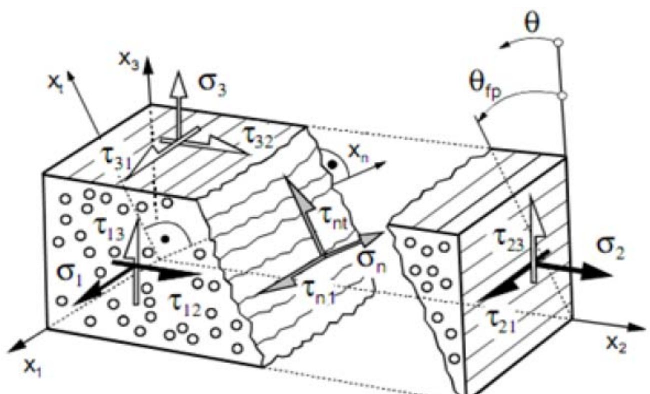


Fig.37: Illustration of Puck action Plane

(Puck & Shurmann,1998)

Table 2 Example stress states for which the Puck failure model predicts IFF (calculated using properties in Table 1)

stress state	σ_{11}	σ_{22}	σ_{33}	σ_{12}	σ_{23}	σ_{13}	σ_n	τ_{n1}	τ_{nt}	θ_{fr}
1 (pure shear)	0.0	0.0	0.0	0.0	59.1	0.0	59.1	0.0	0.0	45°
2 (pure shear)	0.0	0.0	0.0	98.4	0.0	0.0	0.0	98.4	0.0	0°
3 (uniaxial compression)	0.0	-231.2	0.0	0.0	0.0	0.0	-92.0	0.0	-113.2	51°
4 (arbitrary 3D)	0.0	-10.0	40.0	21.0	24.0	43.3	49.7	47.8	0.75	67°

Fig.38: Resultant stresses on the fracture plane and fracture angle

Cuntze ,RG et al ,1997

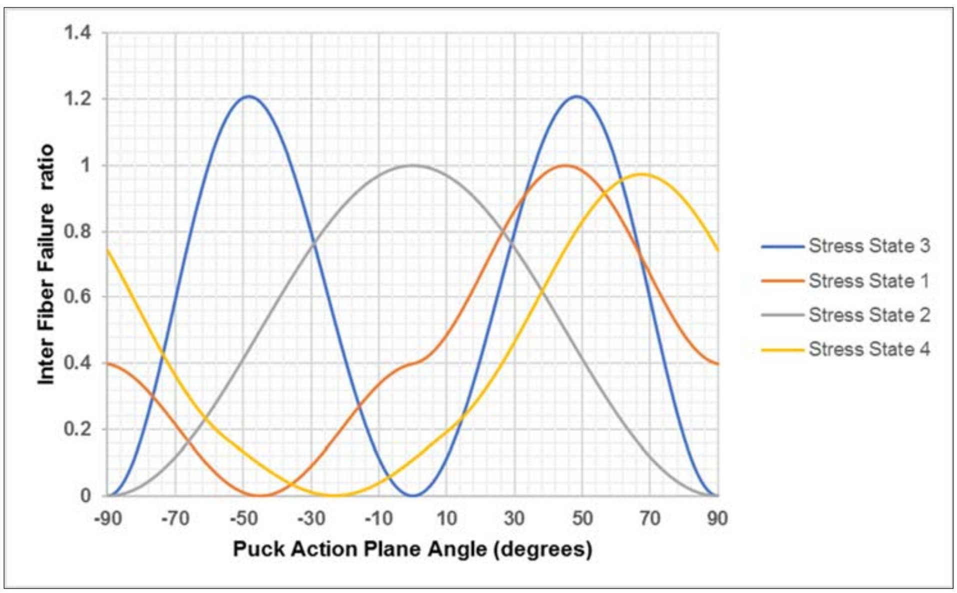


Fig.39: IFF ratio vs Puck angle for prescribed stress states obtained
from PDALAC

References

1. Altug Emiroglu, Master Thesis: Comparative Study of Puck and Tsai-Wu Failure Criteria, Technische Universität München, 2013.

2. M. Knops. Analysis of Failure in Fiber Polymer Laminates. Springer Berlin Heidelberg, Berlin, Heidelberg, 2008.

3. O. Mohr. Welche Umstände bedingen die Elastizitätsgrenze und den Bruch eines Materials? (German). Zeitschrift des Vereins deutscher Ingenieure, 24:1524 ff, 1900.

4. H. Deuschle. 3D failure analysis of UD fibre reinforced composites: Puck's theory within FEA. Phd thesis, Universität Stuttgart, 2010.

5. Daniel, I. M., Ishai, O., Daniel, I. M., & Daniel, I. (1994). *Engineering mechanics of composite materials* (Vol. 3, pp. 256-256). New York: Oxford university press.

6. Abaqus, U. S. M., & Manuals, E. U. S. (2002). Version 6.3, Hibbitt, Karlsson & Sorensen. *Inc. Rhode Island*.

7. Berger, A. (2014). *Numerical Modelling of Composite Materials Based on a Combined Manufacturing-Crash Simulation* (Doctoral dissertation, Queen Mary University of London).

8. Kaw, A. K. (2005). *Mechanics of composite materials*. CRC press.

9. Knight Jr, N. F., & Reeder, J. R. (2006). User-defined material model for progressive failure analysis.

10. Wiegand, J., Petrinic, N., & Elliott, B. (2008). An algorithm for determination of the fracture angle for the three-dimensional Puck matrix failure criterion for UD composites. *Composites Science and Technology*, *68*(12), 2511-2517

11. Sleight, D. W. (1999). Progressive failure analysis methodology for laminated composite structures.

12. Systèmes, D. (2010). Abaqus 6.10 online documentation. Abaqus User Subroutines Reference Manual.

13. 3ds.com. (2019). *Writing User Subroutines with Abaqus*. [online] Available at: https://www.3ds.com/products-services/simulia/services/training-courses/course-descriptions/writing-user-subroutines-with-abaqus/ [Accessed 17 May 2019].

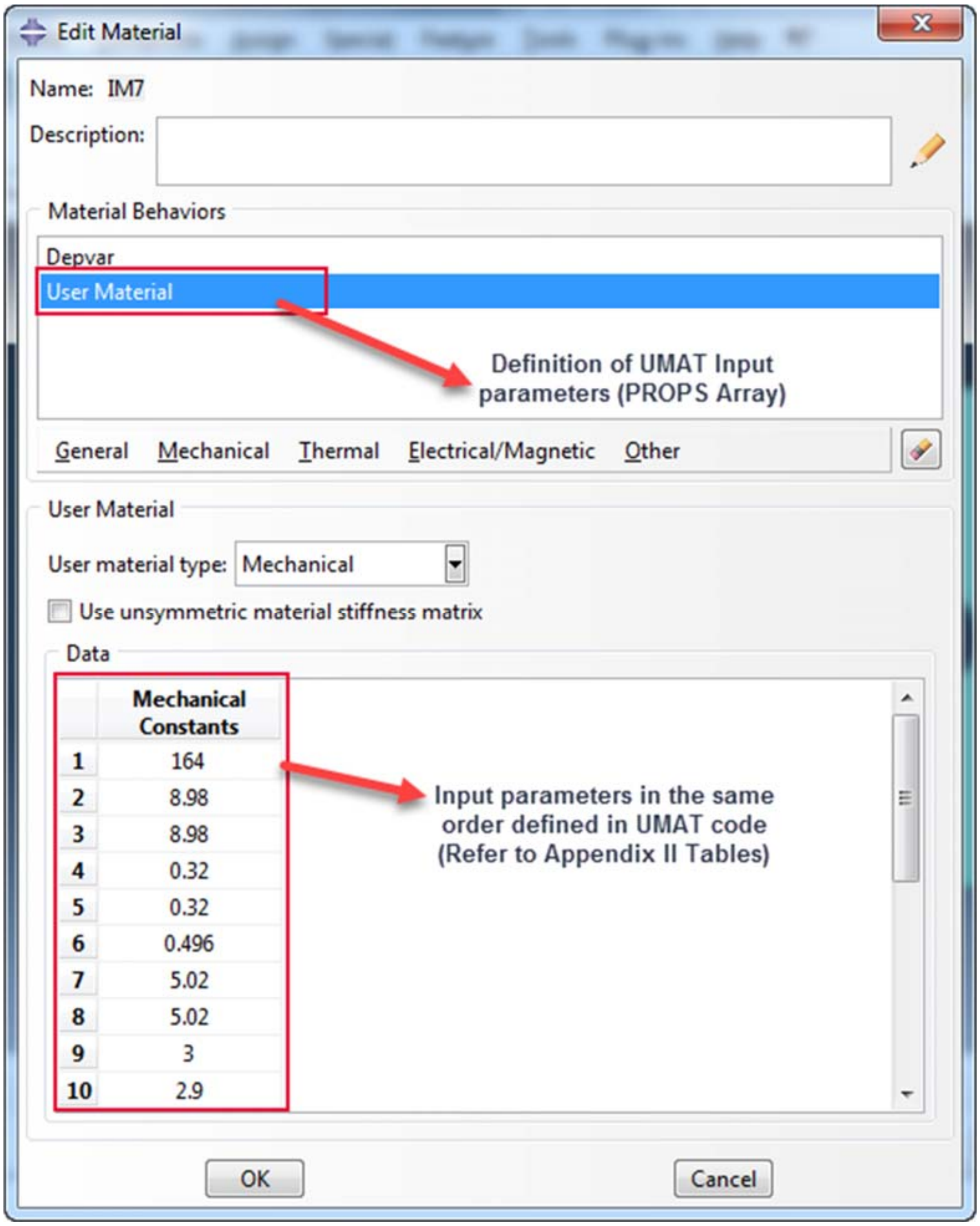
14. Obbink-Huizer C.,6 Tips solving non convergence with Abaqus FEA, retrieved from https://info.simuleon.com/blog/6-tips-solving-non-convergence-with-abaqus-fea

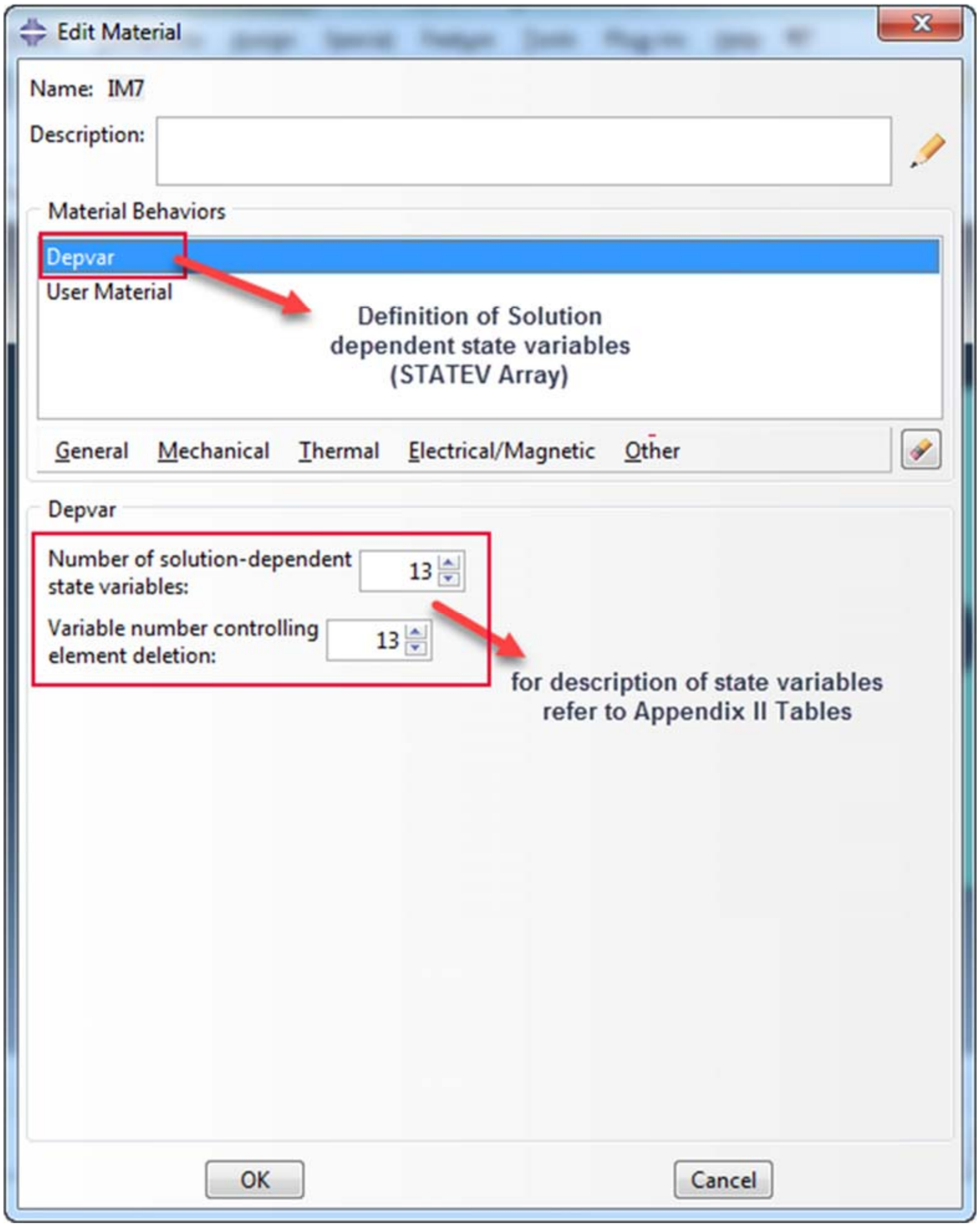
15. Larry Pearce, Sr.Lead application engineer, MSC Software, ‘‘Progressive ply failure in composites,’’ Webinar on May 15, 2012

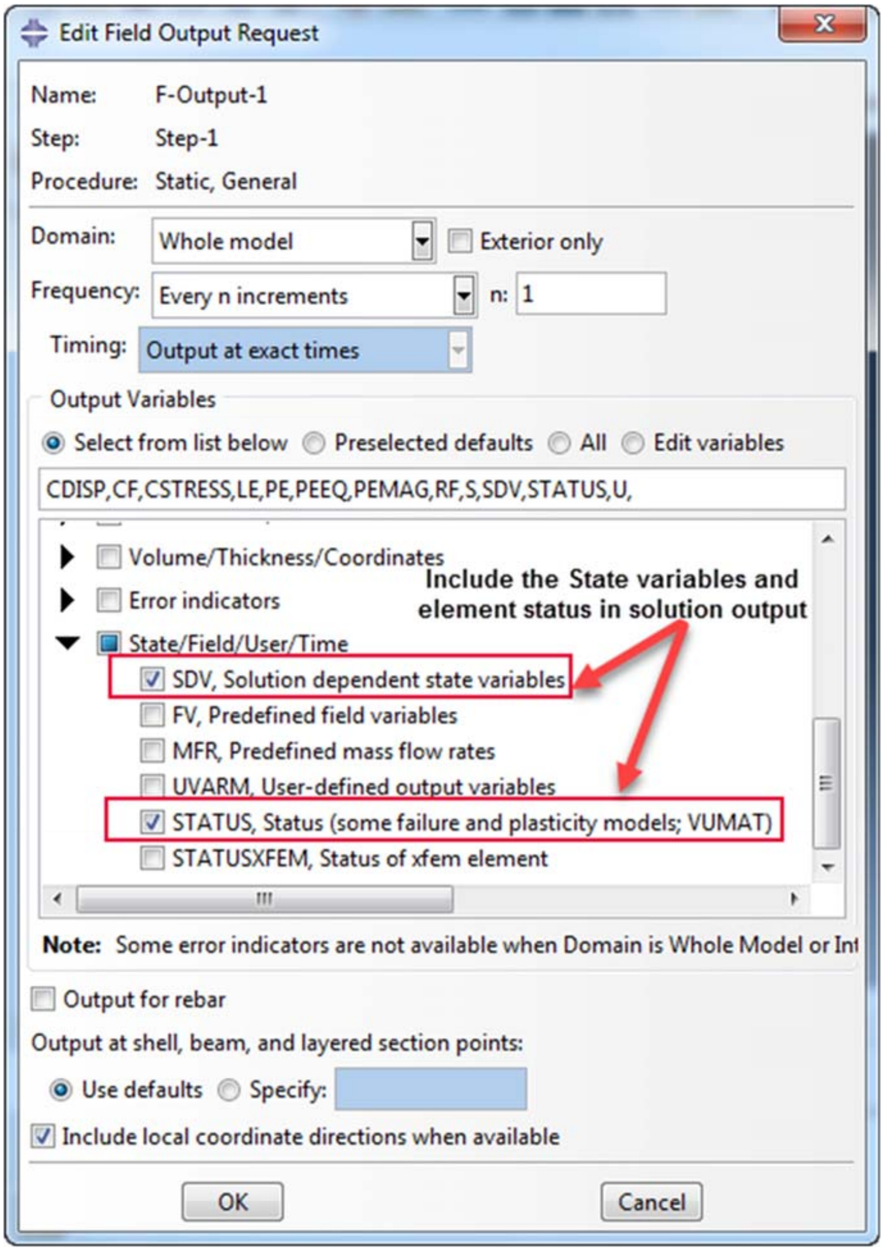
16. Cuntze RG, Deska R, Szelinski B, JeltschFricker R, Meckbach S, Huybrechts D, Kopp J, Kroll L, Gollwitzer S, Rackwitz R. Neue Bruchkriterien und Festigkeitsnachweise fuer unidirektionalen Faserkunststoffverbund unter mehrachsiger Beanspruchung - Modellbildung und Experimente. In: VDI Fortschritt-Berichte. Duesseldorf: VDI Verlag, 1997

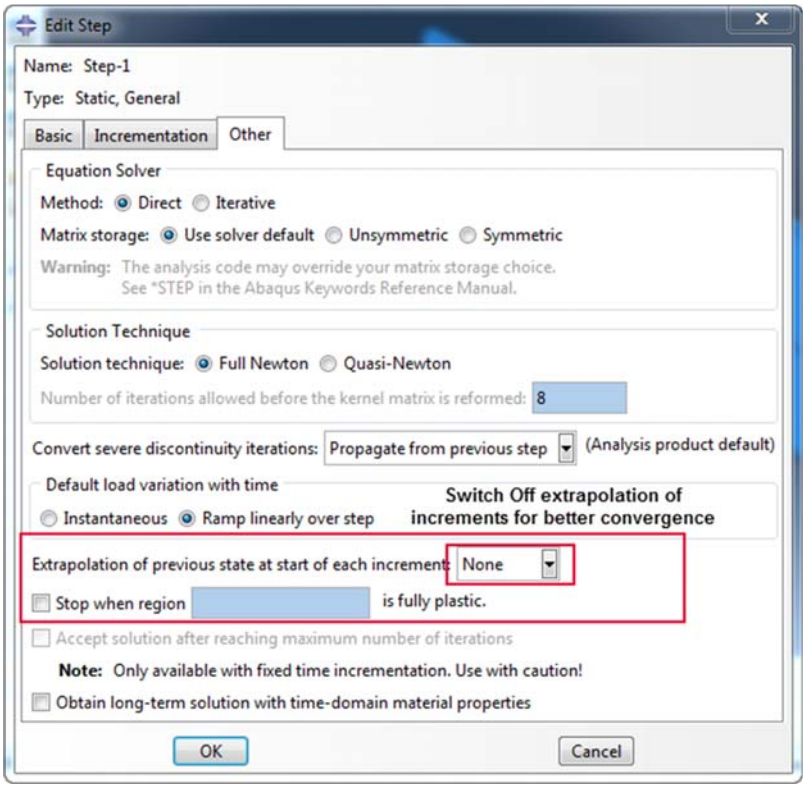
Appendices

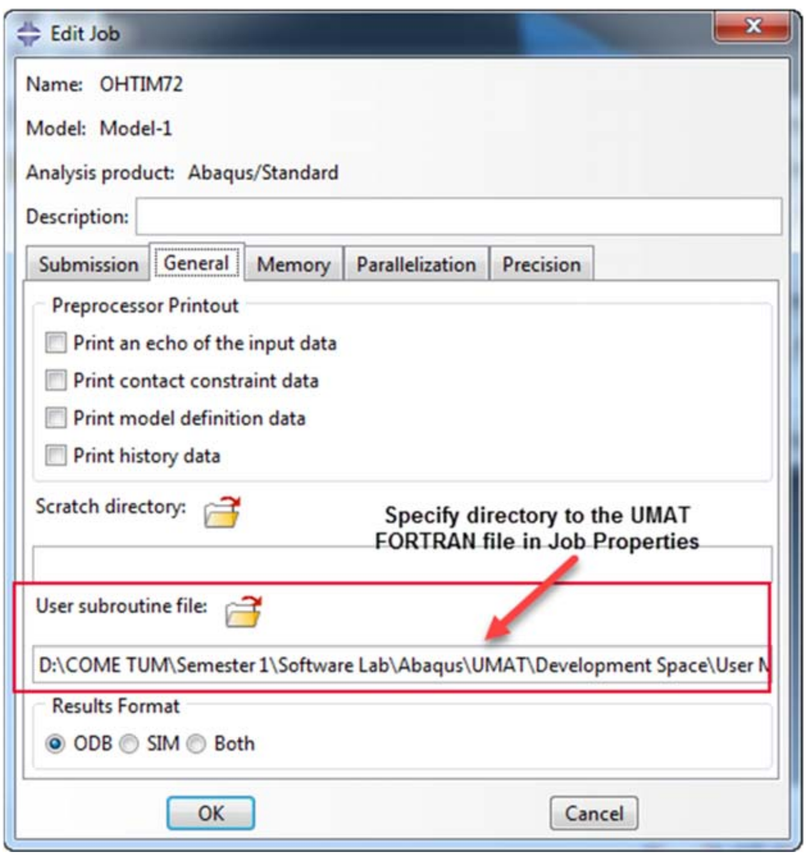
Appendix I: Running User material subroutines inside ABAQUS GUI




Edit Material
Name: IM7
Description:
Material Behaviors
Depvar
User Material
Definition of Solution
dependent state variables
(STATEV Array)
General
Mechanical
Thermal
Electrical/Magnetic
Other
Depvar
Number of solution-dependent
state variables:
13
Variable number controlling
element deletion:
13
for description of state variables
refer to Appendix II Tables
OK
Cancel


Edit Field Output Request
Name: F-Output-1
Step: Step-1
Procedure: Static, General
Domain: Whole model
Exterior only
Frequency: Every n increments
n: 1
Timing: Output at exact times
Output Variables
Select from list below
Preselected defaults
All
Edit variables
CDISP,CF,CSTRESS,LE,PE,PEEQ,PEMAG,RF,S,SDV,STATUS,U,
Volume/Thickness/Coordinates
Error indicators
State/Field/User/Time
SDV, Solution dependent state variables
FV, Predefined field variables
MFR, Predefined mass flow rates
UVARM, User-defined output variables
STATUS, Status (some failure and plasticity models; VUMAT)
STATUSXFEM, Status of xfem element
Include the State variables and element status in solution output
Note: Some error indicators are not available when Domain is Whole Model or Int
Output for rebar
Output at shell, beam, and layered section points:
Use defaults
Specify:
Include local coordinate directions when available
OK
Cancel


Edit Step
Name: Step-1
Type: Static, General
Basic
Incrementation
Other
Equation Solver
Method: Direct Iterative
Matrix storage: Use solver default Unsymmetric Symmetric
Warning: The analysis code may override your matrix storage choice.
See *STEP in the Abaqus Keywords Reference Manual.
Solution Technique
Solution technique: Full Newton Quasi-Newton
Number of iterations allowed before the kernel matrix is reformed: 8
Convert severe discontinuity iterations: Propagate from previous step
(Analysis product default)
Default load variation with time
Instantaneous Ramp linearly over step
Switch Off extrapolation of increments for better convergence
Extrapolation of previous state at start of each increment: None
Stop when region
is fully plastic.
Accept solution after reaching maximum number of iterations
Note: Only available with fixed time incrementation. Use with caution!
Obtain long-term solution with time-domain material properties
OK
Cancel


Edit Job
Name: OHTIM72
Model: Model-1
Analysis product: Abaqus/Standard
Description:
Submission
General
Memory
Parallelization
Precision
Preprocessor Printout
Print an echo of the input data
Print contact constraint data
Print model definition data
Print history data
Scratch directory:
Specify directory to the UMAT FORTRAN file in Job Properties
User subroutine file:
D:\COME TUM\Semester 1\Software Lab\Abaqus\UMAT\Development Space\User N
Results Format
ODB SIM Both
OK
Cancel

Appendix II: Description of UMAT variables in ABAQUS

User-defined property data for the UMAT subroutine		
PROPS Array Entry	Variables Name	Description
1,3	E11,E22,E33	Laminate Orthotropic Young's Moduli values (E11 fiber direction)
4,6	ANU12,ANU13,ANU23	Poisson rations v12,v13,v23
7,9	G12,G13,G23	Laminate Shear Moduli values
10,11	Xt,Xc	Laminate fiber allowable strengths (Tension/Compression)
12,13	Yt,Yc	Laminate matrix allowable strengths (Tension/Compression)
14,15	Zt,Zc	Laminate interlaminar allowable strengths (Tension/Compression)
16,17,18	S12,S13,S23	Laminate allowable shear strengths (1-2,1-3,2-3 planes)
19,20	EpsXt, EpsXc	Laminate fiber allowable strains (Tension/Compression)
21,22	EpsYt, EpsYc	Laminate matrix allowable strains (Tension/Compression)
23,34	EpsZt, EpsZc	Laminate Interlaminar allowable strains (Tension/Compression)
25,26,27	GamS12,Gams13,GamS23	Laminate allowable engineering shear strains (1-2,1-3,2-3 planes)
28	failure_id	Failure criteria selection ID: (1) Max Stress (2)Max Strain (3)Tsai-Wu (4)Hoffman (5)Hashin (6)Hashin-Rotem (7)Puck
29	damage_id	Damage Model selection ID: (1) Instant (2)Recursive (3)Exponential (4)Constant Stress (5)(CDM)Crack-Band Theory
30,31	beta_ft, beta_fc	Instant/Recursive: fiber degradation factors (Tension/Compression)
32,33	beta_mt, beta_mc	Instant/Recursive: matrix degradation factors (Tension/Compression)
34	beta_s	Instant/Recursive: shear degradation factor (Tension/Compression)
35,36	a_ft, n_ft	Exponential: fiber degradation factors (Tension)
37,38	a_fc, n_fc	Exponential: fiber degradation factors (Compression)
39,40	a_mt, n_mt	Exponential: matrix degradation factors (Tension)
41,42	a_mc, n_mc	Exponential: matrix degradation factors (Compression)
43,44	a_s, n_s	Exponential: shear degradation factors
45,46	G_ft, G_fc	CDM: fiber fracture energies (Tension/Compression)
47,48	G_mt, G_mc	CDM: matrix fracture energies (Tension/Compression)
49,50	G_IC,G_IIC	CDM: shear fracture energies (Tension/Compression)
51,52	le,let	CDM: characteristic element length and thickness
53,54	alpha, beta	CDM: nonlinear shear degradation factors
55	THETAF	Puck: Maximum fracture angle in radians
56	MGF	Puck: Magnification factor
57,58	E11F, ANU12F	Puck: Fiber elastic modulus and Poisson ratio

UMAT-defined solution-dependent variables		
STATEV Array Entry	Variables Name	Description
1	dmg(1)	Degradation factor for the σ11 stress component
2	dmg(2)	Degradation factor for the σ22 stress component
3	dmg(3)	Degradation factor for the σ33 stress component
4	dmg(4)	Degradation factor for the σ12 stress component
5	dmg(5)	Degradation factor for the σ13 stress component
6	dmg(6)	Degradation factor for the σ23 stress component
7	fflags(1)	Failure flag for first failure mode
8	fflags(2)	Failure flag for second failure mode
9	fflags(3)	Failure flag for third failure mode
10	fflags(4)	Failure flag for fourth failure mode
11	fflags(5)	Failure flag for fifth failure mode
12	fflags(6)	Failure flag for sixth failure mode
13	DelEl	Element deletion variable

DDSDDE(NTENS, NTENS)

Jacobian matrix of the constitutive model. DDSDDE(I, J) defines the change in the Ith stress component at the end of the time increment caused by an infinitesimal perturbation of the Jth component of the strain increment array. Unless you invoke the unsymmetrical equation solution capability for the user-defined material, ABAQUS/Standard will use only the symmetric part of DDSDDE. The symmetric part of the matrix is calculated by taking one half the sum of the matrix and its transpose.

STRESS(NTENS)

This array is passed in as the stress tensor at the beginning of the increment and must be updated in this routine to be the stress tensor at the end of the increment. If you specified initial stresses ("Initial conditions," Section 19.2.1), this array will contain the initial stresses at the start of the analysis. The size of this array depends on the value of NTENS as defined below. In finite-strain problems the stress tensor has already been rotated to account for rigid body motion in the increment before UMAT is called, so that only the corotational part of the stress integration should be done in UMAT. The measure of stress used is "true" (Cauchy) stress.

STATEV(NSTATV)

An array containing the solution-dependent state variables. These are passed in as the values at the beginning of the increment unless they are updated in user subroutines USDFLD (“USDFLD,” Section 25.2.39) or UEXPAN (“UEXPAN,” Section 25.2.20), in which case the updated values are passed in. In all cases STATEV must be returned as the values at the end of the increment. The size of the array is defined as described in “Allocating space” in “User subroutines: overview,” Section 25.1.1. In finite-strain problems any vector-valued or tensor-valued state variables must be rotated to account for rigid body motion of the material, in addition to any update in the values associated with constitutive behavior. The rotation increment matrix, DROT, is provided for this purpose.

SSE, SPD, SCD

Specific elastic strain energy, plastic dissipation, and “creep” dissipation, respectively. These are passed in as the values at the start of the increment and should be updated to the corresponding specific energy values at the end of the increment. They have no effect on the solution, except that they are used for energy output.

RPL

Volumetric heat generation per unit time at the end of the increment caused by mechanical working of the material.

DDSDDT(NTENS)

Variation of the stress increments with respect to the temperature.

DRPLDE(NTENS)

Variation of RPL with respect to the strain increments.

DRPLDT

Variation of RPL with respect to the temperature.

PNEWDT

Ratio of suggested new time increment to the time increment being used (DTIME, see discussion later in this section). This variable allows you to provide input to the automatic time incrementation algorithms in ABAQUS/Standard (if automatic time incrementation is chosen). For a quasi-static procedure, the automatic time stepping that ABAQUS/Standard uses, which is based on techniques for integrating standard creep laws (see "Quasi-static analysis," Section 6.2.5), cannot be controlled from within the UMAT subroutine. PNEWDT is set to a large value before each call to UMAT. If PNEWDT is redefined to be less than 1.0, ABAQUS/Standard must abandon the time increment and attempt it again with a smaller time increment. The suggested new time increment provided to the automatic time integration algorithms is PNEWDT × DTIME, where the PNEWDT used is the minimum value for all calls to user subroutines that allow redefinition of PNEWDT for this iteration. If PNEWDT is given a value that is greater than 1.0 for all calls to user subroutines for this iteration and the increment converges in this iteration, ABAQUS/Standard may increase the time increment. The suggested new time increment provided to the automatic time integration algorithms is PNEWDT × DTIME, where the PNEWDT used is the minimum value for all calls to user subroutines for this iteration. If automatic time incrementation is not selected in the analysis procedure, values of PNEWDT that are greater than 1.0 will be ignored and values of PNEWDT that are less than 1.0 will cause the job to terminate.

STRAN(NTENS)

An array containing the total strains at the beginning of the increment. If thermal expansion is included in the same material definition, the strains passed into UMAT are the mechanical strains only (that is, the thermal strains computed based upon the thermal expansion coefficient have been subtracted from the total strains). These strains are available for output as the "elastic" strains.

In finite-strain problems the strain components have been rotated to account for rigid body motion in the increment before UMAT is called and are approximations to logarithmic strain.

DSTRAN(NTENS)

Array of strain increments. If thermal expansion is included in the same material definition, these are the mechanical strain increments (the total strain increments minus the thermal strain increments).

TIME(1)

Value of step time at the beginning of the current increment.

TIME(2)

Value of total time at the beginning of the current increment.

DTIME

Time increment.

TEMP

Temperature at the start of the increment.

DTEMP

Increment of temperature.

PREDEF

Array of interpolated values of predefined field variables at this point at the start of the increment, based on the values read in at the nodes.

DPRED

Array of increments of predefined field variables.

CMNAME

User-defined material name left justified. Some internal material models are given names starting with the "ABQ_" character string. To avoid conflict, you should not use "ABQ_" as the leading string for CMNAME.

NDI

Number of direct stress components at this point.

NSHR

Number of engineering shear stress components at this point.

NTENS

Size of the stress or strain component array (NDI + NSHR).

NSTATV

Number of solution-dependent state variables that are associated with this material type (defined as described in “Allocating space” in “User subroutines: overview,” Section 25.1.1).

PROPS(NPROPS)

User-specified array of material constants associated with this user material.

NPROPS

User-defined number of material constants associated with this user material.

COORDS

An array containing the coordinates of this point. These are the current coordinates if geometric nonlinearity is accounted for during the step (see “Procedures: overview,” Section 6.1.1); otherwise, the array contains the original coordinates of the point.

DROT(3,3)

Rotation increment matrix. This matrix represents the increment of rigid body rotation of the basis system in which the components of stress (STRESS) and strain (STRAN) are stored. It is provided so that vector- or tensor-valued state variables can be rotated appropriately in this subroutine: stress and strain components are already rotated by this amount before UMAT is called. This matrix is passed in as a unit matrix for small-displacement analysis and for large-displacement analysis if the basis system for the material point rotates with the material (as in a shell element or when a local orientation is used).

CELENT

Characteristic element length, which is a typical length of a line across an element for a first-order element; it is half of the same typical length for a second-order element. For beams and trusses it is a characteristic length along the element axis. For membranes and shells, it is a characteristic length in the reference surface. For axisymmetric elements it is a characteristic length in the plane only. For cohesive elements it is equal to the constitutive thickness.

DFGRD0(3,3)

Array containing the deformation gradient at the beginning of the increment. See the discussion regarding the availability of the deformation gradient for various element types.

DFGRD1(3,3)

Array containing the deformation gradient at the end of the increment. The components of this array are set to zero if nonlinear geometric effects are not included in the step definition associated with this increment. See the discussion regarding the availability of the deformation gradient for various element types.

NOEL

Element number.

NPT

Integration point number.

LAYER

Layer number (for composite shells and layered solids).

KSPT

Section point number within the current layer.

KSTEP

Step number.

KINC

Increment number.

Appendix III: Project planning chart and meetings protocol