NASA Contractor Report

Progressive Failure and Life Prediction of Ceramic and Textile Composites

David Y Xue, Yucheng Shi, Madhu Katikala, William M. Johnston, Jr.
and Michael F. Card
Analytical and Material Services
Hampton, Virginia

Contract NAS8-40638

March 1998

National Aeronautics and Space Administration
George C. Marshall Space Flight Center
Huntsville Alabama 35812
PROGRESSIVE FAILURE AND LIFE PREDICTION OF CERAMIC AND TEXTILE COMPOSITES

(CONTRACT: NAS8-40638)

(Final Report, March, 1998)

ABSTRACT

An engineering approach to predict the fatigue life and progressive failure of multilayered composite and textile laminates is presented. Analytical models which account for matrix cracking, statistical fiber failures and nonlinear stress-strain behavior have been developed for both composites and textiles. The analysis method is based on a combined micromechanics, fracture mechanics and failure statistics analysis. Experimentally derived empirical coefficients are used to account for the interface of fiber and matrix, fiber strength, and fiber-matrix stiffness reductions. Similar approaches were applied to textiles using Repeating Unit Cells. In composite fatigue analyses, Walker’s equation is applied for matrix fatigue cracking and Heywood’s formulation is used for fiber strength fatigue degradation.

The analyses have been compared with experiments with good agreement. Comparisons were made with Graphite-Epoxy, C/SiC and Nicalon/CAS composite materials. For textile materials, comparisons were made with triaxial braided and plain weave materials under biaxial or uniaxial tension. Fatigue predictions were compared with test data obtained from plain weave C/SiC materials tested at AS&M.

Computer codes were developed to perform the analyses. Composite Progressive Failure Analysis for Laminates is contained in the code CPFail. Micromechanics Analysis for Textile Composites is contained in the code MicroTex. Both codes were adapted to run as subroutines for the finite element code ABAQUS as CPFail-ABAQUS and MicroTex-ABAQUS. Graphic User Interface (GUI) was developed to connect CPFail and MicroTex with ABAQUS.
CONTENTS:

ABSTRACT

SUMMARY

Part A: CPFail Analysis
A-1. Background and Theory
A-2. CPFail Laminate Analysis
A-3. CPFail/ABAQUS Structural Progressive Failure Analysis

Part B: MicroTex Analysis
B-1. Background and Theory
B-2. MicroTex Laminate Analysis
B-3. MicroTex/ABAQUS Structural Progressive Failure Analysis

Part C: Experiments

Part D: Publications

Part E: Commercialization Plan

Part F: Computer Codes
This document is the final report for the SBIR project of NASA CONTRACT NAS8-40638, entitled ‘Progressive Failure and Life Prediction of Ceramic and Textile Composites’.

An engineering approach to predict the fatigue life and progressive failure of multilayered composite and textile laminates is developed. Analytical models which account for matrix cracking, statistical fiber failures and nonlinear stress-strain behavior have been developed for both composites and textiles. The analysis method is based on a combined micromechanics, fracture mechanics and failure statistics analysis. Experimentally derived empirical coefficients are used to account for the interface of fiber and matrix, fiber strength, and fiber-matrix stiffness reductions. Similar approaches were applied to textiles using Repeating Unit Cells. In composite fatigue analyses, Walker’s equation is applied for matrix fatigue cracking and Heywood’s formulation is used for fiber strength fatigue degradation.

The analyses have been compared with experiments with good agreement. Comparisons were made with Graphite-Epoxy, C/SiC and Nicalon/CAS composite materials. For textile materials, comparisons were made with triaxial braided and plain weave materials under biaxial or uniaxial tension. Fatigue predictions were compared with test data obtained from plain weave C/SiC materials tested at AS&M.

Progressive analyses for conventional composite materials and textile composites are named as CPFail (Composite Progressive Failure Analysis) and MicroTex (Micro-mechanics analysis for Textile composites).

The report contains three major parts: CPFail, MicroTex and Experiments. CPFail and MicroTex include Theoretical parts and user’s manuals for the corresponding software codes.

The final products contain four computer software codes with graphics user interfaces: (1) CPFail-ABAQUS, (2) CPFail, (3) MicroTex-ABAQUS and (4) MicroTex.

CPFail-ABAQUS is a code which combines ABAQUS structural analysis and CPFail material degradation analysis for conventional composite materials.

CPFail is an independent code of material degradation analysis for conventional composite materials.

MicroTex-ABAQUS is a code which combines ABAQUS structural analysis and MicroTex textile composite material degradation analysis.

MicroTex is an independent code for textile composite material degradation analysis.
Part A. CPFail (Composite Progressive Failure) Analysis

A-1. Background and Theory

**INTRODUCTION**

An engineering approach CPFail (Composite Progressive Failure Analysis) has been developed to predict composite laminate failure and fatigue life under tensile damage. The stiffness reduction of each layer is computed based on: 1) in the fiber direction, the calculated fiber failure ratio and composite fracture; 2) in the direction normal to the fiber, the estimated matrix cracking. The laminate property is affected by the stiffness reduction of each ply. The analysis consists of micro-mechanics, fracture mechanics and statistics principles to model and predict the mechanical response and failure mechanism of polymer and ceramic matrix composites. The technique uses experimentally derived empirical coefficients to account for the interface of fiber and matrix, fiber strength and fiber-matrix stiffness reductions. In fatigue analysis, the Walker fatigue equation is used for counting matrix fatigue damage and the Heywood fatigue formulation is applied to estimate the fiber strength reduction. Analytical results are generated for Graphite-Epoxy, Nicalon/CAS, and C/SiC and compared with experiments.

The advent of new ceramic materials suitable for high temperature applications makes the capability to predict strength from fundamental principles extremely important. Reinforced materials such as C/SiC and SiC/SiC are finding applications in newly proposed launch vehicles and high performance engines. High temperature induced thermal stress and cyclic loading play an important role in the damage of these new materials.

A number of mathematical models have been proposed for predicting damage in fiber-reinforced composites [1-8]. Composite laminate damage normally starts with matrix cracking in the direction normal to the fibers. The matrix cracking will reach a saturation level with the increase of the applied load or the number of loading cycles. The stiffness of the laminate is controlled by the fiber property of each layer, that is the number of undamaged fibers in the layer.

In this part, the basic principles of the analysis are described. A simplified method of predicting matrix cracking is compared with experimental results for graphite-epoxy laminates. Stress-strain predictions for Nicalon-CAS material are compared with experiments at two temperatures and a study of the strength of angle-plied material is presented. Finally stress-strain curve and fatigue S-N curve for 0-90 C/SiC material are presented and compared with experiment.

CPFail has been successfully combined into ABAQUS structural analysis. Thus complex structural progressive failure analysis can be performed with the CPFail damage criteria. A simple example of a tension plate with a center hole was demonstrated to show the progressive plate damage and stress release process.

**ANALYTICAL THEORY**

A previous study [1] has developed an analysis to predict the fiber breakage in a single composite layer under tension loads. For continuously reinforced ceramic matrix composites, the failure mechanisms involve fiber/matrix debonding, fiber-bridged matrix cracking, and fiber failure in the wake of matrix cracking. In the analysis, the fiber-bridged matrix crack was
idealized by a continuum model in which the effect of the bridging fibers was modeled by an equivalent closure pressure on the crack surface. A rigorous shear-lag model developed by Budiansky et al. [2] was used to evaluate the closure pressure distribution along the crack surface and to compute the shear stress transfer along the debonded, frictionally sliding fiber/matrix interface. A closed-form micro-mechanics analysis was used to compute the thermal and mechanical (axial and transverse) stress along the fiber/matrix interface region. The micro-mechanics, the shear-lag, and the continuum fracture mechanics models were integrated with a statistical model to predict the fiber failure ratio. Fiber failure and matrix cracking reduce the composite modules and number of intact fibers to the point of failure.

The present study improved the analysis procedure and applied the analysis to compute the behavior of multilayer composites. Experimental results showed inelastic strain can occur when the applied stress exceeds a “yielding stress”. A parameter related to the ply stress ratio is introduced to reduce the fiber and matrix stiffness after the “yielding”. Matrix stiffness reduction due to the cracking in the direction normal to the fibers is estimated according to the tensile stress in this direction. A stepwise loading procedure is required since the stress-strain curve is nonlinear due to the material degradation caused by fiber breakage, matrix cracking and inelastic yielding. The analysis steps are summarized in Figure A1-1.

1) Micromechanics analysis:

A three-phase (fiber-matrix-composite) micromechanics model (Fig. A1-1) was developed for computing the normal stresses $\sigma_{rr}$ at the fiber-matrix interface. Using classical elasticity solutions, the resulting 15 equations containing 15 unknown coefficients were solved numerically to obtain the solutions. The shear stresses, $\sigma_{\theta\theta}$ and $\sigma_{r\theta}$ were assumed to be zero. The computed interface normal stress will be used in the next step of shear-lag analysis.

Using a classical elasticity approach, the following expressions can be assumed for the stresses in fiber (f), matrix (m) and composites (c).

\[
\begin{align*}
\sigma^f_{rr} &= [2B_f] + [-2D_f] \cos(2\theta) \\
\sigma^f_{\theta\theta} &= [2B_f] + [2D_f + 12E_f r^2] \cos(2\theta) \\
\sigma^f_{r\theta} &= [2D_f + 6E_f r^2] \sin(2\theta)
\end{align*}
\]

\[\text{(A1-1)}\]

\[
\begin{align*}
\sigma^m_{rr} &= \left[\frac{A_m}{r^2} + 2B_m\right] + \left[-2D_m - \frac{6F_m}{r^4} - 4 \frac{G_m}{r^2}\right] \cos(2\theta) \\
\sigma^m_{\theta\theta} &= \left[-\frac{A_m}{r^2} + 2B_m\right] + \left[2D_m + 12E_{mm} r^2 + 6 \frac{F_m}{r^4}\right] \cos(2\theta) \\
\sigma^m_{r\theta} &= \left[2D_m + 6E_{mm} r^2 - 6 \frac{F_m}{r^4} - 2 \frac{G_m}{r^2}\right] \sin(2\theta)
\end{align*}
\]

\[\text{(A1-2)}\]
Input element stresses, temperature and fiber/matrix properties

Micromechanics analysis

Shear-lag analysis

Fracture mechanics analysis

Estimate matrix stiffness reduction due to matrix cracking

Statistics analysis for failed fiber fraction

Update ply and laminate properties

Fig. A1-1. Flow-chart of combined micromechanics, fracture mechanics and statistical approach of composite damage prediction
The constitutive relation in fiber and matrix is assumed to be the same as of isotropic materials,

\[
\begin{align*}
\{\varepsilon_{rr} & \quad \varepsilon_{\theta\theta} & \quad \varepsilon_{zz} & \quad \gamma_{r\theta}\} = \\
\begin{bmatrix}
1 & -\mu & -\mu & 0 \\
1/E & 1/E & 1/E & 0 \\
-\mu & 1/E & -\mu & 0 \\
-\mu & -\mu & 1/E & 0 \\
0 & 0 & 0 & 2(1+\mu)/E \\
\end{bmatrix}
\begin{bmatrix}
\sigma_{rr} \\
\sigma_{\theta\theta} \\
\sigma_{zz} \\
\sigma_{r\theta} \\
\end{bmatrix}
+ \\
\begin{bmatrix}
0 \\
0 \\
0 \\
0 \\
\end{bmatrix}
\end{align*}
\]

(A1-4)

The constitutive relation of the composite material is,

\[
\begin{align*}
\{\varepsilon_{rr} & \quad \varepsilon_{\theta\theta} & \quad \varepsilon_{zz} & \quad \gamma_{r\theta}\} = \\
\begin{bmatrix}
1 & -\mu_{223} & -\mu_{21} & 0 \\
E_{22} & E_{22} & E_{22} & 0 \\
-\mu_{223} & E_{22} & E_{22} & 0 \\
-\mu_{12} & -\mu_{12} & E_{11} & 0 \\
0 & 0 & 0 & 2(1+\mu_{223})/E_{22} \\
\end{bmatrix}
\begin{bmatrix}
\sigma_{rr} \\
\sigma_{\theta\theta} \\
\sigma_{zz} \\
\sigma_{r\theta} \\
\end{bmatrix}
+ \\
\begin{bmatrix}
\alpha_{22} \Delta T \\
\alpha_{22} \Delta T \\
\alpha_{11} \Delta T \\
\alpha_{11} \Delta T \\
\end{bmatrix}
\end{align*}
\]

(A1-5)

with the strain-displacement relation,

\[
\begin{align*}
\varepsilon_{rr} &= \frac{\partial u_r}{\partial r}, \quad \varepsilon_{\theta\theta} = \frac{1}{r} \frac{\partial u_\theta}{\partial \theta} + \frac{u_r}{r}, \quad \gamma_{r\theta} = \frac{1}{r} \frac{\partial u_r}{\partial \theta} + \frac{\partial u_\theta}{\partial r} - \frac{u_\theta}{r} \\
\end{align*}
\]

(A1-6)

The displacement in fiber, matrix and composite can be written as,
\[ u_r^f = \left( \frac{1 + \mu_f}{E_f} \right) \left[ \frac{2(1 - 2\mu_f)B_f r}{r} + \left( -2D_f r - 4\mu_f E_{rr} r^3 \right) \cos(2\theta) \right] + \left[ (1 + \mu_f) \alpha_f \Delta T - \mu_f \overline{e}_{zz} \right] r \]

\[ u_\theta^f = \left( \frac{1 + \mu_f}{E_f} \right) \left[ 2D_f r + 2(3 - 2\mu_f) E_{rr} r^3 \right] \sin(2\theta) \]

\[ u_r^m = \left( \frac{1 + \mu_m}{E_m} \right) \left[ \frac{A_m}{r} + 2(1 - 2\mu_m)B_m r \right] + \left[ -2D_m r - 4\mu_m E_{mm} r^3 + 2 \frac{F_m}{r^3} + \frac{4(1 - \mu_m)G_m}{r} \right] \cos(2\theta) \]

\[ + \left[ (1 + \mu_m)\alpha_m \Delta T - \mu_m \overline{e}_{zz} \right] r \]

\[ u_\theta^m = \left( \frac{1 + \mu_m}{E_m} \right) \left[ 2D_m r + 2(3 - 2\mu_m) E_{mm} r^3 + 2 \frac{F_m}{r^3} - \frac{2(1 - 2\mu_m)G_m}{r} \right] \sin(2\theta) \]

\[ u_r^c = \left( \frac{1}{E_{22}} \right) \left[ -\left( 1 + \mu_{23} \right) A_c \frac{r}{r} + 2(1 - \mu_{23} - 2\mu_{21})B_c r \right] \]

\[ + \left[ -2(1 + \mu_{23})D_c r + 2 \left( \frac{1 + \mu_{23}}{r^3} \right) F_c + 4 \frac{(1 - \mu_{23} - 2\mu_{21})G_c}{r} \right] \cos(2\theta) \]

\[ + \left[ (\mu_{23} \alpha_{11} + \alpha_{22}) \Delta T - \mu_{12} \overline{e}_{zz} \right] r \]

\[ u_\theta^c = \left( \frac{1}{E_{22}} \right) \left[ 2(1 + \mu_{23})D_c r + 2 \left( \frac{1 + \mu_{23}}{r^3} \right) F_c - \frac{2(1 - \mu_{23} - 2\mu_{21})G_c}{r} \right] \sin(2\theta) \]

Using the following 15 boundary conditions, Eqs. (A1-10),

\[ \sigma_{rr}^f (a, 45^0) = \sigma_{rr}^m (a, 45^0) \]
\[ \sigma_{rr}^f (a, 0^0) = \sigma_{rr}^m (a, 0^0) \]
\[ \sigma_{r\theta}^f (a, 45^0) = \sigma_{r\theta}^m (a, 45^0) \]
\[ \sigma_{rr}^m (b, 45^0) = \sigma_{rr}^c (b, 45^0) \]
\[ \sigma_{rr}^m (b, 0^0) = \sigma_{rr}^c (b, 0^0) \]
\[ \sigma_{r\theta}^m (b, 45^0) = \sigma_{r\theta}^c (b, 45^0) \]
\[ u_r^f (a, 45^0) = u_r^m (a, 45^0) \]
\[ u_\theta^f (a, 45^0) = u_\theta^m (a, 45^0) \]
\[ u_r^f (a, 0^0) = u_r^m (a, 0^0) \]
\[ u_c^m(b,45^0) = u_c^c(b,45^0) \]
\[ u_c^m(b,0^0) = u_c^c(b,0^0) \]
\[ u_c^m(b,0^0) = u_c^c(b,0^0) \]
\[ \sigma_{rr}^m(\infty,0^0) = \sigma_{rr}^c(\infty,0^0) \]
\[ \sigma_{\theta\theta}^m(\infty,0^0) = \sigma_{\theta\theta}^c(\infty,0^0) \]
\[ \frac{1}{\pi b^2} \left[ \int_{\text{fiber}} \sigma_{zz}^r r d\theta + \left[ \int_{\text{matrix}} \sigma_{zz}^{nm} r d\theta + \right] \right] = \sigma_{zz}^{\infty} \]

(A1-10)

15 unknown coefficients \( B_f, D_f, E_f, A_m, B_m, D_m, E_{mm}, F_m, G_m, A_c, B_c, D_c, F_c, G_c \) and \( \varepsilon_{zz} \) can be solved from Eqs. (A1-11)

\[ 2B_f - \frac{A_m}{a^2} - 2B_m = 0 \]
\[ -2D_f + 2D_m + \frac{F_m}{a^4} + 4 \frac{G_m}{a^2} = 0 \]
\[ 2D_f + 6a^2 E_{ff} - 2D_m - 6a^2 E_{mm} + \frac{F_m}{a^2} + 2 \frac{G_m}{a^2} = 0 \]
\[ \frac{1}{b^2} A_m + 2B_m - \frac{1}{b^2} A_c - 2B_c = 0 \]
\[ -2D_m - \frac{6}{b^4} F_m - 4 \frac{G_m}{a^2} + 2D_c + 6 \frac{F_c}{a^4} + 4 \frac{G_c}{a^2} = 0 \]
\[ 2D_m + 6b^2 E_{mm} - \frac{6}{a^2} F_m - 2 \frac{G_m}{a^2} - 2D_c + 6 \frac{F_c}{a^4} + 2 \frac{G_c}{a^2} = 0 \]

\[ \frac{2(1 + \mu_f)(1 - 2\mu_f)a}{E_f} B_f + \frac{1 + \mu_m}{E_m a} A_m - \frac{2a(1 + \mu_m)(1 - 2\mu_m)}{E_m} B_m + a(\mu_m - \mu_f) \varepsilon_{zz} \]
\[ = \left[ (1 + \mu_f) \alpha_f - (1 + \mu_m) \alpha_m \right] \Delta T \]

\[ - \frac{2(1 + \mu_f)a}{E_f} D_f - \frac{2(1 + \mu_f)(3 - 2\mu_f)a^3}{E_f} E_{ff} + \frac{2(1 + \mu_m)a}{E_m} D_m \]
\[ + \frac{2(1 + \mu_m)(3 - 2\mu_m)a^3}{E_m} E_{mm} + \frac{2(1 + \mu_m)}{E_m a^3} F_m \]
\[ - \frac{2(1 + \mu_m)(1 - 2\mu_m)a^3}{E_m a} G_m = 0 \]

\[ - \frac{2(1 + \mu_f)a}{E_f} D_f - \frac{4\mu_f(1 + \mu_f)a^3}{E_f} E_{ff} + \frac{2(1 + \mu_m)a}{E_m} D_m \]
\[ + \frac{4\mu_m(1 + \mu_m)a^3}{E_m} E_{mm} - \frac{2(1 + \mu_m)}{E_m a^3} F_m - \frac{4(1 + \mu_m)(1 - \mu_m)a^3}{E_m a} G_m \]
\[ = 0 \]
\[
\begin{align*}
\frac{1 + \mu_m}{E_m} A_m + \frac{2(1 - 2\mu_m)(1 + \mu_m)b}{E_m} B + \frac{1 + \mu_{23}}{E_{22} b} A_c \\
- \frac{2(1 - \mu_{23} \mu_{23})b}{E_{22}} B_c + (\mu_{12} - \mu_m)b e_{zz} \\
= b[\mu_{12} \alpha_{11} + \alpha_{22} - (1 + \mu_m)\alpha_m] \Delta T
\end{align*}
\]

\[
\begin{align*}
\frac{2(1 + \mu_m)b}{E_m} D_m - \frac{4\mu_m(1 + \mu_m)b^3}{E_m} E_{mm} + \frac{2(1 + \mu_m)}{E_m b^3} F_m + \frac{4(1 - \mu_m)(1 + \mu_m)}{E_m b} G_m \\
+ \frac{2(1 + \mu_{23} b)}{E_{22}} D_c - \frac{2(1 + \mu_{23}) F_c}{E_{22} b^3} - \frac{4(1 - \mu_{12} \mu_{21})}{E_{22} b} G_c
\end{align*}
\]

\[
= 0
\]

\[
\begin{align*}
\frac{2(1 + \mu_m)b}{E_m} D_m + \frac{2(1 + \mu_m)(3 - 2\mu_m)b^3}{E_m} E_{mm} + \frac{2(1 + \mu_m)}{E_m b^3} F_m - \frac{2(1 - 2\mu_m)(1 + \mu_m)}{E_m b} G_m \\
- \frac{2(1 + \mu_{23}) b}{E_{22}} D_c - \frac{2(1 + \mu_{23}) F_c}{E_{22} b^3} + \frac{2(1 - \mu_{23} - \mu_{12} \mu_{21})}{E_{22} b} G_c
\end{align*}
\]

\[
= 0
\]

\[
2B_c - 2D_c = \sigma_{\alpha}
\]

\[
2B_c + 2D_c = \sigma_{\alpha 90}
\]

\[
4a^2 \mu_f B_f + 4(b^2 - a^2) \mu_m B_m + \left[E_f a^2 + (b^2 - a^2)E_m \right] e_{zz}
\]

\[
= b^2 \sigma_{\alpha} + [\alpha_f E_f a^2 + \alpha_m E_m (b^2 - a^2)] \Delta T
\]

(A1-11)

2) Shear-lag analysis:

A shear-lag model (Fig. A1-1) was used to model each fiber along the matrix crack and the slip region at the fiber-matrix interface. The interface shear stress \( \tau \) is related to the normal interface stress \( \sigma_n \) through the friction coefficient \( \mu \), as:

\[
\tau = \mu \sigma_n \mid_{\text{shear}}
\]

(A1-12)

where \( \mu \) is a empirical parameter, needs to be determined from experiments or numerical estimation.

The crack opening displacement, \( u(x) \), can be expressed by

\[
u(x) = AT(x)^2 + BT(x) + C
\]

(A1-13)

where \( T(x) \) is the fiber bridging traction, \( A, B \) and \( C \) are functions of interface shear stress \( \tau \) and applied loads. Both \( u(x) \) and \( T(x) \) are unknown functions, and require a fracture mechanics analysis to solve the problem.
3) Fracture Mechanics Analysis:

In the fracture mechanics analysis (Fig. A1-1), the crack opening displacement, \( u(x) \), can be computed by another form, as:

\[
\begin{align*}
\frac{2c\sigma_c}{\pi E_c} \left[ 1 - \frac{p(t)}{\sigma_c} \right] \ln \left| \frac{\sqrt{1-t^2} + \sqrt{1-x^2}}{\sqrt{1-t^2} - \sqrt{1-x^2}} \right| dt
\end{align*}
\]

(A1-14)

where \( c \) is the initial crack length, estimated according to the size of the fiber diameter and fiber volume fraction [1], \( \sigma_c \) is the applied stress and \( E_c \) is the composite stiffness in the fiber direction.

The bridging pressure, \( p(x) \), and the bridging traction \( T(x) \) are related as:

\[
p(x) = V_f T(x)
\]

(A1-15)

where \( V_f \) is the fiber volume friction. The bridging traction \( T(x) \) then can be solved by using an iterative procedure from the following integral equation:

\[
AT(x)^2 + BT(x) + C = \frac{2c\sigma_c}{\pi E_c} \left[ \pi \sqrt{1-x^2} \right] \frac{V_f}{\sigma_c} \ln \left| \frac{\sqrt{1-t^2} + \sqrt{1-x^2}}{\sqrt{1-t^2} - \sqrt{1-x^2}} \right| dt
\]

(A1-16)

After solving for the bridging pressure, the composite stress intensity factor, \( K \), at the fiber-bridged crack can be calculated as:

\[
K = 2\sqrt{\frac{c}{\pi}} \int_0^1 \left( \frac{\sigma_c - p(x)}{\sqrt{1-x^2}} \right) dx
\]

(A1-17)

The matrix stress intensity factor, \( K_m \), is:

\[
K_m = K \sqrt{\frac{E_m}{V_m E_c}}
\]

(A1-18)

where \( E_m \) and \( V_m \) is the matrix stiffness and volume friction. Matrix crack propagation is predicted when

\[
K_m \geq K_{mc}
\]

(A1-19)

where \( K_{mc} \) the critical matrix stress intensity factor should be determined from experiments or the literature.

4) Statistical Fiber Failure:

From the shear-lag model, the fiber failure statistical value can be found as:
where \( P_f \) is the fiber failure probability, \( M \) is Weibull modules, \( T_z = \frac{\sigma}{V_f} \), \( V_i \) is the initial fiber volume fraction, \( \sigma_0 \) is the equivalent fiber strength, an empirical parameter with the units MPa or KSi, \( k_0 \) is a unit parameter. \( k_0 = 1 \ m^2 \), for MPa units; \( k_0 = 1550 \ in^2 \), for KSi units. The predicted fiber failure probability of the composite is sensitive to the fiber strength properties. Therefore, an accurate knowledge of the \( \sigma_0 \) and \( M \) values for the fibers inside the matrix is necessary. Using the fiber original strength properties prior to the composite fabrication overestimates the strength prediction. In general, the mean strength of the fibers is reduced during composite fabrication [11].

The final fiber volume fraction can be estimated as:

\[
V_f = V_i (1 - P_f)
\]

YIELDING PARAMETER

An important improvement of this analysis in this study is to introduce a yielding parameter to control the stiffness changes of fiber and matrix after a ply passed its yielding point. In the current analysis it is known that a single ply starts yielding when the matrix stress intensity factor in the fiber direction exceeds its critical value, \( K_a > K_{mc} \). To reduce the dependence of the solution on the load steps, the parameter is designed to be related to the ratio of the current ply stress (in the fiber direction) over the critical ply stress. The critical ply stress is the ply stress at yielding. The reduced stiffness is assumed to be:

\[
E = E_0 \left(\frac{\sigma_z}{\sigma_{zc}}\right)^\beta
\]

where \( E_0 \) is the original, undamaged fiber or matrix stiffness. The yielding parameter \( \beta \) needs to be determined according to the composite material testing data. Basically \( \beta \) will control the slope of stress-strain curve after the yielding point.

MATRIX DAMAGE NORMAL TO THE FIBERS

A simplification of matrix stiffness reduction is based on the fact that 1) the fiber modules dominates the stiffness of the laminate, and 2) matrix cracking reaches its saturation level at the very early (low) loading stages. In the analysis the matrix elastic modules was reduced to 30% of the original value when the matrix fracture stress intensity factor reached the critical value. A linear matrix stiffness reduction was assumed before the critical stress intensity level was reached.

The matrix cracking in the off-fiber direction will be estimated according to the tension stress level in that direction. Assume the size of initial matrix cracking is same as the size \( c \) estimated in the fiber direction, the matrix stress intensity factor then can be derived by setting \( p(x) = 0 \) in Eq. (A1-17), as
\[ K_{m0} = \sigma_{c0} \sqrt{\pi c} \]  \hspace{1cm} (A1-23)

where \( \sigma_{c0} \) is the stress in the off-fiber direction. Assume that if \( K_{m0} \) is larger than \( K_{mc} \), matrix cracking reaches a saturation level, the Young's Modules of matrix \( E_m \) reduces to 30% of its original value. Before that level \( E_m \) has a linear decline:

\[ E_m = E_{m0} \delta, \quad \text{if} \quad K_{m0} \geq K_{mc}; \quad (A1-24) \]

\[ E_m = E_{m0} \left[ 1 - \frac{K_{m0}}{K_{mc}} (1 - \delta) \right], \quad \text{if} \quad K_{m0} \leq K_{mc} \quad (A1-25) \]

Since the fiber property dominates in the composite laminate response, the value of \( \delta \) is not critical to the laminate property. In the present study \( \delta = 0.3 \). Eqs. (A1-24) and (A1-25) greatly simplify the computation of off-fiber matrix cracking. Comparison with experimental results showed that the simplification was reasonable.

**FAILURE CRITERIA**

Two criteria were used to test for ply failure:

1) \( V_f = V_{lim, p} \) \hspace{1cm} (\( V_{lim, p} = 0.1 - 0.3 \)) in any ply, or

2) \( K_{m0} \geq K_{mc} \) in all plies

If either condition was met, the laminate was assumed to have failed.
EMPIRICAL PARAMETERS

The analytical theory contains four empirical parameters, $\mu$ (Eq. A1-12), $\beta$ (Eq. A1-22), $K_{nc}$ (Eq. A1-19), and $\sigma_0$ (Eq. A1-20). As indicated in Figure A1-2, they can be used to adjust the shape of the stress-strain curve.

![Diagram showing effects of empirical parameters](image)

Fig. A1-2  Effects of empirical parameters

CPFail FATIGUE ANALYSIS MODEL

The fatigue analysis includes matrix cracking fatigue and fiber fatigue since the static analysis includes matrix and fiber damages.

The matrix cracking fatigue is counted by using a modified Walker fatigue equation:

$$\frac{da}{dN} = C K_m^n$$

where $da$ is the crack increase, $N$ is the number of loading circles, $K_m$ is the stress intensity factor associated with the maximum stress of the loading circle. $C$ and $n$ are empirical parameters.

In the current analysis, the failure of the fiber is controlled by the equation (A1-20) as:

$$P_f = 1 - \exp \left[ -\frac{4\pi a_i}{k_0} \left( \frac{T_c}{\sigma_0} \right)^M \right]$$

(A1-20)
where \( \sigma_0 \) is the equivalent fiber strength. The fatigue of fiber breakage is assumed to be developed with the reduction of \( \sigma_0 \). To estimate the fatigue reduction of the equivalent strength \( \sigma_0 \), a fatigue formulation (Heywood) was introduced as:

\[
\sigma_a = \pm \sigma_{t} \left[ 1 - \frac{\sigma_m}{\sigma_{t}} \left( A_0 + \gamma (1 - A_0) \right) \right] \\
A_0 = \left[ \frac{1 + \frac{p_3 \bar{n}^4}{1 + \sigma_{t}/p_1}}{1 + p_3 \bar{n}^4} \right]^{-1} \\
\gamma = \frac{\sigma_m}{\sigma_{t}} \frac{1}{1 + \left( \frac{\sigma_{t} \bar{n}}{p_2} \right)^4}, \\
\bar{n} = \log(N)
\]

(A1-27)

where

- \( \sigma_a \) is the Alternating stress,
- \( \sigma_m \) is the Mean stress, \( \sigma_m = 0.5(\sigma_{\max} + \sigma_{\min}) = 0.5(1 + R_m)\sigma_{\max} \),
- \( \sigma_t \) is the Tension strength,
- \( R_m \) is the stress ratio, \( R_m = \frac{\sigma_{\min}}{\sigma_{\max}} \).

The parameters \( p_1, p_2 \) and \( p_3 \) are material related, and for aluminum alloy \( p_1 = 153.26 \) MPa, \( p_2 = 2206.9 \) MPa and \( p_3 = 0.0031 \). The following figure (Figure A1-3) shows the effects of parameter changes on the Heywood formulation.

For a given maximum stress, strength, and stress ratio, a relation of alternating stress and the number of the load circles (S-N curve) can be established. The region under the S-N curve is the safe area, so the S-N curve actually represents a fatigue strength curve. In the current analysis, assuming \( \sigma_t = \sigma_0 = \sigma_{\max} \), the result curve represents fatigue reduction of \( \sigma_0 \). Considering the interface of matrix fatigue and fiber fatigue, the final reduced strength is assumed to be:

\[
\sigma_0 = (\sigma_a + \sigma_m)R_k^{-\zeta}
\]

(A1-28)

where \( R_k = K_m / K_{nc} \) is the ratio of matrix stress intensity factor. \( \zeta \) is an empirical parameter.
ANALYSIS RESULTS

MATERIAL PROPERTIES

To verify the developed analysis tool several kinds of composite materials, Graphite/epoxy, Nicalon/CAS, C/SiC and SiC/SiC have been chosen for the comparison between analyses and test. Following are the material properties used in the present study. ($E_f$ - fiber young's modules, $\mu_f$ - fiber Poisson's ratio, $\alpha_f$ - fiber thermal expansion coefficient, $V_f$ - fiber volume friction, $\sigma_0$ - fiber mean strength, $d_f$ - fiber diameter, $M$ - fiber Weibull modules, $E_m$ - matrix Young's modules, $\mu_m$ - matrix Poisson's ratio, $\alpha_m$ - matrix thermal expansion coefficient, $K_{mc}$ - critical matrix stress intensity factor, $T$ - temperature).

Nicalon/CAS

Nicalon fiber:

- $E_f = 195.0$ (176.0)* GPa
- $\mu_f = 0.2$
- $\alpha_f = 3.0E-6$ (4.0E-6)* /°C
- $V_f = 0.65$
- $M = 10$
- $\sigma_0 = 388.4$ MPa

Figure A1-3. Parameter effects on the curve of Log(N) Vs. Maximum Stress (MPa)
\[ d_f = 7.00 \text{E-5} \text{ m} \]

CAS matrix:
\[
\begin{align*}
E_m &= 95.0 \text{ GPa} \\
\mu_m &= 0.3 \\
\alpha_m &= 5.0 \text{E-6} /{ }^0\text{C} \\
K_{nc} &= 2.16(1.73)^* \text{ MPa-m}^{(1/2)} \\
T &= 0 \text{ }^0\text{C} \\
\end{align*}
\]
(* Properties at \( T=400 \text{ }^0\text{C} \))

Empirical parameters:
\[
\begin{align*}
\mu &= 0.01 \text{ Friction parameter} \\
\beta &= 2.4 \text{ Nonlinear parameter} \\
\delta &= 0.3 \text{ Matrix damage limit} \\
\end{align*}
\]

Graphite/Epoxy
Graphite fiber:
\[
\begin{align*}
E_f &= 221.0 \text{ GPa} \\
\mu_f &= 0.48 \\
\alpha_f &= -7.58 \text{E-7} /{ }^0\text{C} \\
V_f &= 0.65 \\
M &= 10 \\
\sigma_0 &= 388.4 \text{ MPa} \\
d_f &= 7.00 \text{E-6} \text{ m} \\
\end{align*}
\]

Epoxy matrix:
\[
\begin{align*}
E_m &= 3.45 \text{ GPa} \\
\mu_m &= 0.2 \\
\alpha_m &= 8.433 \text{E-5} /{ }^0\text{C} \\
K_{nc} &= 0.67 \text{ MPa-m}^{(1/2)} \\
T &= 0 \text{ }^0\text{C} \\
\end{align*}
\]

Empirical parameters:
\[
\begin{align*}
\mu &= 0.01 \text{ Friction parameter} \\
\beta &= 2.4 \text{ Nonlinear parameter} \\
\delta &= 0.3 \text{ Matrix damage limit} \\
\end{align*}
\]

Carbon/SiC
Carbon fiber:
\[
\begin{align*}
E_f &= 231.0 \text{ GPa} \\
\mu_f &= 0.2 \\
\alpha_f &= -6.0 \text{E-5} /{ }^0\text{C} \\
\end{align*}
\]
\[ V_f = 0.40 \]
\[ M = 10 \]
\[ \sigma_0 = 388.4 \text{ MPa} \]
\[ d_f = 7.00 \times 10^{-6} \text{ m} \]

SiC matrix:
\[ E_m = 35.10 \text{ GPa} \]
\[ \mu_m = 0.22 \]
\[ \alpha_m = 1.10 \times 10^{-5} \text{ /°C} \]
\[ K_{nc} = 0.5 \text{ MPa}-m^{1/2} \]
\[ T = 0 \text{ °C} \]

Empirical parameters:
\[ \mu = 0.3 \text{ Friction parameter} \]
\[ \beta = 0.1 \text{ Nonlinear parameter} \]
\[ \delta = 0.1 \text{ Matrix damage limit} \]

**Laminate Response**

Figure A1-4 (taken from Reference 5) shows a schematic of the expected three stages of laminate response: 1) an initial rapid matrix cracking, 2) a stable period of matrix cracking saturation, and 3) fast fiber breakage. The present analysis models matrix cracking and fiber breakage and accounts for delamination and crack-coupling/interfacial debonding in the empirical parameters.

![Fig. A1-4](image_url)  
Damage development in composite laminates (Ref. 5)
Stiffness Degradation

To evaluate the accuracy of the stiffness degradation predictions comparisons were made with the test data given in Reference 3 for graphite-epoxy material. The present results and the experimental results are compared in Figure A1-5. The reduction in the axial laminate modules for a strain level of 0.01 is presented for several laminates. Agreement with test is reasonably good.

![Graphite/Epoxy Stiffness Reduction Prediction Comparison](Fig. A1-5)

Nicalon/CAS Comparison

Unidirectional ply stress-strain curves were computed for a Nicalon/CAS composite investigated experimentally in Reference 9. Data was obtained at room temperature (0 °C) and 400 °C. The solid and dashed stress-strain curves from analyses are compared in Fig. A1-6 and show good agreement with the testing data.

Room temperature angle ply laminate results for Nicalon/CAS material are plotted in Figure A1-7. The higher strength results for 15 and 30 degree laminates are unexpected, and are believed to have resulted from the absence of shear effects in the current analysis.

The complexity of the room temperature response of a (0/-45/45/90), Nicalon/CAS laminate is illustrated in Figures A1-8 and A1-9. The stress-strain curve in Figure A1-6 can be seen to have a discontinuity corresponding to yielding behavior. Figure A1-9 shows the reduction in stiffness of the laminate and several areas where changes in ply behavior are occurring.

C/SiC Results

Room temperature stress-strain and stiffness reduction for (0/90)s C/SiC material are presented in Figure A-10 and A-11. The results are compared with experimental results. The calculations appear to have reasonable agreement with the test data.
Fig. A1-6  Comparison of stress strain results for the Nicalon/CAS composite

Fig. A1-7  Stress-strain relation of angle ply laminates
Fig. A1-8  Stress-strain relation of Nicalon/CAS (0/45/45/90)s laminate under tension

Fig. A1-9  Stiffness reduction of Nicalon/CAS (0/45/45/90) laminate under tension
Fig. A1-10  Stress-strain relation of a C/SIC laminate

Fig. A1-11  Stiffness reduction of a C/SIC laminate
C/SiC Fatigue Results

A comparison (Figure A-12) is made for analysis and test results for a C/SiC specimen. The testing data is from NASA Marshall Center and the analysis is made by applying the CPFail (Composite Progressive Failure) fatigue analysis. The stress ratio is the ratio of applied maximum stress ($\sigma_{\text{max}}$) divided by the static failure stress.

Load ratio, $R_m = \frac{\sigma_{\text{min}}}{\sigma_{\text{max}}} = 0.1$.

The fatigue empirical parameters are:
- Walker constants: $A = 3.13 \times 10^{-15}$, $\alpha = 8.4$
- Heywood parameters: $p_1 = 0.153$ MPa, $p_2 = 2206.9$ MPa, $p_3 = 0.0031$
- Fatigue coefficient: $\zeta = 0.075$

![Figure A1-12. C/SiC fatigue result comparison](image-url)
ABAQUS-CPFai1 Structural Progressive Failure Analysis

Composite structural progressive failure has been demonstrated by integrating the material damage criteria developed in CPFail, and the ABAQUS's user defined material subroutine and step load increment analysis capability.

A 20-ply tensile test specimen containing a centrally located circular hole is being used to investigate damage progression. The composite laminate is 8-inches long and 1-inch wide with a hole diameter of 0.25 inches. The thickness of each ply is 0.00515 inches, and the laminate stacking sequence is \([0/(+45/-45)_3/90_3]_s\). The specimen is fabricated from T300/1034-C graphite/epoxy. A finite element mesh for the analysis is shown in the following figure. The sides of the laminate are free, and the loaded ends are clamped. Load is introduced by displacing the ends. The lamina properties for this laminate are:

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Longitudinal Young's modules</td>
<td>(E_{11} = 21300) ksi</td>
</tr>
<tr>
<td>Transverse Young's modules</td>
<td>(E_{22} = 1650) ksi</td>
</tr>
<tr>
<td>Poisson's ratio</td>
<td>(v_{12} = 0.3)</td>
</tr>
<tr>
<td>In-plane shear modules</td>
<td>(G_{12} = 897) ksi</td>
</tr>
</tbody>
</table>

The model was created from PATRAN, then translated as an ABAQUS input file. Totally it has 768 elements and 844 nodes, in order to simplify the model, each element has 8 layers with different thicknesses. Using S4R element, each element has 4 integration points. The material property recalculation will be performed in every layer and integration point.

Results are shown in Figures AA-1 ~ AA-10. Figures (a) are for stiffness (\(E_{11}\)) damage distribution, and Figures (b) are for stress (\(S_{11}\)) distribution. \(E_{11}\) and \(S_{11}\) are in the local fiber directions. Red color shows relatively high value and blue color shows relatively low value. It can be seen that with the loading increase, the stress concentration locations moved out of hole edge, as the material damage developed. This stress release due to local material damage is an important phenomenon in structural failure studies. Figures AA-1 ~ AA-10 are for 0 degree layer behavior. The 45 degree layer behavior are shown in Figures AA-11 and AA-12.
Figure AA-1 (a). Stiffness E11 distribution for 0 degree layer at step 1

Figure AA-1 (b). Stress S11 distribution for 0 degree layer at step 1
Figure AA-2 (a). Stiffness $E_{11}$ distribution for 0 degree layer at step 2

Figure AA-2 (b). Stress $S_{11}$ distribution for 0 degree layer at step 2
Figure AA-3 (a). Stiffness $E_{11}$ distribution for 0 degree layer at step 3

Figure AA-3 (b). Stress $S_{11}$ distribution for 0 degree layer at step 3
Figure AA-4 (a). Stiffness E11 distribution for 0 degree layer at step 4

Figure AA-4 (b). Stress S11 distribution for 0 degree layer at step 4
Figure AA-5 (a). Stiffness E11 distribution for 0 degree layer at step 5

Figure AA-5 (b). Stress S11 distribution for 0 degree layer at step 5
Figure AA-6 (a). Stiffness $E_{11}$ distribution for 0 degree layer at step 6

Figure AA-6 (b). Stress $S_{11}$ distribution for 0 degree layer at step 6
Figure AA-7 (a). Stiffness E11 distribution for 0 degree layer at step 7

Figure AA-7(b). Stress S11 distribution for 0 degree layer at step 7
Figure AA-8 (a). Stiffness E11 distribution for 0 degree layer at step 8

Figure AA-8 (b). Stress S11 distribution for 0 degree layer at step 8
Figure AA-9 (a). Stiffness E11 distribution for 0 degree layer at step 9

Figure AA-9 (b). Stress S11 distribution for 0 degree layer at step 9
Figure AA-10 (a). Stiffness E11 distribution for 0 degree layer at step 10

Figure AA-10 (b). Stress S11 distribution for 0 degree layer at step 10
Figure AA-11 (a). Stiffness $E_{11}$ distribution for 45 degree layer at step 2-3.

Figure AA-11 (b). Stress $S_{11}$ distribution for 45 degree layer at step 2-3.
Figure AA-12 (a). Stiffness E11 distribution for 45 degree layer at step 6

Figure AA-12 (b). Stress S11 distribution for 45 degree layer at step 6
REFERENCES


ABAQUS - CPFail

Composite Progressive Failure (CPFail) Analysis with ABAQUS using a combined micromechanics, fracture mechanics and statistics failure approach

Version 1.0
Copyright (c) 1998
AS&M, Inc.
107, Research Drive
Hampton, VA - 23666
CONTENTS

I. Preface
   I-1) Introduction
   I-2) System requirement
   I-3) User knowledge requirement
   I-4) Installation and execution
   I-5) Composite material degradation capabilities
   I-6) Limitations

II. Getting Started
   II-1) Preparing an initial ABAQUS input file
   II-2) Using GUI to input material parameters and rewrite cpf-1.inp file for CPFail-ABAQUS analysis

III. GUI Help
   III-1) Initial ABAQUS file
   III-2) [File]
   III-3) [Model]
   III-4) [Database]
   III-5) [Options]
   III-6) [Abaqus]
   III-7) How to adjust parameters

IV. Example
   IV-1) Initial and final ABAQUS runstream files:
   IV-2) Material properties and parameters
   IV-3) Progressive Damage
I. Preface

I-1) Introduction

CPFail is an analysis code for computing the composite material degradation due to tensile damage. CPFail-ABAQUS combines the material degradation with ABAQUS structural analysis through the ABAQUS user defined material entry UMAT. With a step loading procedure, it can show the structural progressive failure process or predict the structure fatigue life.

A user friendly GUI (Graphic User Interface) is designed for helping user to input material information and use ABAQUS's UMAT with CPFail material degradation analysis.
I-2) System requirement

CPFail-ABAQUS can be run on PC or UNIX system. The system requirement for running CPFaiI-ABAQUS is the same as the requirement for running ABAQUS software.

Since the GUI is written by JAVA, it also requires JDK (Java Development Kit 1.1.3 or higher version) on the PC or UNIX system.

I-3) User knowledge requirement

User should be able to run ABAQUS standard linear static analysis and ABAQUS-POST.

User may not know how to use UMAT in ABAQUS.

User is required to input necessary material information and practice adjusting empirical parameters.

I-4) Installation, path and execution

Installation:

The software is distributed in tar format file asm.tar. The file contains all the required data files, source files and java class files required to execute the programs. To install the software components copy change directory to the destination directory and extract the files using the command: tar -xvf asm.tar. This command creates the ASM main directory and sub directories to copy the CPFail, MicroTex and ABAQUS interface programs.

The ABAQUS-CPFail is developed using Sun’s JDK 1.1.4. To execute these components Java runtime environment of version 1.1.3 or higher is required. Refer the JDK or JRE documentation for installation of Java.

Path:

After extracting the files, the following two path settings have to be completed.

1) The ABAQUS executable file with its path, xxx/xxx/xxx/abaqus, has to be set for GUI before running any sample problems. The path, xxx/xxx/xxx/, is the path where the ABAQUS software installed and abaqus is the ABAQUS executable file for UNIX version. This setting can be made by either manually editing the configuration file (abaqus.cfg) or by executing the program and modifying through the GUI using the menu command Configuration under Options menu.

2) In ABAQUS subroutines, SDVIDI and UMAT, user need to set the full path xxx/xxx/ASM/CPF/ABAQUS-CPF/pathname.dat by modifying the corresponding line in each subroutine. These two subroutines are included in the file user.rut under the directory of xxx/xxx/ASM/CPF/ABAQUS-CPF/.

Execution:

The executable commands for ABAQUS-CPFail are listed below:

```
  go to directory /ASM/CPF/ABAQUS-CPF
  execute java asm.abaqus.ABApp CMLIFE
```

Original code Files:

The following files are the original code files, user should carry over when change software installation directory, user can also carry over xxx.cml, xxx.ini and xxx.inp files for previous cases:
1-5) Composite material degradation capabilities

The degradation can be specified to be happened in the whole structure or in the part of
the structure with user defined areas (element groups) and layers.

The material degradation analysis will be performed at each element, each layer and each
integration point in the selected area. For large scale problem, it may necessary to specify part of
the structure to have CPFail degradation analysis.

More than one kind of material can be selected to have material degradation.

I-6) Limitations

Composite material degradation is caused by tensile damage with the effect of shear
stress.

Only applicable to ABAQUS shell elements.

Loading formats for static and fatigue progressive failure analyses are pre-designed in
GUI, and can not be changed, since the material degradation is loading path dependent. Step
linear analysis procedure is used, small load increments are necessary to catch material and
structure nonlinear behavior due to damage.

Geometry nonlinearity is not available for this progressive damage analysis.
II. Getting Started

II-1) Preparing an initial ABAQUS input file

a) The initial ABAQUS input file is an ABAQUS standard static analysis runstream file (for example, cpf-l.ini) for a structure analysis problem with structure FE model, loading and boundary conditions. This file should be pre-tested with no errors in ABAQUS execution. Use ABAQUS shell elements only. The initial file should be written in upcase except the names of linked files. No blank space after comma in the material and section definition lines.

b) If not whole structure, but only part of the structure or some of the material needs material degradation analysis, specify different element group and material names in these areas.

c) The materials which will not be degraded in the analysis should be completed in the initial ABAQUS runstream file, cpf-l.ini. GUI will not modify those materials.

d) The materials which will be degraded in the analysis can have dummy values in the initial file, and should be defined in a normal format, not defined in the format of user defined material. Their true properties and some additional material parameters will be inputted from GUI, and the corresponding ABAQUS user defined material format will be created by GUI.

e) GUI can modify laminate section and orientation settings in the initial file.

II-2) Using GUI to input material parameters and produce ABAQUS runstream file xxx.inp file for CPFail-ABAQUS analysis

a) Open CPFail-ABAQUS GUI window

b) Create material database:
   Go to [Database], select [Fiber] and [CPFail matrix] if user wants to input material properties from fiber and matrix properties; select [Ply] if user wants to input ply properties. (The material degradation analysis will be based on fiber and matrix properties, the program will convert ply property to fiber and matrix properties.)

c) Ply, Fiber or Matrix Database:
   * Define a new ID or select one from database. (The ID will be used in [Model]/[User Materials]).
   * Input or modify material properties. Refer to III-3(d) or III-7 for detailed material information.
   * input a name for user's own remarks

d) Close [Database] windows. Go to [File]/[New], in the Import Model window select the prepared initial runstream file cpf-l.ini, click [Open]. All material names defined in the initial file will be appeared in GUI.

e) Go to [Model]/[User Material], select the material name which need the degradation analysis and click [User Defined ?]. Select [Material Type] for material input format, 'Ply' or 'Matrix and Fiber'. Select material ID which is defined in [Database]. Click [Edit] for [Material Special Parameters], refer to III-3(d) and III-7 for the explanations of the special parameters.
f) Lamination modification

The GUI main window shows graphic laminate information for an element group selected in the [Element Group] dialog box. All element groups defined in the initial file, cpf-1.ini, will appear in the dialog box. User can modify the laminate by going to [Model]/[Geometry] and [Model]/[Orientations].

g) Select analysis

* Go to [Model]/[Control Parameters], fill [Title] with user’s remarks, select ‘Static Analysis’ or ‘Fatigue Analysis’.  
* Select damage output location: Element and Layer. This location should have user defined material.  
* Static Analysis: input the number of total analysis steps  
* Fatigue Analysis: input [Load Ratio] and [Maximum Cycles (Log N)]  
* Note: Load values are inputted in the initial file

h) Save file

Go to [File], click [Save] or [Save As] to save the change in a GUI format file, xxx.cml. This file can be reopen later by using [File]/[Open].

i) Execute analysis

Go to [Model]/[Analysis], chose or input a runstream file name, for example cpf-2.inp, in [File name], then click [Save]. The GUI will produce a ABAQUS runstream file, cpf-2.inp (and several internal files), execute cpf-2.inp for the structural analysis defined in cpf-1.ini with CPFail defined material degradation effects.  
Check cpf-2.sta, cpf-2.msg or cpf-2.log to find out if the ABAQUS analysis finished or not.

j) Results

* Go to [View]  
  Open ‘strain-stress’ for strain-stress at each steps (no last step) for static analysis; or at each Log(N) cycles for fatigue analysis, at selected output location  
  Open ‘Damage’ for material degradation information at selected output location  
* Use ABAQUS-POST to display stress, strain, displacement and other results.  
* Use ABAQUS-POST to display reduced material properties:  
  SDV31=E11  
  SDV32=E22  
  SDV34=G12

where E11, E22 and G12 are the ply Young’s modules in the ply local coordinates.

* For fatigue analysis, the number of Log N is corresponding to (step -3). So, to display results at cycle, Log N = 2, the corresponding ABAQUS step = 2+3 = 5.

k) Structural fatigue S-N curve

It should be pointed out that in the fatigue analysis the step number is for loading cycle, Log(N). In each fatigue analysis case, load will be a constant and the Log(N) of final fatigue cycle will be computed. Thus for one load level, it can only produce one point on the curve (S-N) of Maximum stress verses Fatigue cycle Log(N). In order to have a complete S-N curve, several load levels should be executed. The fatigue failure (final) cycle, Log(N) is printed in the stress-strain result file (s-s.dat) and can be read from [View] window.
III. GUI Help

III-1) Initial ABAQUS file

CPFail-ABAQUS GUI (Graphic User Interface) is designed to help user to input material parameters and use the proper UMAT entry and analysis steps in ABAQUS for progressive failure analysis. This GUI does not have the functions for structural modeling. So the structure model including element mesh, material, shell section, boundary, constraints and loading conditions should be pre-prepared in a file called initial file. The GUI will copy all structure model and material information from the initial file and can make changes only on (1) the material part to form user defined material entries, (2) shell sections and (3) loading format. The laminate which contains the user defined materials should be defined by using ‘**SHELL SECTION’ not ‘*SHELL GENERAL SECTION’. GUI will not change the material properties for which are not selected as user defined materials.

GUI will not debug the errors in the initial file.

In the initial file all the units should be in SI units. GUI will allow user to input material properties in US unit, but will convert them to SI units in ABAQUS runstream files. So, all ABAQUS results will also be read in SI units.

III-2) [File]

(a) [New]: Open a case from an initial xxx.inp ABAQUS file, for example, cpf.ini.

(b) [Open]: Open a case from a previously saved xxx.cml GUI file

(c) [Save]: Save the current modified case to the original xxx.cml file

(d) [Save As]: Save the current case to a new GUI file, for example, xxx-1.cml

(e) [Print]: Print out the lamination figure in the window

(f) [Exit]: Close GUI

III-3) [View]

(a) [Strain-Stress]: View file ‘s-s.dat’. This file presents strain vs. stress at each load step (no last step) for static analysis; or Log(N) vs. maximum stress for fatigue analysis.

(b) [Damage]: View file ‘out.volume’. This file presents material damage information by showing the remaining fiber volume fraction and matrix stiffness ratio.

(c) [Text File]: View text files

III-4) [Model]

(a) [Geometry]:

...
Modify section (lamination) properties. Suggest to choose ‘1’ integration point to reduce computation time.

(b) [Control Parameters]:
* Select damage output location: Element and Layer. Only one location for a structural analysis. Stress-strain, remaining fiber volume fraction and matrix stiffness ratio will be written out at this location. Whole structure damage status can be viewed by showing total stiffness distribution using ABAQUS-POST.
* Select static analysis steps. Increment load value = load/steps.
* Select fatigue analysis load factor and maximum loading cycles, Log(N)
  Load factor = Minimum load / Maximum load in a loading cycle.
* For both static and fatigue analyses load values are inputted from the initial file.

(c) [User Materials]
Select material name which will have degradation analysis

* Material Name: (should be in upcase) All material names defined in the initial file will be in the list. Also user can create new material name.
  Select the materials which will have degradation analysis, click [User Defined], (one by one).

* Material Type: Select material property input format
  ‘Matrix and Fiber’ --- input material properties from matrix and fiber properties
  ‘Ply’ --- input composite ply properties

* --- Material ID: Select material ID from the list which is created in [Database]

* Material Special Parameters: Empirical parameters, see (d) for details.

* Create NEW or DELETE existing materials in xxx.inp file.

(d) [Material Special Parameters]:
Material special parameters are designed for user to adjust material or laminate behavior due to damage, fatigue, delamination, material nonlinearity or plasticity.
* Matrix Damage Limit: matrix modules remaining ratio after matrix cracking, $E_m / E_{m0}$.
* Nonlinear Coefficient: Adjust for material nonlinear behavior due to yielding, plasticity, damage or delamination. The nonlinear effect is bigger for larger value, for example, 0.01 (linear) ~ 2.4 (nonlinear).
* Process Temperature: Composite laminate manufacture curing temperature.
* Walker Constant C and n: from Walker’s fatigue statistic equation for matrix fatigue cracking. Suggested value: C= 3.13E-15 and n=8.4
  \[ \frac{da}{dN} = CK_m^n \]
* Fatigue Coefficient: Empirical fatigue parameter. A smaller value corresponds for smaller damage and longer life. A positive value of (0.01 ~ 3.0).
* Fatigue Final Strain: The fatigue failure criteria to define fatigue life cycles. User assumes that the structure failed as the strain ($\varepsilon_{11}$) at the monitor point reaches this value.
* Heywood Parameters $p_1 \sim p_3$: from Heywood’s fatigue statistic equation for fiber strength fatigue reduction. The effects of $p_1 \sim p_3$ are shown in the following figure. Suggest value: $p_1 = 0.153$ MPa, $p_2 = 2206.9$ MPa, $p_3 = 0.0031$

\[
\sigma_a = \pm \sigma_t \left[ 1 - \frac{\sigma_m}{\sigma_t} \right] \left[ A_0 + \gamma(1 - A_0) \right]
\]

\[
A_0 = \left[ 1 + \frac{p_3 \bar{n}^4}{1 + \frac{p_3 \bar{n}^4}{1 + p_3 \bar{n}^4}} \right] \frac{1}{1 + p_3 \bar{n}^4},
\]

\[
\gamma = \frac{\sigma_m}{\sigma_t} \frac{1}{1 + \left( \frac{\sigma_m \bar{n}^4}{p_2} \right)},
\]

\[
\bar{n} = \log(N)
\]

(e) [Orientations]
Define new or modify old orientations according to ABAQUS format

(f) [Analyze]
* If the case is started from a previous xxx.cml file (when user opened the case), GUI will ask if user wants save the change (to xxx.cml).
* User should then give a runstream xxx.inp file name, for example, cpf-2.inp. User can create a new name or select one from the window list and overwrite it.
* Click [Save], GUI will create the runstream file and execute it.

III-4) [Database]

(a) User can build new or modify old matrix, fiber and ply property databases.

(b) Matrix Critical Stress Intensity Factor, $K_{mc}$, (0.1 – 5.0)

(c) Weibull Modules $(M)$ and Mean Fiber Strength $(\sigma_0)$
Empirical parameters from modified Weibull fiber damage statistic equation, $M=10$ and $\sigma_0 = 388.0$ MPa are default values.

\[
P_f = 1 - \exp \left[ \frac{-4\pi a l}{k_0} \left( \frac{T}{\sigma_0} \right)^M \right]
\]

(d) Friction Coefficient
Empirical parameter of matrix-fiber integration (0.01 – 1.0)

III-5) [Options]

(a) [Units]:
* Let user choose SI or US units.
* All values will be changed according to unit change in GUI and will be written in to ABAQUS runstream file.
* for running a analysis, user should choose the unit which was used in structural modeling, since GUI will not change any value in model.
* For building or modifying database, user can choose any units.

(b) [Configuration]:
User should set the path to indicate where the file abaqus.exe exits, for example, 
C:\ABAQUS\5_6_1\abaqus.exe

III-6) [Abaqus]
Execute an ABAQUS runstream without GUI modification.
This is an option to let user to rerun a final xxx.inp file or a manually modified xxx.inp file.

III-7) How to adjust parameters
In order to make this material degradation analysis tool more general to composite materials, several empirical parameters need to be adjusted according to different materials or different damage conditions. Default values provide a base for the parameter adjustments, so they should be kept for reference. The following figures shown the effects of parameters on material or structural behaviors.

(a) Static analysis:
* Kmc --- Matrix critical stress intensity factor. A larger value will move up 'yielding point'
* \( \beta \) --- Nonlinear coefficient (0.01 ~ 10). A larger value will have stronger nonlinear behavior.
* \( \sigma_0 \) --- Mean fiber strength. A larger value will move up the final failure stress
* \( \mu \) --- Friction coefficient (0.01 ~ 1.0).
(b) Fatigue analysis:

In addition to the parameters in static analysis, in fatigue analysis Walker fatigue parameters for matrix fatigue and Heywood fatigue parameters (p1 ~ p3) for fiber fatigue may need to be adjusted. The Heywood parameters have the effects on S-N curve as shown in the following figure.
IV. Example

IV-1) Initial and final ABAQUS runstream files:

(a) Initial ABAQUS file, cpf.ini, prepared by users

In the following initial file, the elements in group CMLF will have material degradation analysis, the material (USER) then can have dummy values, the real properties will be input through GUI. The element group UNDA will not have degradation analysis, so its material, ORIN, should be correctly defined here and GUI will not change it.

Write a simple static analysis step in the initial file. The real analysis steps will be input from GUI.

*HEADING
CPFail-ABAQUS, cpf.ini
*INCLUDE,INPUT=cpf.model
** END OF MESH GENERATION
*SHELL SECTION,ELSET=CMLF,COMPOSITE
.005,1,user,ORI0
.005,1,user,ORI0
*TRANSVERSE SHEAR STIFFNESS
5.0E4,5.0E4
*SHELL SECTION,ELSET=UNDA,COMPOSITE
.005,1,orin,ORI0
.005,1,orin,ORI90
*TRANSVERSE SHEAR STIFFNESS
5.0E4,5.0E4
*MATERIAL,NAME=orin
*ELASTIC,TYPE=LAMINA
146896.55,11379.31,0.3,6186.21,6186.21,6186.21
*MATERIAL,NAME=user
*ELASTIC,TYPE=LAMINA
10.0,2.0,3.0,4.0,5.0,6.0
*ORIENTATION,NAME=ORI0,SYSTEM=RECTANGULAR
1.,0.,0.,1.,0.,0.
3.0.
*ORIENTATION,NAME=ORI90,SYSTEM=RECTANGULAR
1.,0.,0.,1.,0.,0.
3.0.
*STEP
*STATIC
** Boundary condition
*INCLUDE,INPUT=c:\users\cpf.bc
*RESTART,WRITE
*EL PRINT,ELSET=CMLF,SUMMARY=NO,POSITION=NODE
S
E
*MONITOR,NODE=401,DOF=1
*END STEP
(b) Final ABAQUS runstream files (xxx.inp) produced from GUI

Through GUI, in group CMLF one layer (0 degree) with material ‘user’ was added and the layer thickness were changed to 0.05; in group UNDA layer 2 was changed to be 0 degree and thickness were changed to 0.075. The material ‘user’ was selected as user defined material (to have degradation analysis). The output point is Element=2 and layer=1, which is in the group CMLF. The following files are the GUI result runstream files, under lines show the difference to the initial file. GUI also produced three internal files, control.dat, user_constant.dat and user_material.dat, which are for CPFail analysis and contain material information inputted from GUI.

(b-1) Static Analysis Runstream CPF-S.inp:

In [Model]/[Control Parameters]/[Analysis], select static_analysis and 10_steps, the following runstream file will be produced from GUI.

```
*HEADING
CPFail-ABAQUS, cpf.ini
*INCLUDE,INPUT=cpf.model
** END OF MESH GENERATION
*SHELL SECTION,ELSET=CMLF,COMPOSITE
  0.05,1,user,ORI0
  0.05,1,user,ORI90
  0.05,1,user,ORI0
*TRANSVERSE SHEAR STIFFNESS
  5.0E4,5.0E4
*MATERIAL, NAME=user
*DEPVAR
  40
*USER MATERIAL, CONST=2
  2, 1
*SHELL SECTION,ELSET=UNDA,COMPOSITE
  0.075,1,orin,ORI0
  0.075,1,orin,ORI0
*TRANSVERSE SHEAR STIFFNESS
  5.0E4,5.0E4
*MATERIAL,NAME=orin
*ELASTIC>Type=LAMINA
  146896.55,11379.31,0.3,6186.21,6186.21,6186.21
*ORIENTATION,NAME=ORI90,DEFINITION=COORDINATES,SYSTEM=RECTANGULAR
  1,0,0,0,1,0
  3,90
*ORIENTATION,NAME=ORI0,DEFINITION=COORDINATES,SYSTEM=RECTANGULAR
  1,0,0,0,1,0
  3,0
*INITIAL CONDITIONS,TYPE=SOLUTION,USER
*USER SUBROUTINE, INPUT=user-t.mt
*STEP,INC=10
*STATIC,DIRECT
  0.1, 1.0
```
Fatigue Analysis Runstream CPF-F.inp:
In [Model]/[Control Parameters]/[Analysis], select Fatigue Analysis and Maximum Log(N) = 6, Load Ratio = 0.1, the following runstream file will be produced from GUI.

*HEADING
CPFfail-ABAQUS, cpf.ini
INCLUDE,INPUT=cpf.model
** END OF MESH GENERATION
*SHELL SECTION,ELSET=CMLF,COMPOSITE
0.05,1,user,ORI0
0.05,1,user,ORI90
0.05,1,user,ORI0
*TRANSVERSE SHEAR STIFFNESS
5.0E4,5.0E4
*MATERIAL,NAME=user
*DEPVAR
40
*USER MATERIAL,CONST=2
2,1
*SHELL SECTION,ELSET=UNDA,COMPOSITE
0.075,1,orin,ORI0
0.075,1,orin,ORI90
*TRANSVERSE SHEAR STIFFNESS
5.0E4,5.0E4
*MATERIAL,NAME=orin
*ELASTIC,TYPE=LAMINA
146896.55,11379.31,0.3,6186.21,6186.21,6186.21
*ORIENTATION,NAME=ORI0,DEFINITION=COORDINATES,SYSTEM=RECTANGULAR
1,0,0,1,0
3,0
*ORIENTATION,NAME=ORI90,DEFINITION=COORDINATES,SYSTEM=RECTANGULAR
1,0,0,1,0
3,90
*INITIAL CONDITIONS,TYPE=SOLUTION,USER
*USER SUBROUTINE,INPUT=user-t.rut
*STEP
STEP 1
*STATIC
1.0,1.0
*INCLUDE, INPUT=cpf.bc
*RESTART, WRITE
*EL PRINT, ELSET=CMLF, SUMMARY=NO, POSITION=NODE
S
E
*MONITOR, NODE=401, DOF=1
*END STEP
*STEP
STEP 2
*STATIC
1.0, 1.0
*END STEP
*STEP
STEP 3
*STATIC
1.0, 1.0
*END STEP
*STEP
STEP 4
*STATIC
1.0, 1.0
*END STEP
*STEP
STEP 5
*STATIC
1.0, 1.0
*END STEP
*STEP
STEP 6
*STATIC
1.0, 1.0
*END STEP
*STEP
STEP 7
*STATIC
1.0, 1.0
*END STEP
*STEP
STEP 8
*STATIC
1.0, 1.0
*END STEP
*STEP
STEP 9
*STATIC
1.0, 1.0
*END STEP
IV-2) Material properties and parameters

Several cases of static and fatigue analyses for different materials are listed here for references.

( $E_f$ - fiber young's modules, $\mu_f$ - fiber Poisson's ratio, $\alpha_f$ - fiber thermal expansion coefficient, $V_f$ - fiber volume fraction, $\sigma_0$ - fiber mean strength, $d_f$ - fiber diameter, $M$ - fiber Weibull modules, $E_m$ - matrix Young’s modules, $\mu_m$ - matrix Poisson’s ratio, $\alpha_m$ - matrix thermal expansion coefficient, $K_{mc}$ - critical matrix stress intensity factor, $T$ - temperature).

(a) Nicalon/CAS
Nicalon fiber:

- $E_f = 195.0 \ (176.0)^*$ GPa
- $\mu_f = 0.2$
- $\alpha_f = 3.0E-6 \ (4.0E-6)^*$ /°C
- $V_f = 0.65$
- $M = 10$
- $\sigma_0 = 388.4$ MPa
- $d_f = 7.00E-5$ m

CAS matrix:

- $E_m = 95.0$ GPa
- $\mu_m = 0.3$
- $\alpha_m = 5.0E-6$ /°C
- $K_{mc} = 2.16(1.73)^*$ MPa-m$^{(1/2)}$
- $T = 0$ °C

(* Properties at $T=400$ °C)

Empirical parameters:

- Friction coefficient = 0.01
- Nonlinear coefficient = 2.4
- Matrix damage limit = 0.3

(b) Graphite/Epoxy
Graphite fiber:

- $E_f = 221.0$ GPa
- $\mu_f = 0.48$
- $\alpha_f = -7.58E-7$ /°C
- $V_f = 0.65$
- $M = 10$
- $\sigma_0 = 388.4$ MPa
- $d_f = 7.00E-6$ m

Epoxy matrix:

- $E_m = 3.45$ GPa
- $\mu_m = 0.2$
\[ \alpha_m = 8.433 \times 10^{-5} \quad /{^\circ}C \]
\[ K_{nc} = 0.67 \quad \text{MPa-m}^{(1/2)} \]
\[ T = 0 \quad ^{0}{C} \]
Empirical parameters:
Friction coefficient = 0.01
Nonlinear coefficient = 2.4
Matrix damage limit = 0.3

(c) Carbon/SiC
Carbon fiber:
\[ E_f = 231.0 \quad \text{GPa} \]
\[ \mu_f = 0.2 \]
\[ \alpha_f = -5.40 \times 10^{-7} \quad /{^\circ}C \]
\[ V_f = 0.40 \]
\[ M = 10 \]
\[ \sigma_0 = 388.4 \quad \text{MPa} \]
\[ d_f = 7.00 \times 10^{-6} \quad \text{m} \]

SiC. matrix:
\[ E_m = 35.10 \quad \text{GPa} \]
\[ \mu_m = 0.22 \]
\[ \alpha_m = 1.10 \times 10^{-5} \quad /{^\circ}C \]
\[ K_{nc} = 0.5 \quad \text{MPa-m}^{(1/2)} \]
\[ T = 0 \quad ^{0}{C} \]
Empirical parameters:
Friction coefficient = 0.3
Nonlinear coefficient = 0.1
Matrix damage limit = 0.1
Fatigue final strain = 0.0115
Fatigue coefficient = 0.075
Walker constant C = 3.13E-15
Walker constant n = 8.4
Heywood parameter p1 = 0.153 \quad \text{MPa}
Heywood parameter p2 = 2206.9 \quad \text{MPa}
Heywood parameter p3 = 0.0031

IV-3) Progressive Damage
An example of tensile laminate plate with a central hole shows the progressive material damage and stress re-distributions, Figure PD-1 ~ PD-4. In order to reduce computation time, PD-5 shows that material degradation analysis performed only in the elements near the hole.

Figures are plotted by using ABAQUS-POST, valuable = s11 for stress, valuable = sv31 for stiffness E11.
Figure PD-1 (1). Stiffness $E_{11}$ distribution for 0 degree layer at step 1

Figure PD-1 (2). Stress $S_{11}$ distribution for 0 degree layer at step 1
Figure PD-2 (1). Stiffness E11 distribution for 0 degree layer at step 4

Figure PD-2 (2). Stress S11 distribution for 0 degree layer at step 4
Figure PD-3 (1). Stiffness E11 distribution for 0 degree layer at step 6

Figure PD-3 (2). Stress S11 distribution for 0 degree layer at step 6
Figure PD-4 (a). Stiffness E11 distribution for 0 degree layer at step 7

Figure PD-4 (b). Stress S11 distribution for 0 degree layer at step 7
Figure PD-5(1). Stiffness E11 for 0 degree layer at step 4 (partly damage analysis)

Figure PD-5(2). Stress S11 for 0 degree layer at step 4 (partly damage analysis)
CONTENTS

I. Preface
   I-1) Introduction
   I-2) System requirement
   I-3) Installation and execution
   I-6) Limitations

II. Getting Started
   II-1) Using GUI to input material parameters

III. GUI Help
   III-1) [File]
   III-2) [View]
   III-3) [Command]/[Execute] or [Execute]
   III-4) [Model]/[Laminate] or [Laminate]
   III-5) [Model]/[Load] or [Load]
   III-6) [Model]/[Analysis] or [Analysis]
   III-7) [Database]/ or [Fiber], [Matrix] and [Ply]
   III-8) [Options]

IV. Material Parameters Information
   IV-1) Parameter definition
   IV-2) How to adjust parameters

V. Example of Material Properties and Parameters
   V-1) Nicalon/CAS
   V-2) Graphite/Epoxy
   V-3) Carbon/SiC
I. Preface

I-1) Introduction

CPFail (Composite Progressive Failure Analysis) is an analysis code for computing the composite material degradation due to tensile damage. The material degradation is estimated by using a combined micromechanics, fracture mechanics and statistics failure approach. With a step loading procedure, it can show the structural progressive failure or predict the structure fatigue life. A user friendly GUI (Graphic User Interface) is designed for helping user to input material information and loading condition.

I-2) System requirement

The CPFail is developed in C/C++ using X-Windows and Motif toolkits. It can be run only on UNIX system.
I-3) Installation and execution

Installation:
The software is distributed in tar format file asm.tar. The file contains all the required data files, source files and java class files required to execute the programs. To install the software components copy change directory to the destination directory and extract the files using the command: tar -xvf asm.tar. This command creates the ASM main directory and sub directories to copy the CPFail, MicroTex and ABAQUS interface programs.

Execution:
The executable command for CPFail is:

```
go to directory /ASM/CPF/CPFAIL and execute ./cpfail command
```

Original code Files:
The following files are the original code files, user should carry over when change software installation directory, user can also carry over xxx.par files for previous cases:

cpfail
cma
fiber.dbf
matrix.dbf
ply.dbf
cpfail.tif

I-4) Limitations
Composite material degradation is caused by tensile damage with the effect of shear stress.
II. Getting Started

a) Open CPFail GUI window

b) Create material database:
   In main screen or go to [Database], select [Fiber] and [Matrix] if user wants to input material properties from fiber and matrix properties; select [Ply] if user wants to input ply properties. (The material degradation analysis will be based on fiber and matrix properties; the program will convert ply properties to fiber and matrix properties.)

c) Ply, Fiber or Matrix Database:
   * Define a new name or select one from database. (The material name will be used in [Laminate]).
     Ply (Fiber, Matrix) Name: To select an existing material
     New Ply (Fiber, Matrix) Name: To create a new material or modify the existing material name
   * Input or modify material properties. Refer to III and IV for detailed material information.
   * input a description for user's own remark

d) Go to [Laminate], input layer thickness, orientation, fiber volume fraction and friction coefficient, select material name, than click [Add] to build a laminate layer by layer.

e) Go to [Load]
   * Select load type: Stress or Displacement
   * Input load values: start initial values and increment values
   * Input analysis environment temperature and composite manufacture processing temperature

f) Go to [Analysis], select Static Analysis or Fatigue Analysis.
   f-1. Static Analysis:
   * Input maximum steps
   * Input static analysis parameters: see IV for detailed information
   * Input final strain (in 0 degree direction) to stop analysis
   f-2. Fatigue Analysis:
   * Input log maximum cycles, Log(N)
   * Input maximum load value (peak load)
   * Input loading ratio, load ratio = Minimum load / Maximum load
   * Input fatigue analysis parameters: see IV for detailed information
   * Input fatigue final strain (in 0 degree direction)

 g) Go to [File]/[Save As], to save the case as a xxx.par GUI file

h) Go to [Execute], to start analysis
i) Go to [View], to read result figures and text files
See III. GUI Help for detailed information.

j) Structural fatigue S-N curve
   It should be pointed out that in the fatigue analysis, the loading cycle increases as,
   \( \log(N) = 1, 2, 3 \ldots \), to the maximum \( \log(N) \). In each fatigue analysis, load will be a
   constant and the final fatigue cycle, \( \log(N) \), will be computed. Thus for one load level, it
   can only produce one point on the curve (S-N) of Maximum stress verses Fatigue cycle
   \( \log(N) \). In order to have a complete S-N curve, several load levels should be executed.
   The fatigue failure (final) cycle, \( \log(N) \) is printed in the result file, s-s.dat, and can be
   read from [View]/[Text File] window.
III. GUI Help

III-1) [File]
(a) [New]: Open a new case, after input, save the case in a new GUI file, xxx.par
(b) [Open]: Open a case from a previously saved xxx.par GUI file
(c) [Save]: Save the current modified case to the original xxx.par file
(d) [Save As]: Save the current case to a new GUI file, for example, xxx-1.par
(e) [Print]: Print out results or the lamination figure in the window
(f) [Exit]: Close GUI

III-2) [View]
(a) [Model]: Lamination figure
(b) [Ply Damage]: Steps vs. remaining ply fiber volume fraction
(c) [Lam Prop Change]: Steps vs. laminate stiffness ratio, $E_{11}/E_0^{11}$ and $E_{22}/E_0^{22}$, $E_0$ s are the undamaged original value
(d) [Lam Stresses]: Steps vs. laminate stresses
(e) [Lam Displacements]: Steps vs. laminate displacements
(f) [Lam Stress-Strain]: Laminate strain vs. stress
(g) [Text File]: View text files

III-3) [Command]/[Execute] or [Execute]
executing analysis

III-4) [Model]/[Laminate] or [Laminate]
* Input or modify ply numbering, ply thickness, orientation, fiber volume fraction
* Adjust friction coefficient according to testing results
* Select material input format: ply property or fib/matrix properties. Select material name from [Database], [Fiber], [Matrix] or [Ply]

III-5) [Model]/[Load] or [Load]
(a) Load Type: Stress or displacement load. For fatigue analysis, apply stress load only.
(b) Initial Value: Starting load. For fatigue analysis, it is the maximum (peak) load.
(c) Increment Value: Constant increment load. For fatigue analysis, it is zero.
(d) Temp. Difference: Environment temperature
(e) Process Temperature: Composite manufacture processing temperature

III-6) [Model]/[Analysis] or [Analysis]
(a) Static Analysis:
* Steps --- Total load increments
* Matrix Damage Limit --- (0.1 ~ 0.3) see IV for details
* Nonlinear Coefficient --- (0.01 ~ 10) see IV for details
* Static Final Strain --- Strain value to stop the static analysis

(b) Fatigue Analysis:
* Maximum Cycles Log(N) --- User defined stopping point
* Maximum Load --- Load peak value
* Load Ratio --- (= Minimum load / Maximum load) in a load cycle
* Fatigue Final Strain --- Strain value to stop the fatigue analysis
* Fatigue Parameter --- (0.01 ~ 3.0), see IV for information
* Walker Constants (C and n) --- (3.13E-15, 8.4), see IV for information
* Heywood Parameters (p1 ~ p3) --- (0.153 MPa, 2206.9 MPa and 0.0031), see IV for information

III-7) [Database]/or [Fiber], [Matrix] and [Ply]
(a) User can build new or modify old matrix, fiber and ply property databases.
   * Ply (Fiber, Matrix) Name: To select an existing material
   * New Ply (Fiber, Matrix) Name: To create a new material or modify the existing material name
(b) Matrix Critical Stress Intensity Factor, Kmc, (0.1 ~ 5.0)
(c) Weibull Modules (M) and Mean Fiber Strength ($\sigma_0$)
   Empirical parameters from modified Weibull fiber damage statistic equation, $M=10$ and $\sigma_0=388.0$ MPa are default values. See IV for details
(d) Friction Coefficient
   Empirical parameter of matrix-fiber integration (0.01 ~ 1.0)

III-8) [Options]:
(a) Units:
   * Let user to choose SI or US units.
   * All values will be changed according to unit change in GUI and will be written in to ABAQUS runstream file.
(b) Graphic Settings:
   * Adjust output figure settings
IV. Material Parameters Information

Material special parameters are designed for user to adjust material or laminate behavior due to damage, fatigue, delamination, material nonlinearity or plasticity.

IV-1) Parameter definition:

* **Weibull Modules** (M) and **Mean Fiber Strength** ($\sigma_0$)

Empirical parameters from modified Weibull fiber damage statistic equation, M=10 and $\sigma_0$=388.0 MPa are default values.

$$P_f = 1 - \exp\left[ -\frac{4\pi a t}{k_0}\left(\frac{T_i}{\sigma_0}\right)^M \right]$$

* **Friction Coefficient**: Empirical parameter of matrix-fiber integration (0.01 ~ 1.0)
* **Matrix Damage Limit**: Matrix modules remaining ratio after matrix cracking
* **Nonlinear Coefficient**: Adjust for material nonlinear behavior due to yielding, plasticity, damage or delamination. The nonlinear effect is bigger for lager value, for example, 0.01 (linear) ~ 2.4 (nonlinear).
* **Process Temperature**: Composite laminate manufacture curling temperature.
* **Walker Constant C and n**: from Walker's fatigue statistic equation for matrix fatigue cracking. Suggested value: $C=3.13\times10^{-15}$ and $n=8.4$

$$\frac{da}{dN} = CK_m^n$$

* **Fatigue Coefficient**: Empirical fatigue parameter. A smaller value corresponds to smaller damage and longer life. A positive value of (0.01 ~ 3.0).
* **Fatigue Final Strain**: The fatigue failure criteria to define fatigue life cycles. User assume that the structure failed as the strain ($\epsilon_{11}$) at the monitor point reaches to this value.
* **Heywood Parameters $p_1$ – $p_3$**: from Heywood's fatigue statistic equation for fiber strength fatigue reduction. The effects of $p_1$ ~ $p_3$ are shown in the following figure. Suggest value: $p_1 = 0.153$ MPa, $p_2 = 2206.9$ MPa, $p_3 = 0.0031$

Heywood equation:

$$\sigma_u = \pm \sigma_i \left[ 1 - \frac{\sigma_m}{\sigma_i} \right] \left[ A_0 + \gamma(1 - A_0) \right]$$

$$A_0 = \left[ 1 + \frac{p_3 \bar{n}^4}{1 + \sigma_i/p_1} \right] \frac{1}{1 + p_3 \bar{n}^4},$$

$$\gamma = \frac{\sigma_m}{\sigma_i} \left[ 1 + \left( \frac{\sigma_i \bar{n}}{p_2} \right)^4 \right],$$

$$\bar{n} = \log(N)$$
IV-2) How to adjust parameters

In order to make this material degradation analysis tool more general to composite materials, the empirical parameters need to be adjusted according to different materials, structure or different damage conditions. Default values provide a base for the parameter adjustments, so they should be kept for references. The following two figures show the effects of parameters on material or structural behaviors.

(a) Static analysis:

* $K_{mc}$ --- Matrix critical stress intensity factor. A larger value will move up 'yielding point'
* $\beta$ --- Nonlinear coefficient (0.01 ~ 10). A larger value will have stronger nonlinear behavior.
* $\sigma_0$ --- Mean fiber strength. A larger value will move up the final failure stress
* $\mu$ --- Friction coefficient (0.01 ~ 1.0).
(b) Fatigue analysis:

In addition to the parameters in static analysis, in fatigue analysis Walker fatigue parameters for matrix fatigue and Heywood fatigue parameters (p1 – p3) for fiber fatigue may need to be adjusted. The Heywood parameters have the effects on S-N curve as shown in the following figure.
V. Examples of Material Properties and Parameters

Several cases of static and fatigue analyses for different materials are listed here for references.

\( E_f \) - fiber young’s modules, \( \mu_f \) - fiber Poisson’s ratio, \( \alpha_f \) - fiber thermal expansion coefficient, \( V_f \) - fiber volume fraction, \( \sigma_0 \) - fiber mean strength, \( d_f \) - fiber diameter, \( M \) - fiber Weibull modules, \( E_m \) - matrix Young’s modules, \( \mu_m \) - matrix Poisson’s ratio, \( \alpha_m \) - matrix thermal expansion coefficient, \( K_{mc} \) - critical matrix stress intensity factor, \( T \) - temperature).

V-1) Nicalon/CAS

Nicalon fiber:

\[
\begin{align*}
E_f &= 195.0 \ (176.0)^* \text{ GPa} \\
\mu_f &= 0.2 \\
\alpha_f &= 3.0E-6 \ (4.0E-6)^* \ /°C \\
V_f &= 0.65 \\
M &= 10 \\
\sigma_0 &= 388.4 \text{ MPa} \\
d_f &= 7.00E-5 \text{ m}
\end{align*}
\]

CAS matrix:

\[
\begin{align*}
E_m &= 95.0 \text{ GPa} \\
\mu_m &= 0.3 \\
\alpha_m &= 5.0E-6 \ /°C \\
K_{mc} &= 2.16(1.73)^* \text{ MPa} \cdot m^{(1/2)} \\
T &= 0 \ /°C
\end{align*}
\]

(* Properties at \( T=400 \ /°C \))

Empirical parameters:

Friction coefficient = 0.01
Nonlinear coefficient = 2.4
Matrix damage limit = 0.3

V-2) Graphite/Epoxy

Graphite fiber:

\[
\begin{align*}
E_f &= 221.0 \text{ GPa} \\
\mu_f &= 0.48 \\
\alpha_f &= -7.58E-7 \ /°C \\
V_f &= 0.65 \\
M &= 10 \\
\sigma_0 &= 388.4 \text{ MPa}
\end{align*}
\]
\( d_f = 7.00 \times 10^{-6} \text{ m} \)

**Epoxy matrix:**
- \( E_m = 3.45 \text{ GPa} \)
- \( \mu_m = 0.2 \)
- \( \alpha_m = 8.433 \times 10^{-5} \text{ /°C} \)
- \( K_{mc} = 0.67 \text{ MPa-m}^{(1/2)} \)
- \( T = 0 \text{ °C} \)

**Empirical parameters:**
- Friction coefficient = 0.01
- Nonlinear coefficient = 2.4
- Matrix damage limit = 0.3

**V-3) Carbon/SiC**

**Carbon fiber:**
- \( E_f = 231.0 \text{ GPa} \)
- \( \mu_f = 0.2 \)
- \( \alpha_f = -5.40 \times 10^{-7} \text{ /°C} \)
- \( V_f = 0.40 \)
- \( M = 10 \)
- \( \sigma_0 = 388.4 \text{ MPa} \)
- \( d_f = 7.00 \times 10^{-6} \text{ m} \)

**SiC. matrix:**
- \( E_m = 35.10 \text{ GPa} \)
- \( \mu_m = 0.22 \)
- \( \alpha_m = 1.10 \times 10^{-5} \text{ /°C} \)
- \( K_{mc} = 0.5 \text{ MPa-m}^{(1/2)} \)
- \( T = 0 \text{ °C} \)

**Empirical parameters:**
- Friction coefficient = 0.3
- Nonlinear coefficient = 0.1
- Matrix damage limit = 0.1

- Fatigue final strain = 0.0115
- Fatigue coefficient = 0.075
- Walker constant C = 3.13 \times 10^{-15}
- Walker constant n = 8.4
- Heywood parameter p1 = 0.153 \text{ MPa}
- Heywood parameter p2 = 2206.9 \text{ MPa}
- Heywood parameter p3 = 0.0031
Textile composites are being considered for potential structural application in the aerospace and automotive industries because of the better impact and delamination resistance over tape laminates. However, their architectures are also more complex due to the weaving, braiding, and knitting of fiber yarns. Thus, along with fabrication processes and test methodologies, the development of analytical models to predict the mechanical properties and strength of the textile composites is of increasing importance.

A general purpose micro-mechanics analytical technique was developed by Naik [1,2] in the *Textile Composite Analysis for Design* (TEXCAD) code to predict the overall mechanical and thermal properties of textile composites. The calculation of material properties in this code is based on a simple geometry analysis which transforms the repeating unit cell (RUC) of textile composites into a typical micro-structure with uniform strains; failure criterion of textile composites is based on maximum stress or maximum strain criterion of the matrix and yarn materials.

The failure mechanism of the yarn slice in a textile composite is quite similar to that of a laminated composite, which involves fiber/matrix debonding, fiber-bridged matrix cracking, and statistical fiber failure. An analytical model based on micro-mechanics, fracture mechanics, and statistical concept was developed in the Phase I study [3] to predict the failure of laminate composite under high temperature and tension loads. In the analysis, the fiber-bridged matrix crack was idealized by a continuum model in which the effect of the bridging fibers was modeled by an equivalent closure pressure on the crack surface. A rigorous shear-lag model was used to evaluate the closure pressure distribution along the crack surface and to compute the shear stress transfer along the debonded, frictional sliding fiber/matrix interface. A closed-form micro-mechanics analysis was used to compute the thermal and mechanical (axial and transverse) stress along the fiber/matrix interface region.

In this report, an engineering approach (MicroTex) is developed for textile composites to predict the initial thermal and mechanical properties based on simple geometry, to predict damage progress, and to predict the failure of textile composites. The basic procedure is shown in Figure B1-1. The statistical failure criterion based on fracture mechanics, shear-lag theory, and fiber pull-out concept is modified and combined with other maximum stress or maximum strain criteria to provide a complete set of standards for yarn and matrix slices failure prediction. Nonlinear material properties for both matrix and fiber are included in addition to the failure progress of material slices to calculate the three dimensional stiffness reduction of the RUC. The failure of textile composites or its RUC as a structure with yarn and matrix slices as its components is predicted based on the overall stiffness calculation of RUC. Comparisons with various experimental results for various different textile composites are also presented.

Fatigue analysis for the textile composite is presented in this report also. The assumption is that the stiffness reduction due to fatigue loading when the composite is under uniaxial loading contains information necessary for fatigue analysis for this composite under complex stress states. A fatigue model is developed based on this assumption, empirical parameters are determined using testing data for (0/90) C/SiC composite and then applied to for a (0/-60/60) C/SiC composite.
B1-2. Geometry Model of Textile Composites

Real textile composites have very complicated geometry structures. Figure B1-2 shown a two dimensional C/SiC plain weave composite laminate which is one of the simplest textile composites. It consists of stacked, pre-impregnated layers of woven fabric which are cured and consolidated by a process similar to tape laminates. Each yarn is a bundle of filaments (or fibers) and the yarn size is measured by the number of filaments in the yarn. To model this geometry is impossible without certain simplifications.

A general geometry model was developed [1-2] for textile composites based on several assumptions to describe the fabric architectures and then calculate the mechanical and thermal properties of the textile composites. By observing the periodicity of the repeating pattern in a woven (or braided) fabric, it is assumed that a small repeating unit cell (RUC) can be isolated which is sufficient to describe the fabric architecture. Each individual yarn architecture is discretely modeled using sinusoidal undulations at yarn crossovers and a straight portion. And finally, the iso-strain assumption was used to calculate the overall thermal and mechanical properties and average strains over RUC.

This section will give a brief derivation of geometry model for several textile composites, and then the calculation for overall thermal and mechanical properties of RUCs.

Laminated composite

The RUC for this classical lamination is a two dimensional orientated yarn as shown in Figure B1-3. The ply longitudinal and transverse directions are indicated by 1 and 2, respectively. In the model of laminated composite there is only one yarn slice, and there is no matrix slice.

2D weaves and 2D braids

The geometry model of 2D weaves and 2D braids was developed for plain weave, 5-harness satin weave, 8-harness satin weave, plain 2D braid, 5-harness satin 2D braid, and 8-harness satin 2D braid.

The RUC of a 5-harness satin weave and plain 2D braids are shown in Figures B1-4 and B1-5, respectively. The sectional view A-A shows the undulation of a yarn. The known quantities are assumed to be braid angle, $\theta_b$ (for weave, 45 degree), yarn spacing, $d_o$, yarn filament counts $n$, yarn packing density, $p_d$, filament diameter, $d_f$, and the thickness of each layer, $H$. Those known quantities were used to calculate the unknown quantities such as yarn thickness, yarn cross-sectional areas, yarn crimp angle, overall fiber volume fraction, and yarn undulating paths which are required to discretely model each yarn within the RUC. Here we assume that the fill and warp yarns are identical.

The yarn cross-sectional area, $A$, was assumed to remain constant along the entire yarn path. With the yarn filament counts, the yarn packing density, and filament diameter, the cross-sectional area could be calculated as

$$A = \frac{\pi d_f^2 n}{4 p_d}$$

Assuming a sinusoidal path, the yarn centerline path can be expressed as
\[ Z_c = \pm \frac{H}{4} \sin \left[ \frac{L_c \pi \sin(\theta_b)}{L_u} \right] \]  

(B-1-2)

where \( L_c \) is measured from the corresponding cross-over point, \( L_u \) is the projection of the undulating part of the yarn path. Since the cross-sectional areas of yarns can be written as

\[ A = \frac{d_o H}{2} - L_u H \left( 1 - \frac{2}{\pi} \right) \sin \theta_b \cos \theta_b \]  

(B1-3)

we have the projection of the undulating part of the yarn paths from equation (B1-1) and (B1-3) as

\[ L_u = \frac{d_o H - 2A}{(1 - 2/\pi)H \sin(2\theta_b)} \]  

(B1-4)

Then the length of each yarn can be obtained as

\[ L = \frac{Kd_o}{\sin(2\theta_b)} + 2 \left( \frac{L_u}{2} - \int Z(x) \, dx \right) \]  

(B1-5)

where \( K = 2 \) for plain weave or plain 2D braid, \( 5 \) for 5-harness satin weave and 5-harness satin 2D braid, and \( 8 \) for 8-harness satin weave and 8-harness satin 2D braid. The overall fiber volume fraction becomes

\[ V_f = \frac{2p_d LA}{Hd_o K} \]  

(B1-6)

and the crimp angle as

\[ \tan(\theta_c) = \frac{H \pi}{2L_u} \sin(2\theta_b) \]  

(B1-7)

**1x1, 2D triaxial braids**

The RUC of a 2D 1x1 triaxial braids is shown in Figure B1-6. The sectional view A-A shows the undulation of a braider yarn which undulates over and under the axial yarns in the RUC. The known quantities are assumed to be braid angle, \( \theta_b \), axial yarn spacing, \( d_o \), yarn filament counts for the axial and braider yarns, \( n_a \) and \( n_b \), respectively, yarn packing density, \( p_d \), filament diameter, \( d_f \), and the thickness of each layer, \( H \). Those known quantities were used to calculate the unknown quantities such as yarn thickness, yarn cross-sectional areas, yarn crimp angle, overall fiber volume fraction, and yarn undulating paths which are required to discretely model each yarn within the RUC.

The cross-sectional areas, \( A_a \) and \( A_b \), for the axial and braider yarns, respectively, are assumed to remain constant along the entire yarn path. With the yarn filament counts, the yarn packing density, and filament diameter, the cross-sectional areas can be calculated as

\[ A_a = \frac{\pi d_f^2 n_a}{4p_d a}, A_b = \frac{\pi d_f^2 n_b}{4p_d b} \]  

(B1-8)

The projected lengths, \( L_a \) and \( L_b \), of the axial and braider yarn paths are functions of the axial yarn spacing and braid angle, and can be given by
\[ L_a = \frac{2d_o \cos(\theta_b)}{\sin(\theta_b)}, \quad L_b = \frac{d_o}{\sin(\theta_b)} \]  \hspace{1cm} (B1-9)

respectively. Then the total volume occupied by the axial and braider yarns was given by \((A_a L_a + 4 A_b L_b)\). Since the dimensions of the parallelepiped RUC are \(L_b \times L_b \times H\), where \(H\) is the layer thickness, the overall fiber volume fraction can be calculated as

\[ V_f = \frac{p_{da} A_a \cos(\theta_b) + p_{db} A_b \sin(\theta_b)}{H d_o \cos(\theta_b)} \]  \hspace{1cm} (B1-10)

if the ratio of the braider yarn thickness, \(t_b\), to axial yarn thickness, \(t_a\), is assumed to be \(m\), we have

\[ t_b = \frac{H}{2 + m} \]  \hspace{1cm} (B1-11)

This unknown thickness ratio is determined using cross-sectional areas. The braider yarn width, \(w_b\) (perpendicular to the yarn direction), is related to the axial yarn spacing by assuming the connection between axial yarns as \(w_b = d_o \cos(\theta_b)\). The braider yarn centerline path, on the other hand, is described using sine function as

\[ Z_c = \pm \frac{t_a + t_b}{2} \sin \left[ \frac{L_c \pi \sin(\theta_b)}{L_u} \right] \]  \hspace{1cm} (B1-12)

where \(L_c\) was measured from the corresponding cross-over point. The parameter \(L_u\), that is the projection of the undulating part of the yarn path, is related to the cross-sectional areas of axial and braider yarns as

\[ A_a = w_a t_a - L_u \left( t_a + t_b \right) \left[ 1 - \frac{2}{\pi} \left( s + \cos(s) \right) \right] \]  \hspace{1cm} (B1-13)

\[ A_b = w_b t_b - \left( \frac{2}{\pi} - 1 \right) \frac{L_u t_b \cos(\theta_b)}{1 + m} \]  \hspace{1cm} (B1-14)

\[ s = \sin^{-1} \left( \frac{1}{1 + m} \right), \quad w_a = \frac{d_o}{2} - 2s \frac{L_u}{\pi} \]

obviously, equations (B1-13) and (B1-14) has only two unknowns: \(m\) and \(L_u\), an iteration scheme is employed to solve these equations, and then the total length of the straight portion of each braider yarn becomes

\[ L_{st} = \frac{d_o}{\sin(\theta_b)} - L_u \left[ 1 + \frac{2 \cos(\theta_b)}{1 + m} \right] \]  \hspace{1cm} (B1-15)

and yarn crimp angle \(\theta_c\):

\[ \tan(\theta_c) = \frac{(t_a + t_b) \pi}{2 L_u} \sin(\theta_b) \]  \hspace{1cm} (B1-16)
2x2, 2D triaxial braids

For 2D, 2x2 triaxial braids (see Figure B1-7), the known quantities are assumed to be braid angle, $\theta_b$, axial yarn spacing, $d_o$, yarn filament counts for the axial and braid yarns, $n_a$ and $n_b$, respectively, yarn parking density for axial and braid yarns, $p_{da}$ and $p_{db}$, respectively, filament diameter, $d_f$, the thickness of each layer, $H$, and the axial yarn content, $y_{av}$. Those known quantities are used to calculate the unknown quantities such as yarn thickness, yarn cross-sectional areas, yarn crimp angle, overall fiber volume fraction, and yarn undulating paths which are required to discretely model each yarn within the RUC.

The length of total axial yarn and each braid yarn can be expressed as

$$L_a = 8d_o \cot(\theta_b) \quad \text{and} \quad L_b = \frac{A_a L_a (1 - y_{av})}{8 A_b y_{av}} \quad \text{(B1-17)}$$

where $A_a$ and $A_b$ are the cross-sectional areas of axial and braid yarns

$$A_a = \frac{\pi d_f^2 n_a}{4 p_{da}}, \quad A_b = \frac{\pi d_f^2 n_b}{4 p_{db}} \quad \text{(B1-18)}$$

With a layer thickness $H$, the overall fiber volume fraction becomes:

$$V_f = \frac{L_a \left[p_{da} y_{pa} + (y_{pa} - 1) p_{db}\right]}{H d_o y_{pa}} \quad \text{(B1-19)}$$

Following a similar procedure as 2D, 1x1 triaxial braids, we can form the equation for the yarn thickness ratio $m = t_a / t_b$ as

$$\frac{A_a - d_a H m/(2 + m)}{\left[(1 + m)(\cos s - \pi/2) + s\right]} = \frac{A_b - d_a H \cos \theta_b / (2 + m)}{\left[(1 + m)(1 - \cos s) - s\right]} \quad \text{(B1-20)}$$

where $s = \sin^{-1}\left(\frac{1}{1 + m}\right)$.

We then can calculate the crimp angle as

$$\tan(\theta_c) = \frac{t_b^2 (1 + m)[(1 + m)(\cos s - \pi/2) + s]}{A_a - md_o t_b} \quad \text{(B1-21)}$$

and the projection of the undulating part of the braider yarn path as

$$L_u = \frac{(t_a + t_b)\pi}{2 \tan(\theta_c)} \quad \text{(B1-22)}$$

3D spatially oriented composite

The geometry model for 3D spatially oriented composite was developed to describe composites which is made up of straight yarns oriented along different directions. The RUC of a example composite is shown in Figure B1-8. In the RUC of this composite each yarn becomes a yarn slice, and as always, all the interstitial matrix form a matrix slice.
Thermal and mechanical properties of RUC

All the analysis in this report is assumed to apply to the RUC, so that the analyzed specimen of textile composite is so large that all the thermal and mechanical properties computed with its RUC can be considered as the same of the whole specimen. The overall composite properties are determined by discretizing all the yarns within the RUC. The straight portions of each yarn path are modeled as a single yarn slice. Along an undulating portion, the yarn is divided into piecewise straight slices made perpendicular to its in-plane direction. Thus, the sinusoidal yarn undulating portion is approximated by interconnected straight yarn slices.

The RUC of a textile composite was treated as a system consists of a set of spatially oriented yarn slices and an interstitial matrix material slice. The matrix is assumed to be isotropic, while the yarn slices transversely isotropic. For the mth material slice (yarn or matrix), the three dimensional stiffness matrix \([C']_m\) can always be calculated from given material constants (Young’s modulus and Poisson’s ratios). Assuming the strains are uniform inside RUC, the stress-strain relation becomes

\[
\{\sigma\} = [C]\{\epsilon\}
\]

where the 6x6 matrix \([C]\) is the overall stiffness matrix of RUC, which can be expressed as a summation over all the N yarn and matrix slices in terms of the materials slices stiffness matrix:

\[
[C] = \sum_{m=1}^{N} \left(V_m[T]_m^T[C']_m[T]_m\right)
\]

where \(V_m\) is the volume fraction of the mth material slice, matrix \([T]_m\) is the transformation between the global coordinates and the local coordinates of the mth material slice, and superscript \(T\) indicates transpose. With a given loading, the average strains in a RUC, that is also the strains of each material slice, can be calculated from equation (B1-23). The stress components in global coordinates of each material slice can be obtained and transformed to the local material coordinates as

\[
\{\sigma\}_m = \left([T]_m[C']_m[T]_m^T\right)\{\epsilon\}
\]

Thermal properties are also calculated based on the iso-strain assumption. For the mth material slice, thermal residual stresses are given by

\[
\{\sigma_T\}_m = \Delta T[C']_m\left([T]_m^T\{\alpha\}_m - \{\alpha'\}_m\right)
\]

where \(\Delta T\) is the change in temperature from the stress free state, \(\{\alpha'\}_m\) is the coefficients of thermal expansion (CTE) of the mth material slice and \(\{\alpha\}\) is assumed to be the overall CTE for the RUC, given as

\[
\{\alpha\} = [C]^{-1} \sum_{m=1}^{N} V_m[T]_m^T[C']_m\{\alpha'\}_m
\]

This formulation enables the calculation of stresses and strains of each material slice when the thermal or mechanical loads are given. This forms the foundation for the damage analysis and failure analysis of textile composites.
B1-3. Statistical Failure Criterion of Yarns

The geometry model of textile composites assumes that the RUC of a textile composite is a system consists of matrix and yarn slices. The geometry model has already considered the orientation and geometry shape of each material slice. For the matrix slice, since the matrix material is assumed to be isotropic, only the volume of one matrix slice is necessary for the geometry modeling, while for a yarn slice, other information like orientation and fiber volume fraction is also important. As we introduced above, the overall stiffness and coefficients of thermal expansion of the RUC, also of the textile composite, are calculated as a summation of all the material slices, the damage properties of the RUC, that are the stiffness reduction and strength of the RUC, are also calculated based on each individual material slice. In what follows we introduce a failure criterion of yarn slices under tension, based on micro-mechanics analysis and a statistical model.

A yarn slice is the same as a laminated composite. It is consist of fibers and matrix. The failure criterion proposed here is based on the fiber pull-out concept of laminated composites. As shown in Figure B1-9, assuming the matrix has a crack and the fiber behaves as a bridge that has a traction $T$, the crack causes the sliding between the matrix and fiber over a length of $l$, and allows the fiber to open out of the matrix a distance $u$, that is also half of the crack opening. We assume that the slipping on the fiber-matrix interface is local, that is beyond the distance $l$, the matrix and fiber are continue. Within the slipping distance, the stress redistributed between the matrix and fiber through a shear stress $\tau$ on the fiber-matrix interface is called shear stress parameter.

Shear-lag Model

A shear-lag model developed by Marshall and Cox [5] was used in the present analysis to establish the relationship between the fiber traction and the crack opening displacements. This relation is then provided to the fracture mechanics analysis to determine the fiber traction. As shown in Figure B1-9, assuming purely frictional fiber-matrix bond, the slipping distances can be determined by the length over which the interface shear stresses exceed the maximum shear stress the fiber-matrix interface can resist. The stress $\sigma_m$ for matrix and $\sigma_f$ for fiber under far-field stress $\sigma_c$ can be related to each other by neglecting the effect of shear stress beyond the slipping distance, as

$$\frac{\sigma_m(l)}{E_m} = \frac{\sigma_f(l)}{E_f} \quad \text{(B1-28)}$$

where $E_m$ and $E_f$ are longitudinal Young’s modulus of matrix and fiber, respectively.

Assume the extension due to stress $\sigma_c$ is $\delta$ for matrix, then the extension for fiber will be $u + \delta$, considering the equilibrium of the matrix and fiber separately, we have a set of equations as:

$$\sigma_m(z)A_m = 2\pi R z \tau$$

$$TA_f = 2\pi R z \tau + \sigma_f(z)A_f \quad \text{(B1-29)}$$

$$\delta = \int_0^l \frac{\sigma_m(z)}{A_m E_m} dz, \quad (\delta + u) = \int_0^l \frac{\sigma_f(z)}{A_f E_f} dz \quad \text{(B1-30)}$$
where $R$ is the radius of the fiber, $A_f$ is the fiber cross-section area, $A_m$ is the area of matrix per fiber, or $A_f/(A_f + A_m) = V_f$ is the fiber volume fraction. Notice that in the slipping area, the shear stress on the fiber-matrix interface reaches its maximum value, the constant shear stress parameter, then equation (B1-31) becomes

$$\delta = \frac{\pi R l^2 \tau}{A_m E_m}, \quad (\delta + u) = \frac{T l}{E_f} - \frac{l^2 \tau}{R E_f} \tag{B1-32}$$

This set of equations then gave the relation between the crack opening displacement and fiber traction as

$$u = \frac{R E_m (1 - V_f)}{4 E_f E_c \tau} \tau^2 \tag{B1-33}$$

also we have the slipping length when the matrix crack is fully developed can be expressed as

$$\frac{2 \tau}{a} = \frac{\sigma_c E_m (1 - V_f)}{E_c V_f} \tag{B1-34}$$

and the fiber stress within the slipping region as

$$\sigma_f(z) = \frac{\sigma_c}{V_f} - \frac{2 \tau z}{a} \tag{B1-35}$$

### Fracture Mechanics Analysis

The fracture mechanics analysis is required to provide the relationship between the unknown crack opening displacement $u(x)$ and fiber bridge traction $T(x)$. As shown in Figure B1-9, considering a crack under far-field applied stress, $\sigma_c$, and a closure pressure, $p(x)$, the crack opening displacement $u(x)$ can be written [5] as:

$$u(x) = \frac{2 c \sigma_c}{\pi E_c} \int_0^{\infty} \left[ 1 - \frac{p(t)}{\sigma_c} \right] \ln \frac{\sqrt{1 - t^2} + \sqrt{1 - x^2}}{\sqrt{1 + t^2} - \sqrt{1 + x^2}} dt \tag{B1-36}$$

where the $E_c$ is the longitudinal Young’s Modulus of the composite (or yarn), $c$ is the half crack length. The bridging pressure, $p(x)$, and the bridging traction $T(x)$ is related by the fiber volume fraction, $V_f$, as:

$$p(x) = V_f T(x) \tag{B1-37}$$

Hence, the bridging traction $T(x)$ can be solved by using an iterative procedure from the following integral equation [3]:

$$\frac{R E_m (1 - V_f)}{4 E_f E_c \tau} \tau^2(x) = \frac{2 c \sigma_c}{\pi E_c} \left[ \frac{\sqrt{1 - x^2} - V_f}{\sigma_c} \int_0^{\infty} \left[ T(t) \ln \frac{\sqrt{1 - t^2} + \sqrt{1 - x^2}}{\sqrt{1 + t^2} - \sqrt{1 + x^2}} \right] dt \right] \tag{B1-38}$$

After the solving of bridging pressure, the stress intensity factor for yarn material, $K$, at the fiber-bridged crack can be calculated as:
The matrix stress intensity factor, $K_m$, can be related to $K$ as:

$$K_m = K \left(\frac{E_m}{(1-V_f)E_c}\right)$$  \hspace{1cm} (B1-40)

where $E_m$ is the matrix Young's modulus. With a given critical stress intensity factor of matrix, $K_{lc}$, crack propagation is predicted when

$$K_m \geq K_{lc}$$  \hspace{1cm} (B1-41)

### Statistical Failure Criterion

The Weibull distribution was used to describe the probability of fiber failure. The probability was defined as [6]

$$P_f = 1 - \exp \left[-2\pi a \int_{-\infty}^{w} \left(\frac{\sigma}{\sigma_o}\right)^m dz\right]$$  \hspace{1cm} (B1-42)

where $M$ is the Weibull modulus and $\sigma_o$ is the mean fiber strength, both are material constants. Substituting equations (B1-34) and (B1-35) in (B1-42), we have

$$P_f = 1 - \exp \left[\frac{\pi a^2 \sigma_o^{m+1}}{(m+1)\sigma_o^{m}V_f^{m+1}} \left(\frac{E_m(1-V_f)}{E_c}\right)^m \left(1 + \frac{E_m(1-V_f)}{E_c}\right)^{m+1}\right]$$  \hspace{1cm} (B1-43)

With a given critical value $P_{fc}$, the fiber failure criterion, or the yarn failure criterion, was set to be

$$P_f \geq P_{fc}$$  \hspace{1cm} (B1-44)

### B1-4. Failure Criteria and Stiffness Reduction

The statistical failure criterion and other maximum stress or maximum strain strength criteria are combined together to predict the failure of individual material slice and overall textile composite. The maximum stress or maximum strain criteria are adopted from TExCAD code developed by Naik [3]. The failure criterion in equation (B1-44) is added to predict the tensile failure of yarn slices only. A matrix crack development model is also added for the transverse tensile failure for yarn slices and tensile failure of matrix slices. Textile composite failure is reached when the stiffness corresponding to each load is less than a critical value.

**Failure criteria for yarn slice**

Yarn material slices failures are divided into matrix dominated failure and fiber dominated failure. Shear failures in all direction and transverse failures belong to matrix dominated failure, and only tensile or compressive failure in the fiber direction are considered as fiber dominated failure.

For matrix dominated yarn slices failure, maximum stress criteria are used, the critical values are user inputted material parameters. For fiber dominated yarn slices failure, when the fracture mechanics model is used, the statistical failure criterion is employed, while for no-fracture mechanics model, the maximum strain criterion is used. The critical values, that is the
maximum and minimum strains are user supplied parameters, and the failure probability is given in the code.

Failure criteria for matrix slice
For matrix material slices, when the in-plane shear load is not applied, the maximum octahedral stress criterion are used. The critical value is calculated from the inputted matrix material strengths.

When there is no in-plane shear load, two failure criteria are employed. In addition to the octahedral stress criterion, a maximum principal stress criterion is used to judge the matrix failure. For fracture model, when the maximum principle stress is tension, the critical value for the maximum principle stress criterion is the stress at which the initial cracks in matrix material as a uniform material start to develop. When no-fracture model is chosen or the maximum principal stress is negative (compressive), the material strength of the matrix is used as the critical value.

Nonlinear Material
The nonlinear response of textile composites in MicroTex came from two sources. The first source is the assumed nonlinear material (matrix and yams). The user of MicroTex code could chose linear o nonlinear materials. When nonlinear material was chosen, the Young’s modulus and shear modulus of the matrix and yarn material was assumed to obey following form:

\[ E = E_0 \left[ 1 + \left( \frac{S}{S_c} \right)^\alpha \right]^{-\left( 1 + \frac{1}{\alpha} \right)} \]

where \( E_0 \) represents the initial Young's modulus or shear modulus, \( S \) is the corresponding stress level, \( S_c \) is the critical values of failure criteria used, \( \alpha \) and \( \beta \) are empirical constants, for matrix slices, \( \beta \) is taken as unity. For transverse Young's modulus and shear modulus (which are dominated by the matrix behavior) of yarn slices, the empirical constants \( \alpha \) and \( \beta \) are taken the same as the matrix slice.

When the fracture mechanics model is chosen, the yarn material properties in fiber direction changes due to the matrix crack development. This affect is included by assuming the Young’s modulus in fiber direction of the matrix in yarn slices obeys equation \((B1-45)\), where, of course, \( \beta \) is unity and \( S_c \) is the stress at which the fiber bridged crack develops.

Stiffness Reduction due to slice failure
The other source of stiffness reduction comes from material slice failure. When a material slice is failed based on judgments mentioned above in this section, the Young's moduli and shear moduli are reduced coordinately. For matrix material slices, all the moduli are reduced 99% when slice failure is reached. For yarn slices with matrix dominated failure, the corresponding Young's modulus and shear modulus are reduced to 1%, and for yarn slices, with fiber dominated failure, only the Young's modulus in fiber direction is reduced 99%.

Failure criteria for textile composites
The failure of textile composites is judged using the same concept as a structure failure. The textile composites are considered as systems of matrix slices and yarn slices, or a structure its components are material slices. A structure is failed when it cannot carry some kind of
loading, and here a textile composite is failed when the RUC of a textile composite cannot carry some kind of loading. In MiroTex, the composite failure is reached when the stiffness in the loading direction is reduced to a critical value, like 10% of the original value. This critical value is a user input parameter.

**B1-5. Numerical Examples for Textile Composites**

Numerical examples for various textile composites under various loading are given in this section. The textile architectures studied include laminated composites, 2D triaxial braided composites, and plain weave composites. The materials involved include C/SiC, Graphite/Epoxy, and Nicalon/CAS. The loading includes tension, compression, shear, and temperature change. The results include geometry parameters, overall stiffness, overall coefficients of thermal expansion, strengths, strain-stress relations, and stiffness reduction. The results are compared with available test data. Also in this chapter, the effect of some parameters are investigated.

Some of the material parameters are fixed in all the following examples. They are the Weibull modulus $M=10.0$, the mean fiber strength $\sigma_o = 388.4\text{MPa} - m (\sqrt{m})$, and the fiber failure probability $P_{fc} = 0.6$.

**Tension Failure of Laminated Composites**

The laminated composite is treated as a special case of textile composite in MicroTex, that is, layered yarns. Figures B1-10 shows the stress-strain results for Nicalon/CAS laminate composites under tension. The comparison with experimental results[3] was performed for room temperature (25°C) and 400°C, respectively. The mechanical and thermal properties for Nicalon fiber and CAS matrix are given in Table B1-1 (where values in () represent properties at 400°C)[3]. Significant agreement for damage progress and ultimate failure was obtained, the predicted strength and ultimate strain for room temperature case are 380.0MPa and 1.007%, and 390.0MPa and 1.354% for 400°C case. The shear stress parameter is taken as 0.4 in room temperature case and 0.7 in 400°C case, and the composite failure stiffness is assumed as 1% of the initial stiffness. The two nonlinear parameters for yarn ($\alpha$ and $\beta$) are taken to be 4.3 and 0.63 for both cases. The increase of shear stress parameter with the increase of temperature is reasonable because the Nicalon/CAS material has a compressive radial thermal stresses at the fiber-matrix interface with increasing temperature.

<table>
<thead>
<tr>
<th>Fiber Properties</th>
<th>Matrix Properties</th>
</tr>
</thead>
<tbody>
<tr>
<td>$E_f = 195\text{GPa}$ (175GPa)</td>
<td>$E_m = 95\text{GPa}$ (94GPa)</td>
</tr>
<tr>
<td>$v_f = 0.2$</td>
<td>$v_m = 0.3$</td>
</tr>
<tr>
<td>$\alpha_f = 3.0e - 06/\circ C$ (4.0e - 06/° C)</td>
<td>$\alpha_m = 5.0e - 06/\circ C$</td>
</tr>
<tr>
<td>$K_{lc} = 2.16\text{MPa} \sqrt{m}$ (1.73MPa√m)</td>
<td></td>
</tr>
</tbody>
</table>

**2D triaxial braided composites**

Experimental results for 2D triaxial braided Graphite/Epoxy composites with different architectures under uniaxial or biaxial tension and compression [7-9] are compared to MicroTex predictions. Two sets of notation are used to describe each individual architecture[7]. The first is
a shorthand notation consisting of three letters, the second one is a longhand notation which describes in detail the construction of the braid as shown in Figure B1-11. The first number in the longhand notation indicates the angle for the axial yarns with respect to the longitudinal axis, its subscript designates the size of the axial yarns in thousands of fibers. The second number indicates the angles of the braid yarns with respect to the longitudinal axis; its subscript designates the size of the braid yarns in thousands of fibers. The subscript outside the brackets is the percent of the total volume fraction of the yarns occupied by axial yarns.

Uniaxial tension failure of 2D triaxial braided Graphite/Epoxy composites

Burr and Morris [7] gave experimental data for 2D triaxial braided composites subject to uniaxial tension. The materials are Graphite fiber and Epoxy matrix. Four different braided architectures, so-called LLL, SLL, LLS, and SSL, were investigated. The corresponding shorthand and longhand notation and geometry parameters used in this calculation are listed in Table B1-2, and the calculated geometry parameters of the RUC are given Table B1-3. Since there are no details of material properties, the data from reference [8] are adopted and listed in Table B1-4, and the yarn fiber volume fractions are taken as 0.75. The predicted Young's moduli of the four architectures gives good agreement with the test data (Figure B1-12), that indicate this adoption is reasonable. The comparison of the present strain-stress results with the testing data are shown in Figure B1-13. The shear stress parameters used in the analysis, and the comparison of strengths and ultimate strains are given in Table B1-5. Since only one tension test for each architecture is available, the shear stress parameters in both axial and braided yarns are assumed to be the same.

Table B1-2. Architecture parameters for 2D triaxial braided composites

<table>
<thead>
<tr>
<th>Architecture</th>
<th>Axial yarn size(K)</th>
<th>Braid yarn size(K)</th>
<th>Braid angle (degree)</th>
<th>Axial Yarn spacing</th>
<th>Axial yarn percentage</th>
<th>Layer thickness</th>
</tr>
</thead>
<tbody>
<tr>
<td>LSS</td>
<td>6</td>
<td>15</td>
<td>45.0</td>
<td>6.95mm</td>
<td>12%</td>
<td>0.500mm</td>
</tr>
<tr>
<td>LLS</td>
<td>36</td>
<td>15</td>
<td>45.0</td>
<td>5.35 mm</td>
<td>46%</td>
<td>0.927mm</td>
</tr>
<tr>
<td>SLL</td>
<td>30</td>
<td>6</td>
<td>70.0</td>
<td>6.1mm</td>
<td>46%</td>
<td>0.605mm</td>
</tr>
<tr>
<td>LLL</td>
<td>75</td>
<td>15</td>
<td>70.0</td>
<td>11.2 mm</td>
<td>46%</td>
<td>0.935mm</td>
</tr>
</tbody>
</table>

Table B1-3. Calculated geometry parameters for 2D triaxial braided composites

<table>
<thead>
<tr>
<th>Architecture</th>
<th>Axial yarn thickness</th>
<th>Braid yarn thickness</th>
<th>Crimp angle (degree)</th>
<th>Axial Yarn length</th>
<th>Braid yarn length</th>
<th>Volume fraction</th>
</tr>
</thead>
<tbody>
<tr>
<td>LSS</td>
<td>0.36 mm</td>
<td>0.07 mm</td>
<td>3.22</td>
<td>47.60 mm</td>
<td>17.45 mm</td>
<td>0.65</td>
</tr>
<tr>
<td>LLS</td>
<td>0.66 mm</td>
<td>0.13 mm</td>
<td>11.68</td>
<td>42.80 mm</td>
<td>15.07 mm</td>
<td>0.61</td>
</tr>
<tr>
<td>SLL</td>
<td>0.43 mm</td>
<td>0.09 mm</td>
<td>7.74</td>
<td>17.76 mm</td>
<td>13.03 mm</td>
<td>0.68</td>
</tr>
<tr>
<td>LLL</td>
<td>0.67 mm</td>
<td>0.13 mm</td>
<td>5.61</td>
<td>32.61 mm</td>
<td>23.93 mm</td>
<td>0.60</td>
</tr>
</tbody>
</table>

Table B1-4. Yarn and matrix properties used for 2D triaxial braided composites

<table>
<thead>
<tr>
<th>Material</th>
<th>$E_{11}$ (GPa)</th>
<th>$E_{22}$ (GPa)</th>
<th>$G_{12}$ (GPa)</th>
<th>$\nu_{12}$</th>
<th>$\nu_{23}$</th>
<th>$\alpha_{11}(10^{-6}/^\circ C)$</th>
<th>$\alpha_{22}(10^{-6}/^\circ C)$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yarn</td>
<td>144.80</td>
<td>11.73</td>
<td>5.52</td>
<td>0.23</td>
<td>0.30</td>
<td>-0.324</td>
<td>14.0</td>
</tr>
<tr>
<td>Matrix</td>
<td>3.45</td>
<td>3.45</td>
<td>1.28</td>
<td>0.35</td>
<td>0.35</td>
<td>40.0</td>
<td>40.0</td>
</tr>
</tbody>
</table>
Table B1-5. Failure Strength and ultimate strain for 2-D braided composite under tension.

<table>
<thead>
<tr>
<th>Architecture</th>
<th>Shear Stress Parameter (MPa)</th>
<th>Failure Strength (MPa)</th>
<th>Failure Strain (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>LSS</td>
<td>1.5</td>
<td>370.0</td>
<td>1.24</td>
</tr>
<tr>
<td>LLS</td>
<td>0.6</td>
<td>644.5</td>
<td>0.91</td>
</tr>
<tr>
<td>SLL</td>
<td>3.0</td>
<td>843.5</td>
<td>1.33</td>
</tr>
<tr>
<td>LLL</td>
<td>0.006</td>
<td>487.1</td>
<td>0.93</td>
</tr>
</tbody>
</table>

Both experiment and prediction indicate for LLS, SLL, and LLL architectures, the load-response is very linear, while for LSS case, the response is nonlinear (Figure B1-13). The analysis indicates the non-linearity of LSS architecture is due to the large shear stresses in braid yarns. Figure B1-14 shows the decrease of tangent axial modulus and failure information for LSS architecture. It indicates that the in-plane shear failure in braided yarns contributes to the non-linearity.

Biaxial tension failure of 2D triaxial braided AS4/1895 composites

Swanson and Smith [9] reported the experimental results of four different braided architectures consisting of AS4/1895 carbon fibers and epoxy matrix subject to biaxial tension loads. The yarn and matrix properties are given in Table B1-6 as AS4/3501-6 lamina with a fiber volume fraction of 0.60. The measured crimp angles given in reference [9] are used in calculating the necessary input data for the MicroTex analysis. Tables B1-7 and B1-8 give the input geometry parameters and calculated dimensions from MicroTex. The predicted axial and hoop stiffness and the in-plane Poisson’s ratios are compared with testing data for four different architectures in Figures B1-15 to B1-17. Reasonable agreement can be observed.

MicroTex was then used to predict the composite strength under biaxial tension. The shear stress parameters of axial yarn and braid yarns are assumed to be different, and determined using experimental results for uniaxial tension in the axial and hoop direction, respectively. The obtained shear stress parameters are listed in Table B1-9. The shear stress parameters are then used to predict the strengths of the four different architectures, and the results were compared with testing data in Figures B1-18 to B1-21. The agreement between experimental results and predictions is reasonable for all four architectures under various axial stress and hoop stress ratios.

Table B1-6. Yarn and resin properties used for 2D triaxial braided composites

<table>
<thead>
<tr>
<th>Material</th>
<th>$E_{11}$ (GPa)</th>
<th>$E_{22}$ (GPa)</th>
<th>$G_{12}$ (GPa)</th>
<th>$\nu_{12}$</th>
<th>$\nu_{23}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yarn</td>
<td>127.0</td>
<td>11.0</td>
<td>6.55</td>
<td>0.28</td>
<td>0.30</td>
</tr>
<tr>
<td>Matrix</td>
<td>3.45</td>
<td>3.45</td>
<td>1.28</td>
<td>0.35</td>
<td>0.35</td>
</tr>
</tbody>
</table>

Table B1-7. Architecture parameters for 2D triaxial braided composites

<table>
<thead>
<tr>
<th>Architecture</th>
<th>Axial yarn size (K)</th>
<th>Braid yarn size (K)</th>
<th>Braid angle (degree)</th>
<th>Axial Yarn spacing</th>
<th>Axial yarn percentage</th>
<th>Layer thickness</th>
</tr>
</thead>
<tbody>
<tr>
<td>LSS</td>
<td>9</td>
<td>12</td>
<td>47.0</td>
<td>4.65 mm</td>
<td>20%</td>
<td>0.79 mm</td>
</tr>
<tr>
<td>LLS</td>
<td>27</td>
<td>12</td>
<td>45.0</td>
<td>4.54 mm</td>
<td>44%</td>
<td>1.06 mm</td>
</tr>
<tr>
<td>SLL</td>
<td>33</td>
<td>6</td>
<td>73.0</td>
<td>9.73 mm</td>
<td>35%</td>
<td>0.64 mm</td>
</tr>
<tr>
<td>LLL</td>
<td>54</td>
<td>12</td>
<td>70.0</td>
<td>9.21 mm</td>
<td>43%</td>
<td>1.06 mm</td>
</tr>
</tbody>
</table>

B - 1 - 13
Table B 1-8. Calculated geometry parameters for 2D triaxial braided composites

<table>
<thead>
<tr>
<th>Architecture</th>
<th>Axial yarn thickness (mm)</th>
<th>Braid yarn thickness (mm)</th>
<th>Crimp angle (degree)</th>
<th>Axial Yarn length (mm)</th>
<th>Braid yarn length (mm)</th>
<th>Volume fraction</th>
</tr>
</thead>
<tbody>
<tr>
<td>LSS</td>
<td>0.57</td>
<td>0.12</td>
<td>7.11</td>
<td>34.69</td>
<td>13.01</td>
<td>0.47</td>
</tr>
<tr>
<td>LLS</td>
<td>0.76</td>
<td>0.15</td>
<td>15.04</td>
<td>36.32</td>
<td>13.00</td>
<td>0.49</td>
</tr>
<tr>
<td>SLL</td>
<td>0.46</td>
<td>0.91</td>
<td>4.10</td>
<td>23.80</td>
<td>14.82</td>
<td>0.58</td>
</tr>
<tr>
<td>LLL</td>
<td>0.76</td>
<td>0.15</td>
<td>7.44</td>
<td>26.82</td>
<td>20.00</td>
<td>0.50</td>
</tr>
</tbody>
</table>

Table B 1-9. Shear stress parameters for 2D triaxial braided composites

<table>
<thead>
<tr>
<th>Architecture</th>
<th>Shear stress parameter for axial yarns (MPa)</th>
<th>Shear stress parameter for braid yarns (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>LSS</td>
<td>0.60</td>
<td>3.0е-06</td>
</tr>
<tr>
<td>LLS</td>
<td>0.12</td>
<td>1.0е-08</td>
</tr>
<tr>
<td>SLL</td>
<td>0.50</td>
<td>2.0е-08</td>
</tr>
<tr>
<td>LLL</td>
<td>0.012</td>
<td>5.0е-07</td>
</tr>
</tbody>
</table>

Plain weave composite under in-plane shear stress

2-D plain weave composites under tension, compression, and shear loading[8] are analyzed here using MicroTex. The fiber is Hercules AS4 graphite and the matrix is Hercules 3501-6 epoxy, and yarn fiber volume fraction is 0.70, the material properties are given in Table B 1-4. The results are compared with testing data[8] in Table B 1-10 for strengths and stress-strain relation in Figures B 1-22 and B 1-23. The shear stress parameters in those calculation were taken as 0.3MPa.

Plain weave C/SiC composites under tension

Testing and analysis on C/SiC plain weave composites under uniaxial tension were performed at AS&M. The material properties for T300 fiber and SiC matrix are listed in Table B1-11, and also the input and calculated geometry parameters in Tables B1-12 and B1-13. The testing data for the two-layer (0/90) plain weave composite are employed to determine the shear stress parameter (we assume that the warp and fill yarns are identical, thus the shear stress parameters for those two yarns are the same) and other parameters like nonlinear parameters for yarns and matrix; the agreement is excellent as shown in Figure B1-24. Those parameters are
then applied to predict the strain-stress relation and strength for the three-layered (0/-60/60) C/SiC plain weave composite, the results are compared in Figures B-1-25 with the experimental results. The prediction indicated that the failure of the (0/+60/-60) composite is due to in-plane shear stresses. Considering the assumption of identical warp and fill yarns, the results is reasonable.

The MicroTex code is then employed to analyze the failure processes of this composite under tension at different directions. The results are shown in Figure B1-26. The strength is largely reduced when the applied load is off axial. For axial load (0°), the failure is due to tension failure in loading direction, the strength is 502MPa and the ultimate strain is 0.1235%. For all the others, in-plane shear failure of the yarns causes the composite failure. Also the stress-strain results of two-layer angle-ply plain weave composite are shown in Figure B-27.

Table B1-11. Fiber and matrix properties used for C/SiC Plain weave composites

<table>
<thead>
<tr>
<th>Material</th>
<th>Young’s Modulus (GPa)</th>
<th>Possion’s ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fiber</td>
<td>231.0</td>
<td>0.20</td>
</tr>
<tr>
<td>Matrix</td>
<td>3.51</td>
<td>0.22</td>
</tr>
</tbody>
</table>

Table B1-12. Architecture parameters for C/SiC plain weave composites

<table>
<thead>
<tr>
<th>Yarn size(K)</th>
<th>Braid angle</th>
<th>Yarn spacing</th>
<th>Yarn fiber volume fraction</th>
<th>Layer thickness</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>47.0</td>
<td>3.22 mm</td>
<td>0.6</td>
<td>0.60 mm</td>
</tr>
</tbody>
</table>

Table B1-13. Calculated geometry parameters for C/SiC plain weave composites

<table>
<thead>
<tr>
<th>Yarn thickness</th>
<th>Crimp angle</th>
<th>Yarn length</th>
<th>Volume fraction</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.30 mm</td>
<td>2.26</td>
<td>6.40 mm</td>
<td>0.40</td>
</tr>
</tbody>
</table>

The tension strength of plain weave composites with different crimp angles are given in Figure B1-28 and compared with the laminate composite case. It can be seen that the increase of crimp angle will decrease the composite strength. Detailed analysis also indicated that, when the crimp angle is less than 10°, the composite failure is caused by loading direction tension. When the crimp angle is larger than 10°, failure is caused by local transverse shear failure of axial yarns.

B1-6. Fatigue Simulations for Plain Weave Composites

It is well-known that fatigue analysis is experimental orientated, that experimental works play an critical role. Fatigue testing was performed in this project for both (0/90) and (0/-60/60) C/SiC plain weave composites. The stress ratio and frequency for both cases are 0.1 and 5Hz, respectively. In this section, a fatigue damage model was developed based on the observation of experimental data. This model assumed that the stiffness reduction during the fatigue process contains all the information about the effect of fatigue damage to material strength, and assumed, for simplicity, that the fatigue damage to all the materials (matrix and yarn materials) are proportional.

The fatigue model

The mean strength in the Weibull distribution and all the critical values in material failure criteria are related to the stiffness reduction of the composite to reflect the affect of fatigue loading on the composite strength. They can be expressed as
\[
\sigma_N = \sigma_o \left( \frac{E}{E_0} \right)^M \left(b_1 + b_2 \sigma_{\text{max}} \right) \sigma_s
\]  
(B1-46)

\[
\sigma_{cN} = \sigma_c \left( \frac{E}{E_0} \right)^M \left(b_1 + b_2 \sigma_{\text{max}} \right) \sigma_s
\]  
(B1-47)

where \( b_1 \) and \( b_2 \) are parameters, \( M \) is the Weibull modulus, \( \sigma_{\text{max}} \) is the maximum loading, \( \frac{E}{E_0} \) is the overall stiffness reduction, \( \sigma_o \) is the mean strength of the yarn material, and \( \sigma_N \) is the mean strength after \( N \) cycles, \( \sigma_c \) is the original critical value in any failure criteria, and \( \sigma_{cN} \) is the critical value after \( N \) cycles, and \( \sigma_s \) is the static strength of composite.

**Stiffness reduction due to fatigue damage**

Fatigue testing for the C/SiC (0/90) plain weave composite analyzed in previous section was performed. Based on the observation of testing data, the stiffness reduction for (0/90) C/SiC plain weave composite under fatigue loading is assumed as:

\[
\frac{E}{E_0} = \frac{1}{1 + \left(c_1 + c_2 \sigma_{\text{max}} \right) \log(N)}
\]  
(B1-48)

where \( N \) is the number of cycle, \( \sigma_{\text{max}} \) is the maximum stress, \( c_1 \) and \( c_2 \) are material constants to be determined by experimental data.

**Fatigue results**

The parameters in equations (B1-46) and (B1-47) are determined from experimental stiffness reduction data and the S-N curve for (0/90) composite. Then those parameters are substituted into these two equations to predict the fatigue damage for (0/-60/60) composite.

Using the stiffness reduction data for (0/90) composite, the two parameters in equation (B1-48) were determined as:

\[ c_1 = -1.54e - 02, \quad c_2 = 7.32e - 05 \text{ / MPa} \]

The comparisons of stiffness reductions for maximum stress at 458.85MPa and 410.6MPa were given in Figures B1-29 and B1-30 as examples.

The S-N data of (0/90) composite are then used to determine two constants in equations (B1-46) and (B1-47) with the help of equation (B1-48) which provides the stiffness reduction relation. The two coefficient were obtained as

\[ b_1 = 0.0415 \text{ / MPa}, \quad b_2 = -8.22e - 05 \]

The comparison with testing data was shown in Figure B1-31.

The four parameters and the static strength of (0/-60/60) C/SiC composite, which were obtained in the previous section as 272.0MPa (experimental strength 289.8MPa), are used to predict the fatigue curve for (0/-60/60) C/SiC plain weave composite. The predicted S-N curve is compared with experimental result in Figure B1-32, and predicted stiffness reductions for maximum stress at 248.0MPa and 258.0MPa are shown in Figure B1-33.

**Discussion**

Since all the fatigue testing were performed under same stress ratio and frequency, we were unable to include two very important parameters in this model. As a consequence, the static strength is used as one point in the S-N curve.
B1-7. Conclusions

Geometry models for various textile composites are presented in this section. The RUC is used to represent the textile composite by taking advantage of the periodically repeating pattern of textile composite materials. The RUC is represented by a set of yarn and matrix slices based on the sinusoidal assumption of the undulation portion of the yarn paths. Thermal and mechanical properties calculation of the RUC are then carried out based on an iso-strain assumption.

An analytical approach based on fracture mechanics, shear-lag theory, and fiber pull-out concept is developed for a statistical failure criterion of the yarn slice under tension. Statistical analysis based on the fiber pull-out concept is used as a yarn failure criterion, while shear-lag analysis and fracture mechanics are employed to determine relations needed for statistical computation.

Stiffness reduction of RUC comes from the material non-linearity and the failure of material slices. The complete set of failure criteria, including the statistical criterion, is proposed for yarn and matrix slices within the RUC. When a material slice failure is reached, the corresponding moduli are reduced.

A textile composite analysis code, named MicroTex, has been developed by following the above mentioned engineering approach. The following capabilities were included: geometry analysis for various textile composite architectures; calculation of mechanical and thermal properties, such as overall Young’s moduli, Poisson’s ratios, and coefficients of thermal expansion for RUCs; calculation of strains and local stresses using plate theory or 3D iso-strain model; local material slice failure judgment; stiffness reduction due to material slice failure and non-linear material properties; and overall composite failure judgment.

The textile composite or its RUC is considered as a structure with yarn and matrix slices as its components. When this structure can not resist some kind of loading, the composite or RUC reaches its failure point.

Fatigue analysis is performed based on the assumption that simple fatigue test for the same textile composite is performed.

The prediction based on above analysis are compared with experimental results for laminate composites, plain weave composites under uniaxial loading, and various 2D triaxial braided composites under both uniaxial and biaxial tensions. The agreement is reasonable.

Figure B1-1. MicroTex for textile composites analysis.
COORDINATE SYSTEMS FOR 2-D LAMINATE

- 1-2-3 Coordinates refer to yarn material coordinates. The 2-axis is assumed to be always perpendicular to the Z-axis.

Figure B1-3. 2D laminated composite and coordinate systems
UNIT CELL FOR 5-HARNESS SATIN WEAVE

Figure B1-4. RUC of 5-harness satin weave composite

UNIT CELL FOR 2-D BRAIDED COMPOSITE

Figure B1-5. RUC of 2D plain braid composite
UNIT CELL MODEL FOR 1x1 2-D TRIAXIAL BRAID

Figure B1-6. RUC of 2D, 1x1 triaxial braid composite

UNIT CELL FOR 2x2, 2-D TRIAXIAL BRAID

Figure B1-7. RUC of 2D, 2x2 triaxial braid composite
ORIENTATION ANGLES FOR 3-D COMPOSITE

1-2-3 Coordinates refer to yam material coordinates. The 2-axis is assumed to be always perpendicular to the Z-axis.

Figure B1-8. 3D oriented composite
Shear-lag analysis

Fracture mechanics analysis

Figure B1-9. Shear-lag analysis and fracture mechanics model
Figure B1-10. Stress-strain Result for Nicalon/CAS laminate composite under tension

Figure B1-11. Longhand nomenclature for 2D triaxial braided composites

\[
\begin{bmatrix}
0 & 12 & K \\
\pm 45 & 30 & K \\
\end{bmatrix}^{46\% \text{ axial}}
\]

Percentage of Axial Bundles
Figure B1-12. Elastic moduli for 2D triaxial braided Graphite/Epoxy composites

Figure B1-13. Stress-strain results for 2D triaxial braided Graphite/Epoxy composites
Figure B1-14. Failure process for 2D triaxial braided composite: architecture SLL.

Figure B1-15. Comparison of axial Young’s moduli for 2D triaxial braid composites.
Figure B1-16. Comparison of Young's moduli in hoop direction for 2D triaxial braid composites

Figure B1-17. Comparison of in-plane Poisson's ratios for 2D triaxial braid composites
Figure B1-18. Comparison of Strength for Architecture LSS

Figure B1-19. Comparison of Strength for Architecture LLS
Figure B1-20. Comparison of Strength of Architecture SLL

Figure B1-21. Comparison of Strength of Architecture LLL
Figure B1-22. Shear stress-strain response for the plain weave composite (3k yarns)

Figure B1-23. Shear stress-strain response for the plain weave composite (12k yarns)
Figure B-24. Stress-strain results for a (0/90) C/SiC plain weave composite

Figure B-25. Stress-strain results for a (0/+60-60) C/SiC plain weave composite
Figure B1-26. Stress-strain results for plain weave C/SiC composite under tension at different directions

Figure B1-27. Strain-stress results for angle-ply plain weave composite under tension
Figure B1-28. Tension strength of plain weave composite with different crimp angles.

Figure B1-29. Stiffness Reduction for (0/90) C/SiC Plain Weave Composite for Maximum Stress at 458.85Mpa.
Figure B1-30. Stiffness Reduction for (0/90) C/SiC Plain Weave Composite for Maximum Stress at 410.60Mpa.

Figure B1-31. S-N Curve for (0/90) C/SiC Plain Weave Composite
Figure B1-32. S-N Curve for (0/-60/+60) C/SiC Plain Weave Composite

Figure B1-33. Stiffness Reduction for (0/-60/60) C/SiC Plain Weave Composite for Maximum Stresses at 258.0MPa and 248.0MPa.
Introduction

The Micro-mechanics analysis for Textile composites (MicroTex) code provides the materials/design engineer with a user-friendly tool for the analysis of a wide variety of multi-layered, oriented, fabric reinforced woven and braided composites and laminated composites. Improved from a prior code (TEXCAD), MicroTex can be used to calculate overall thermal and mechanical properties along with engineering estimates of damage progression and strength. This code discretely models the yarn centerline paths within the textile repeating unit cell (RUC) by assuming sinusoidal undulations at yarn cross-over points and uses a yarn discretization scheme (which subdivides each yarn into smaller, piecewise straight yarn slice) together with either a 3D strain averaging procedure or a thin composite plate theory to calculate overall thermal and mechanical properties, stress distributions, and layer average strains[1-3].

For the failure analysis, the textile composite failure is defined as the loss of loading capability of the RUC, which depends on the stiffness reduction due to material slice failure and nonlinear material properties. Two models, a fracture model and a continuum model, are employed for material slices (matrix and yarn slices) failure analysis. In addition to the maximum...
strain and maximum stress criteria of TEXCAD, a statistical criterion based on shear-lag model, fracture mechanics, and statistical principles are used to predict the tensile failure of yarn slices.

A fatigue analysis capability is also included in MicroTex. The damage due to fatigue loading is considered by overall stiffness reduction and material (matrix and yarn) strengths decreasing with cycle load.

Input to MicroTex consists of: (1) material parameters like yarn (or fiber) and matrix properties such as moduli, Poisson’s ratios, coefficients of thermal expansion, nonlinear parameters, shear stress parameters, matrix stress intensity factor, initial matrix crack length, failure strains and stresses; (2) fabric parameters like yarn sizes, braid angle, yarn packing density, fiber diameter, yarn spacing, composite layer thickness; (3) applied loading like temperature and applied stresses. MicroTex output includes overall thermoelastic constants, yarn and matrix slices strains and stresses, slice failure history, in-plane stress-strain relation, ultimate strength, and overall stiffness reductions.

This manual consists of two parts. The first part includes the installation. The second part is a demonstration of the MicroTex application. The input to MicroTex is explained in detail according to the windows of the GUI and demonstrated using input files, output files are also explained with a sample problem.

System Requirement

MicroTex can be run on PC or UNIX system. Since the GUI is written by JAVA, it requires JDK (Java Development Kit 1.1.3 or higher version) on the PC or UNIX system.

Installation and Execution

The software is distributed in tar format file asm.tar. The file contains all the required data files, source files and java class files required to execute the programs. To install the software components copy change directory to the destination directory and extract the files using the command: tar -xvf asm.tar. This command creates the ASM main directory and sub directories to copy the CPFail, MicroTex and ABAQUS interface programs. After extracting the files the default executable file names and paths have to set before running the any sample problems. This can be set either manually editing the configuration files or by executing the programs and modifying through the GUI using the menu command Configuration under Options menu. The executable commands for MicroTex are listed below:

change directory to \ASM\TEX\MICROTEX
execute java asm.microtex.TXApp command

The MicroTex is developed using Sun’s JDK 1.1.4. To execute these components Java runtime environment of version 1.1.3 or higher is required. Refer the JDK or JRE documentation for installation of Java.
Original code Files:
The following files are the original code files, user should carry over when change software installation directory, user can also carry over old saved files for previous cases:

- pctexcad.exe  
- utexcad.exe  
- /asm/abaqus/.*  
- microtex.gif  
- microtex.cfg  
- TXMatrix.db  
- TXYarn.db  

(for PC version, modify microtex.cfg)  
(for UNIX version, modify microtex.cfg)  
(all files)

MicroTex Input

MicroTex is a user-friendly textile composites analysis code. Based on micro-geometry and micro-mechanics analysis, MicroTex can perform geometry analysis for various textile composite architectures which includes laminated composites, 2D, 2x2 and 1x1 triaxial braids, weaves (plain weave, 5-harness satin weave, and 8-harness satin weave), 2D braids (2D plain braid, 5-harness satin braid, and 8-harness satin braid), and 3D spatial oriented composites, and calculation of mechanical and thermal properties, such as overall Young’s moduli, Poisson’s ratios, and coefficients of thermal expansion for RUCs. MicroTex can calculate average and local strains and local stresses using plate theory or 3D iso-strain model, and performs local material slice failure judgment. MicroTex can also perform the overall stiffness reduction calculation due to material slice failure and non-linear material properties, and overall composite failure judgment.

MicroTex can be executed for static analysis by entering all the required material and architecture parameters through MicroTex Graphical User Interface (GUI). The GUI will then generate four data files: control.dat for control parameters, geom.dat for architecture parameters, mater.dat for material parameters, and load.dat for loading information. For fatigue analysis, MicroTex can be executed by providing fatigue information through the MicroTex GUI, and the GUI will generate an additional data file, fatigue.dat.

Control data
Control data are input on “Control” window. The “Failure criteria” is for the judgment of composite failure. If the user assumes that the composite is failed when the overall stiffness of RUC is reduced a% of the initial stiffness, the number “a” is the input value.

This window also provides the options between static or fatigue analysis, plate model or 3D solid model, linear or nonlinear materials, and also if the fracture model should be employed.

If the fracture model is the choice, the statistical failure criterion will be used instead of the maximum tension strain criterion of yarns. Also matrix crack development will replace the strengths criteria to judge if a matrix slice is failed.

When the plate model is used, MicroTex will employ thin composite plate theory to analyze the stress and strain distribution among different layers, while for solid model, the strains are assumed to be uniform through thickness.
When the nonlinear material is used, MicroTex assumes both matrix and yarn material are nonlinear, and obey following equation

\[
E = E_0 \left[1 + \left(\frac{s}{\beta s_c}\right)^\alpha\right]^{(1 + \frac{1}{\alpha})}
\]  

(B2-1)

where \(E_0\) represents the initial Young’s modulus, \(S\) is the corresponding stress level, \(S_c\) is the critical values of failure criteria used, \(\alpha\) and \(\beta\) are empirical constants; for matrix slices, \(\beta\) is taken as unity. Noticing that the transverse and shear moduli of the yarn slice are dominated by matrix material, the empirical constant \(\alpha\) for transverse and shear moduli for the yarn slice is taken as the same value as matrix material, and \(\beta\) is unity. In the case of linear model, the only stiffness reduction comes form material slice failure.

When fatigue analysis is wanted, the user will be required to supply fatigue damage information like stiffness reduction due to fatigue loading for basic fatigue testing. This version assumes that the stiffness reduction due to fatigue loading obeys:

\[
\frac{E}{E_0} = \frac{1}{1 + (c_1 + c_2 \sigma_{\text{max}}) \text{Log}(N)}
\]  

(B2-2)

where modulus \(\frac{E}{E_0}\) is the overall stiffness reduction, \(N\) is the number of cycles, \(\sigma_{\text{max}}\) is the maximum stress, \(c_1\) and \(c_2\) are material constants to be determined by experimental data. The mean strength in the Weibull distribution and all the critical values in material failure criteria are assumed to be related to the stiffness reduction of composite to reflect the affect of fatigue loading on the composite strength. They can be expressed as

\[
\sigma_N = \sigma_o \left(\frac{E}{E_0}\right)^{M(b_1 + b_2 \sigma_{\text{max}})}\sigma_s
\]  

(B2-3)

\[
\sigma_{cN} = \sigma_c \left(\frac{E}{E_0}\right)^{M(b_1 + b_2 \sigma_{\text{max}})}\sigma_s
\]  

(B2-4)

where \(b_1\) and \(b_2\) are parameters, \(M\) is the Weibull modulus, \(\sigma_o\) is the mean strength of the yarn material, and \(\sigma_N\) is the yarn mean strength after \(N\) cycles, \(\sigma_c\) is the original critical value in any failure criteria, and \(\sigma_{cN}\) the critical value after \(N\) cycles, and \(\sigma_s\) is the static strength of composite which can be obtained by performing static analysis.
Architecture data

Architecture data can be provided on window "Geometry". The user is required to input the number of layers, layer orientation, layer thickness, and type of architectures. Then for each different architecture, the information user should provided are given as follows:

Architecture Option No. 1 -- Laminated composite

This option provides a classical lamination theory analysis for modeling two dimensional, stacked, orientated yarns (or plies). MicroTex requires material name for each ply. The layer orientations for laminated composite are measured with respect to x-axis (see Figure B2-1). The ply longitudinal and transverse direction are indicated by 1 and 2, respectively.

COORDINATE SYSTEMS FOR 2-D LAMINATE

\[
\begin{array}{c}
\text{Yarn} \\
\text{x}
\end{array}
\]

- 1-2-3 Coordinates refer to yarn material coordinates. The 2-axis is assumed to be always perpendicular to the Z-axis.

Figure B2-1. Reference coordinate system for the 2D laminated composite
Architecture Option No. 2 -- 2D, 2x2 triaxial braids

This option allows the user to model two dimensional, 2x2 triaxial braids. For each layer, MicroTex requires braid angle, axial yarn spacing, axial yarn contest, number of fiber in axial and braid yarns, and also material name ID for both matrix and yarns. The RUC is shown in Figure B2-2.

UNIT CELL FOR 2x2, 2-D TRIAXIAL BRAID

Figure B2-2. RUC for the 2D, 2x2 triaxial braided composites
Architecture Option No. 3 -- 2D, 1x1 triaxial braids

This option allows the user to model two dimensional, 1x1 triaxial braids. The RUC of this architecture is shown in Figure B2-3. For each layer, MicroTex requires braid angle, axial yarn spacing, axial yarn contest, number of fiber in axial and braid yarns, and also material name ID for both matrix and yarns.

UNIT CELL MODEL FOR 1x1 2-D TRIAXIAL BRAID

Figure B2-3. RUC for the 2D, 1x1 triaxial braided composites
Architecture Option No. 4 -- Weave and 2D braids

This option allows the user to model two dimensional weaves and two dimensional braids. The RUCs for plain weave, 5-harness satin weave, 8-harness satin weave, and plain 2D braid are shown in Figures B2-4 to B2-7.

For each layers, MicroTex requires weave type (for plain weave and plain 2D braid, 2; 5-harness satin weave and 5-harness satin 2D braid, 5; and for 8-harness satin weave and 8-harness satin 2D braid, 8), braid angle (for plain weave, 5-harness satin weave and 8-harness satin weave, 45 degree), axial yarn spacing , number of fiber in yarns, and also material name ID for both matrix and yarns.

UNIT CELL MODEL FOR PLAIN WEAVE COMPOSITE

Figure B2-4. RUC for the plain weave composites
Figure B2-5. RUC for the 5-harness satin weave composites

Figure B2-6. RUC for the 8-harness satin weave composites
UNIT CELL FOR 2-D BRAIDED COMPOSITE

- Yarn ID
- Yarn spacing

X - Cross-over points

Projected length
Crimp angle
Vertical shift

Unit cell thickness
Undulating length

SECTION A:A

Figure B2-7. RUC for the plain 2D braided composites
Architecture Option No. 5 -- 3D spatially Oriented Composites

This option allows the user to model a three dimensional, spatially oriented composite which is made up of straight yarns oriented along different directions. The RUC is shown in Figure B2-8. For each layer, MicroTex requires the lengths of two sides of the unit cell, included angle, number of yarns, orientation, length, and cross-section area of each yarn, and also material name ID for both matrix and yarns.

ORIENTATION ANGLES FOR 3-D COMPOSITE

2-a is assumed to be always perpendicular to the Z-axis.

Figure B2-8. Reference coordinate system for the 3D spatially oriented composite
Loading information

Loading information should be input on the window "Loading". The user is required to provide processing temperature, temperature load, applied forces (stresses) in each direction, and number of steps. MicroTex will apply the temperature at the first step and equally divide the applied forces into each step.

Materials

Material parameters, for both yarn and matrix, are inputted use “database” windows.

Yarn Database:
This window allows user to input yarn material properties. The yarn is assumed to be anisotropic. The fiber direction here is denoted as 1, 2 and 3 are transverse directions.

The "kind" has two options: fiber or yarn. When fiber is chosen, the Young's moduli ( $E_{11}$ for fiber direction, $E_{22}$ and $E_{33}$ for transverse direction), shear moduli ($G_{12}$, $G_{13}$, and $G_{23}$), Passion's ratios ($\mu_{12}$, $\mu_{13}$, and $\mu_{23}$), and coefficient of thermal expansion ($\alpha_{1}$, $\alpha_{2}$, and $\alpha_{3}$) should be entered. MicroTex will calculate the correspondent yarn parameters. If yarn is the choice, yarn properties should be inputted, and MicroTex will calculate the corresponding fiber parameters.

The "Assoc. Matrix" specifies the matrix material used in this yarn.

For the yarn material strengths, the maximum tension ($E_{11T}$) and compression ($E_{11C}$) strains in fiber direction, maximum tension, compression, and shear stresses in transverse direction ($S_{22T}$, $S_{22C}$, $S_{33T}$, $S_{33C}$, $S_{12S}$, $S_{23S}$, and $S_{13S}$), should be also input on this window.

B - 2 - MicroTex Manual - 12
The two nonlinear parameters for yarn material are defined to include possible material nonlinearity, they are the constant $\beta$ and $\alpha$ in equation (B2-1).

The last three parameters are the shear stress parameter, fiber diameter, and yarn fiber volume fraction. The shear stress parameter is the maximum shear stress that the fiber-matrix interface can resist, and an empirical constant which be determined by testing.

Matrix Database:
This window is used for create or modify matrix materials. The matrix is assumed to be isotropic. The "Nonlinear Coefficient" is the parameter $\alpha$ in equation (B2-1). The "Crack Length $c$" is the initial matrix crack length, which can be calculated using:

$$c = a \left[ 2 \sqrt{\frac{\pi}{2 \cdot 3V_f}} - 1 \right]$$ (B2-5)

where $a$ is the fiber diameter, $V_f$ is the fiber volume fraction.
Sample problem:
A sample of a 2D 2x2 triaxial braid composite is provided here to demonstrate all the input files. Those data files are given here in case that user misunderstands any parameters. Users are usually not required to view those data files.

Material:
The material of this composite is graphite AS4 yarns and 3501-6 epoxy resin. The Young’s modulus and shear modulus for matrix are 3448.0MPa and 1276.0MPa, respectively. Poisson’s ratio is 0.35, coefficient of thermal expansion is 4.0e-05/°C, the nonlinear coefficient is taken as 2.34, and the normal and shear strength for matrix is 84.85MPa and 98.30MPa, respectively. The initial crack length is 1.1e-05m, and the stress intensity factor is 0.2 MPa√m.

For yarn material, the moduli, Poisson’s ratio, and coefficient of thermal expansion are given as:

\[ E_{11} = 144.8e03\text{MPa}, E_{22} = E_{33} = 1.173e03\text{MPa}, G_{12} = G_{23} = G_{13} = 5.52e03\text{MPa} \]
\[ \mu_{12} = \mu_{13} = 0.23, \mu_{23} = 0.3, \alpha_1 = -3.24e-07/°C, \alpha_2 = \alpha_3 = 1.4e-05/°C \]

The yarn material strengths are:

\[ ST1T=0.014, ST1C=0.01, S22T=S33T=26.0\text{MPa}, S22C=S33C=206.0\text{MPa} \]
\[ S12S=102.4\text{MPa}, S23S=87.5\text{MPa} \]

The yarn fiber volume fraction is 0.75, fiber diameter is 7.0e-06m, and the shear stress parameter is 0.4. The two nonlinear parameters are equation to 1.0 and 10.0, respectively.

We have the mater.dat file as:

```plaintext
*** 
1,
1
1
3448.0,1276.0,0.35,4.0E-5, 0.23,0.3,0.23,-3.24E-07,1.4E-05, 1.0,10.0 
2,0.75,2, 
1
144800.0,11730.0,11730.0,0.5516,0.5516,0.5516,0.23,0.3,0.23,-3.24E-07,1.4E-5 
1,0,10,0, 0.014,0.01,26.0,206.0, 26.0,206.0,87.5,102.4,87.5 
0.4 
7.0E-6 
***
```
Geometry:
The composite is a single layer 2D, 2X2 triaxial braid. The layer thickness is 7.9e-04m, orientation is 0 degree, braid angle is 62.3 degree. Axial yarn spacing is 6.1e-03m, Number of fibers in axial yarn is 24K, number of fibers in braid yarn is 12K, percent of axial yarns is 0.37, and fiber diameter is 7.0e-06m.

We have the geom.dat file as:

```plaintext
***
1
2
0.0
1.79E-4,
62.3,
7.0E-6,0.0061,
24,
12,
0.37,
2,
2
***
```

Loading:
Assuming there is no temperature change, and applied stress in transverse direction as 600.0Mpa, and number of steps is 50. We have the load.dat file as:

```plaintext
***
0.0 0.0
0.0 600.0 0.0 0.0 0.0 0.0
50
***
```

Control:
Assuming the failure criterion is 10%, and using static analysis, fracture, and nonlinear model. There is no difference between plate model and solid mode for this case. We have the control.dat file as:

```plaintext
***
1,1,1,0,0.1,
***
```

Fatigue:
For fatigue analysis, The four parameters $c_1, c_2, b_1, b_2$, and the static strength $\sigma_s$ in equations (B2-2) to (B2-4) should be entered. For a C/SiC plain weave composite [3], the parameters in equation (B2-2) comes out as $-1.537e-02$ and $7.317e-05/MPa$, $b_1$ and $b_2$ in equations (B2-3) and (B2-4) come out as $4.147e-02$ and $-8.216e-05/MPa$, and static strength for (0/90) composite is 504.0MPa. we have the fatigue.dat file as:

```plaintext
***
-1.537E-02,7.317E-05,
-8.216e-05,0.04147,
504.0,
***
```
MicroTex Output

The results created by the above MicroTex execution are stored in three different files. The input and calculated geometry parameters are saved in `geomout.dat`, the overall stress-strain relation and stiffness reduction are saved in file `stress.dat`, and the initial mechanical and thermal parameters, the local stresses and strains of each material slice at each step, and the failure information are saved in file `output.sta`. For fatigue analysis, an additional file, `damage.dat`, is created for predicted S-N curve.

The geometry model, stress-strain relation curve, stiffness reduction curve, S-N curve, and `output.sta` file can also be displaced in the "view" window.

Sample problem:
The result files for the sample problem are listed here.

`geomout.dat`:

Geometry Parameter
for 2D 2X2 Triaxial Braided Composite

Input data:

- Braid angle: 62.300
- Number of fibers in axial yarn: 24000.000
- Number of fibers in braid yarn: 12000.000
- Percent of axial yarns: 0.370
- Thickness of each layer (m): 0.7900E-03
- Axial yarn spacing (m): 0.6100E-02
- Fiber diameter (m): 0.7000E-05

Calculated data:
- Axial yarn thickness (m): 0.5659E-03
- Braid yarn thickness (m): 0.1121E-03
- Length of axial yarn (m): 0.2562E-01
- Length of braid yarn (m): 0.1091E-01
- Volume of RUC (m^3): 0.1235E-06
- Fiber volume fraction: 0.52
- Crimp angle (deg): 6.58

`stress.dat`

\[
\begin{array}{cccccccccc}
\sigma_x & \varepsilon_x & E_x & \sigma_y & \varepsilon_y & E_y & \sigma_{xy} & \varepsilon_{xy} & G_{xy} \\
\hline
0.0000E+00, -0.5960E-04, 0.8138E+05, 0.3600E+02, 0.4588E-03, 0.7288E+05, 0.0000E+00, 0.4588E-03, 0.2273E+05, \\
0.0000E+00, -0.7960E-04, 0.7970E+05, 0.4800E+02, 0.6323E-03, 0.6918E+05, 0.0000E+00, 0.6323E-03, 0.2247E+05, \\
0.0000E+00, -0.9967E-04, 0.7818E+05, 0.6000E+02, 0.8137E-03, 0.7200E+05, 0.0000E+00, 0.8137E-03, 0.2207E+05, \\
0.0000E+00, -0.1198E-03, 0.7678E+05, 0.7200E+02, 0.1002E-02, 0.6599E+05, 0.0000E+00, 0.1002E-02, 0.2192E+05, \\
0.0000E+00, -0.1401E-03, 0.7547E+05, 0.8400E+02, 0.1198E-02, 0.6137E+05, 0.0000E+00, 0.1198E-02, 0.2192E+05.
\end{array}
\]
UNIT CELL OVERALL PROPERTIES

EXX = 0.43473E+05  EYY = 0.40531E+05  EZZ = 0.10209E+05   !moduli

NuXY = 0.25985  NuYX = 0.24227   !Poisson's ratio

NuXZ = 0.25196  NuYZ = 0.28887

GXY = 0.12217E+05  GYZ = 0.41796E+04  GXZ = 0.41292E+04   !shear moduli

Alfa1= 0.23804E-05  Alfa2= 0.24915E-05  Alfa3= 0.26612E-04  Alfa12=-0.32894E-05

steps, load, stress, strain and failure flag:

step = 1  load = 0.000E+00, 12.00, 0.00, 0.00, 0.00, 0.00,
layer number: 1
yarn slice number: 1

σ₁₁  σ₂₂  σ₃₃  τ₁₂  τ₁₃  τ₂₃  !local stress
-0.96934E+01, 0.32277E+01,-0.21554E+00, 0.89005E-16,-0.12821E-15,-0.89963E+00,
ε₁₁  ε₂₂  ε₃₃  ε₁₂  ε₁₃  ε₂₃  !local strain
-0.71728E-04, 0.29607E-03,-0.85526E-04, 0.16136E-19,-0.23244E-19,-0.16309E-03,
yarn slice number: 2
0.30429E+02, 0.60667E+00,-0.17046E+00, 0.17237E+01, 0.67227E+00,-0.65681E+00,
0.20945E-03, 0.77454E-05,-0.78381E-04, 0.31250E-03, 0.12188E-03,-0.11907E-03,

failure slice and model: 1, 1, 2  !failure location and model

step = 14  load = 0.000E+00, 168.00, 0.00, 0.00, 0.00, 0.00,
layer number: 1
yarn slice number: 1

-0.13670E+03, 0.34486E+02,-0.32706E+01,-0.26481E-14, 0.18309E-14,-0.12681E+02,
-0.10942E-02, 0.46035E-02,-0.13087E-02,-0.48008E-18, 0.33423E-18,-0.22990E-02,

damage.dat
Log(N)   E/Eo
0.5000E+00, 0.9910E+00,
0.6000E+00, 0.9892E+00,
0.7000E+00, 0.9874E+00,
0.8000E+00, 0.9856E+00,
0.9000E+00, 0.9839E+00,
0.1000E+01, 0.9821E+00,
0.1100E+01, 0.9804E+00,
0.1200E+01, 0.9786E+00,
0.1300E+01, 0.9769E+00,
0.1400E+01, 0.9752E+00,
0.1500E+01, 0.9734E+00,
0.1600E+01, 0.9717E+00,
0.1700E+01, 0.9700E+00,
0.1800E+01, 0.9683E+00,
0.1900E+01, 0.9666E+00,

References

System Requirement

ABAQUS-MicroTex can be run on PC or UNIX system. The system requirement for running ABAQUS-MicroTex is the same as the requirement for running ABAQUS software.

Since the GUI is written by JAVA, it also requires JDK (Java Development Kit 1.1.3 or higher version) on the PC or UNIX system.

Installation, Path and Execution

Installation:

The software is distributed in tar format file asm.tar. The file contains all the required data files, source files and java class files required to execute the programs. To install the software components copy change directory to the destination directory and extract the files using the command: tar -xvf asm.tar. This command creates the ASM main directory and sub directories to copy the CPFail, MicroTex and ABAQUS interface programs.
The ABAQUS-MicroTex is developed using Sun’s JDK 1.1.4. To execute these components Java runtime environment of version 1.1.3 or higher is required. Refer the JDK or JRE documentation for installation of Java.

Path:

After extracting the files, the following two path settings have to be completed.
1) The ABAQUS executable file with its path, xxx/xxx/xxx/abaqus, has to be set for GUI before running any sample problems. The path, xxx/xxx/xxx/, is the path where the ABAQUS software installed and abaqus is the ABAQUS executable file for UNIX version. This setting can be made by either manually editing the configuration file (abaqus.cfg) or by executing the program and modifying through the GUI using the menu command Configuration under Options menu.
2) In ABAQUS subroutine, UMAT, user need to set the full path in two ‘open’ statements. Modify the path ‘/xxx/xxx/ASM/TEX/ABAQUS-TEX/’ for ‘open(101,file= --- )’ and ‘open(102.file= --- )’. The UMAT subroutine is in the file jacobia.rut under the directory of xxx/xxx/ASM/TEX/ABAQUS-TEX/.

Execution:

The executable commands for ABAQUS-CPFail are listed below:

```
go to directory /ASM/TEX/ABAQUS-TEX
execute java asm.abaqus.ABApp TEXCAD command
```

Original code Files:

The following files are the original code files, user should carry over when change software installation directory, user can also carry over xxx.txc, xxx.ini and xxx.inp files for previous cases:

- initial.rut
- jacobia.rut
- /asm/abaqus/*.* (all files)
- abmicrotex.gif
- abaqus.env
- abaqus.cfg
- TXMatrix.db
- TXYarn.db
Finite Element Analysis of Textile Composite Structures

Introduction

As we mentioned in Part B1, the repeating unit cell (RUC) of a textile composite is usually used to represent the textile composite, and the thermal and mechanical properties of the RUC are considered as the same as the composite. The assumption behind this consideration is the same as the continuous assumption in Elasticity or other rational mechanics. In Elasticity, what we deal with is the material particle, the material continuity means that the particle is small enough that we can treat it as a mathematical point when we consider structures, and large enough that increasing the size will not affect the average material properties. For a textile composite, the concept of RUC is similar to the particle, it is so small that a textile composite structure can be considered as materially continuous, and so large it can fully represent the effect of textile composite microstructures.

It is clear that usually this argument is not true when we notice the dimensions of a RUC. The sizes of a RUC can easily reach to the magnitude of thousandth even hundredth of a meter. But the adopting of RUC concept may be the only way to perform stress analysis and failure prediction of textile composites without micro-level finite element analysis. Analyzing a real textile structures and considering all the woven and braided structures will be the last thing any engineer want to do, without even mention the matrix cracking or matrix-fiber interface slipping. On the other hand, the microstructures of textile composites do not have any repeating pattern in mathematical sense, the yarn paths, the distribution of matrix, even the cross-sections of yarns are far from regular.

The concept of RUC provides a reasonable approach for analyzing textile composite structures. The analysis in Part B1 is in fact a constitutive model of textile composite material. With this model the stress analysis of textile composite structures has no difference from traditional laminated composite structures but a different material. That means any finite element package with the capability of adoption to new materials can be employed to analyze textile composite structures. In this Part of this report, the combination of previous approach with ABAQUS is presented.

Two different material models, three dimensional solid model and two dimensional thin plate model, are developed in this Part for textile composite structures. Element tests are performed for both brick element and shell element and compared to the constitutive model. The failure analysis of a C/SiC plain weave composite plate with a circular hole located at the center is provided as a example.

The combining of MicroTex with ABAQUS: UMAT

As a commercial finite element analysis package, ABAQUS provides an extremely powerful and flexible tool for analysis by user subroutines. Since MicroTex only provides a material model to the finite element analysis, UMAT, the user subroutine to define material behavior, is the main subroutine we need to provide to ABAQUS. Another user subroutine, SDVINI, is also used to define the initial values of solution depended variables (SDV).

The UMAT subroutine retains part of the MicroTex's capabilities, such as the calculation of initial thermal and mechanical properties, material slices failure judgment, and the
calculation of stiffness reduction. The UMAT, in fact, calculates the Jacobean's matrix of the RUC at each stress states, the iteration scheme of ABAQUS is used to solve the system of nonlinear equations. The local stresses, strains, failure information, along with overall RUC material properties (Young's moduli and shear moduli), for each material slice are defined as solution dependent variables. Since the UMAT is called at each integral point on each iteration, all this information will be stored at each integral point, and can be output by the same way as ABAQUS standard variables. The capability of MicroTex to calculate geometry parameters for each material slice and material parameters for each matrix, yarn, and fiber, are provided using a separate code. The results of this code are sent to ABAQUS through input data files of UMAT.

Element test: Comparison with constitutive model

Generally, textile composites are three-dimensional architectures, and the constitutive relation developed in MicroTex is also three dimensional. The combining of MicroTex with ABAQUS through a 3D solid element is then straightforward. A single brick element model as shown in Figure B3-1 was analyzed to verify the UMAT code. As defined in Part B1, the material is C/SiC and the composite is plain weave, and the element type here is C3D8R which is a 3D 8-node brick element with reduced integral. An iterative scheme was used and the increment of applied force at each of the four nodes at the right end of the element is $P = 30.0 \times H$, where $H$ is the plate thickness. The comparison with MicroTex results for the two-layer (0/90) composite is given in Figure B3-2. The MicroTex analysis has a criterion for composite failure, so the stress-strain curve stops at the failure load, while for ABAQUS-MicroTex, there is no failure criteria, the predicted stress-strain curve using a single brick element model indicates a dramatic stiffness reduction. The maximum difference between MicroTex and ABAQUS model is less than 2%.

Usually, textile composites are used as plate or shell components which are difficult to model using 3D solid elements. Thus, the constitutive model has to be simplified to two dimensional. The 3D incremental stress-strain relation can be expressed as

$$
\begin{bmatrix}
\varepsilon_x \\
\varepsilon_y \\
\varepsilon_z \\
\varepsilon_{x1} \\
\varepsilon_{xy} \\
\varepsilon_{xz} \\
\varepsilon_{yz} \\
\end{bmatrix}
= 
\begin{bmatrix}
c_{xx} & c_{xy} & c_{xz} & 0 & 0 & 0 \\
c_{yx} & c_{yy} & c_{yz} & 0 & 0 & 0 \\
c_{zx} & c_{zy} & c_{zz} & 0 & 0 & 0 \\
0 & 0 & 0 & c_x & 0 & 0 \\
0 & 0 & 0 & 0 & c_y & 0 \\
0 & 0 & 0 & 0 & 0 & c_z \\
\end{bmatrix}
\begin{bmatrix}
\sigma_x \\
\sigma_y \\
\sigma_z \\
\tau_{xy} \\
\tau_{xz} \\
\tau_{yz} \\
\end{bmatrix}
$$

For thin plate or shell, assuming that $[\sigma_z, \tau_{xy}, \tau_{xz}] = 0$, we have

$$
\begin{bmatrix}
d\sigma_x \\
d\sigma_y \\
d\tau_{xy} \\
\end{bmatrix}
= 
\begin{bmatrix}
c_{xx} & c_{xy} & 0 \\
c_{yx} & c_{yy} & 0 \\
0 & 0 & c_{zz} \\
\end{bmatrix}^{-1}
\begin{bmatrix}
d\varepsilon_x \\
d\varepsilon_y \\
d\varepsilon_{xy} \\
\end{bmatrix}
$$
A single shell element model was analyzed for the same problem, the element type is S8R5. The results shown in figure B3-3 indicated that the UMAT for those two elements is correct.

**MicroTex-ABAQUS User Interface: GUI**

![MicroTex-ABAQUS GUI](image)

A graphical user interface was developed for finite element analysis using ABAQUS package. The user is required to provide finite element model, boundary conditions, and loading history through a standard ABAQUS input file as an initial file.

No blank space after comma in the material and section definition lines.

ABAQUS allows different material in single model. The user should define textile composite materials (assuming MicroTex as their constitutive model) through the GUI and all other materials as standard ABAQUS input.

The ABAQUS-MicroTex GUI provides windows for user to input yarn and matrix material properties, such as moduli, Poisson's ratio, coefficients of thermal expansion, maximum strains and stresses, nonlinear coefficients, stress intensity factor, initial crack length, shear stress parameters, and yarn packing density; geometry parameters, such as yarn orientation, layer thickness, axial yarn spacing, axial yarn content, and yarn sizes. of textile composites; and analytical control parameters, such as linear or nonlinear materials, plate theory or 3D iso-strain analysis, fracture model or continuum model. The GUI also provides user opportunity to modify orientations of each composite layers. Those input windows are very much similar to the MicroTex GUI. The main difference is that the user is required to specify element groups to which all material and geometry properties are defined.

The GUI will first generate a set of data files containing material, geometry and control data, and will allow the user to choice between creating a new ABAQUS input file or
overwriting the original one. Before executing ABAQUS, the GUI will execute a FORTRAN code based on the input material and geometry parameters to generate two data files for UMAT for each textile composite material.

Following is the ABAQUS input file for the shell element testing problem. It is noticed that element group "allel" is made up of material "shell" which is not defined (no modulus or Poisson's ratio is given). ABAQUS-MicroTex reserves material names with the first five letters being "shell" to materials defined by UMAT with the 2D stress-strain relation, and all the other user-defined material names as defined by UMAT with the 3D stress-strain relation. There is neither indication in the input file that "shell" is a user defined material, nor the location of UMAT. The user is required to define this material with GUI by specifying material and geometry parameters for this material under element group "allel".

*heading
Textile composite plate, single shell elements model
S8R5 element
SI units
**

** Model definition **

*node, nset=alln
101, 0.00, 0.00, 0.00
102, 0.25e+03, 0.00, 0.00
103, 0.50e+03, 0.00, 0.00
201, 0.00, 0.25e+03, 0.00
202, 0.50e+03, 0.25e+03, 0.00
301, 0.00, 0.50e+03, 0.00
302, 0.25e+03, 0.50e+03, 0.00
303, 0.50e+03, 0.50e+03, 0.00

*nset, nset=middle
201, 202

*nset, nset=left
101, 201, 301

*element, type=s8r5, elset=allel
1, 101, 103, 303, 301, 102, 202, 302, 201

*shell selection, elset=allel, composite
6.0e-04, 1, shell, np0
6.0e-04, 1, shell, p90

*transverse shear stiffness
5.0e04, 5.0e04, 5.0e04

*material, name=shell

!!shell material is used for element group allel

!!layer orientation is defined (p90), and can be modified in GUI

*material, name=shell

!! shell is the material to be defined using UMAT

*elastic
1.0,2.0,3.0,4.0,5.0,6.0

*orientation, name=np0, system=rectangular
1.0, 0.0, 0.0, 0.0, 1.0, 0.0
3, 00.0

*orientation, name=p90, system=rectangular
1.0, 0.0, 0.0, 0.0, 1.0, 0.0
3, 90.0

**
Finite Element Analysis of Textile Composite Structures

Introduction

As we mentioned in Part B1, the repeating unit cell (RUC) of a textile composite is usually used to represent the textile composite, and the thermal and mechanical properties of the RUC are considered as the same as the composite. The assumption behind this consideration is the same as the continuous assumption in Elasticity or other rational mechanics. In Elasticity, what we deal with is the material particle, the material continuity means that the particle is small enough that we can treat it as a mathematical point when we consider structures, and large enough that increasing the size will not affect the average material properties. For a textile composite, the concept of RUC is similar to the particle, it is so small that a textile composite structure can be considered as materially continuous, and so large it can fully represent the effect of textile composite microstructures.

It is clear that usually this argument is not true when we notice the dimensions of a RUC. The sizes of a RUC can easily reach to the magnitude of thousandth even hundredth of a meter. But the adopting of RUC concept may be the only way to perform stress analysis and failure prediction of textile composites without micro-level finite element analysis. Analyzing a real textile structures and considering all the woven and braided structures will be the last thing any engineer want to do, without even mention the matrix cracking or matrix-fiber interface slipping. On the other hand, the microstructures of textile composites do not have any repeating pattern in mathematical sense, the yarn paths, the distribution of matrix, even the cross-sections of yarns are far from regular.

The concept of RUC provides a reasonable approach for analyzing textile composite structures. The analysis in Part B1 is in fact a constitutive model of textile composite material. With this model the stress analysis of textile composite structures has no difference from traditional laminated composite structures but a different material. That means any finite element package with the capability of adoption to new materials can be employed to analyze textile composite structures. In this Part of this report, the combination of previous approach with ABAQUS is presented.

Two different material models, three dimensional solid model and two dimensional thin plate model, are developed in this Part for textile composite structures. Element tests are performed for both brick element and shell element and compared to the constitutive model. The failure analysis of a C/SiC plain weave composite plate with a circular hole located at the center is provided as a example.

The combining of MicroTex with ABAQUS: UMAT

As a commercial finite element analysis package, ABAQUS provides an extremely powerful and flexible tool for analysis by user subroutines. Since MicroTex only provides a material model to the finite element analysis, UMAT, the user subroutine to define material behavior, is the main subroutine we need to provide to ABAQUS. Another user subroutine, SDVINI, is also used to define the initial values of solution depended variables (SDV).

The UMAT subroutine retains part of the MicroTex's capabilities, such as the calculation of initial thermal and mechanical properties, material slices failure judgment, and the
** analysis history  **
*step
*static
0.05, 1.0
*boundary
left, 1
left, 3, 6
middle, 2
*load
103, 1, 2.5
202, 1, 10.0
303, 1, 2.5
**
** output  **
*restart, write
*node print, nset=alln, frequency=1
u
*node file, nset=alln, frequency=1
u
*el print, elset=allel, frequency=1
s,e
*el file, elset=allel, frequency=1
s,e
*end step

The GUI can save all the GUI input information in a file, xxx.txc, for re-opening. A new ABAQUS input file will be created by GUI. This new file will be the same as the original one for the model, boundary condition and loading history, output requirement, the only difference is the material property part. The new one will appear as follows. The solution dependent variables are defined at each integral point and updated in each iteration. The first six solution dependent variables here are used to store the local moduli, and can be exported as standard ABAQUS variables.

......
......
*shell selection, elset=allel, composite
6.0e-04, 1, shell, np0
6.0e-04, 1, shell, p90
......
......
*material, name=shell
*depvar
822
*user material, constant=0
*initial conditions, type=solution, user
*user subroutine, input=...\...\initial.rut
*user subroutine, input=...\...\jacobiania.rut
......
......

!!number of solution variables
!!initial data for UMAT
!!UMAT, define material
Finite element analysis of a C/SiC plain weave composite plate with a center hole

A two-layer (0/90) C/SiC plain weave composite plate with a hole at the center has been analyzed using ABAQUS. The plate is 8 inches long and 1 inch wide, the radius of the hole is 0.125 inches. A total of 768 shell elements (S4R) and 844 nodes were used to model the plate, and 3 integral points were used for each layer; the mesh which was generated by PATRAN is shown in Figure B3-4. The load is applied through uniform displacement boundary condition at two ends. In each increment, the displacement increases 5.0e-04 m.

Figures B3-5 to B3-9 give the distribution of normal stress and tangent stiffness moduli in loading direction at increments 5, 7, 8, and 10, respectively. The moduli are all normalized by the initial material properties of the RUC. When any material slice fails, the corresponding modulus is reduced to 1%.

The distributions of moduli clearly indicate the failure process of the plate on the element base. It can be seen that, at increment 5, the maximum stress is less than 490.0MPa, the modulus indicates that there is no element failed although the nonlinear material properties cause some stiffness reductions. At increment 7, the distribution of modulus indicates that element failure happens near the hole. The failure range increases at increment 8, and at increment 10, the plate failed in the sense that the modulus in loading direction across the width of the plate had been reduced near to 1%.

Figure B3-1. Brick element model for (0/90) C/SiC plain weave composite
Figure B3-2. Comparison for a 2-layer (0/90) C/SiC plain weave composite: Brick element

Figure B3-3. Comparison for a (0/90) C/SiC plain weave composite: Shell elements
Figure B3-4. Finite element model for a two layer plane weave composite plate with a hole.

Figure B3-5. Distribution of normal stress and tangent modulus at increment 5.
Figure B3-6. Distribution of normal stress and tangent modulus at increment 7.

Figure B3-7. Distribution of normal stress and tangent modulus at increment 8.
Figure B3-8. Distribution of normal stress and tangent modulus at increment 10.
Part C. Experiments

Introduction

An experimental investigation was performed to determine the fatigue and static behavior of two lay-ups of Carbon/Silicon Carbide (C/SiC) composite specimens. The specimens were loaded in static tension and in tension-tension fatigue under isothermal fixed-frequency conditions. Two temperature levels and a range of stress levels for fatigue were used to assess performance. The tests were conducted at 23°C and 1000°C. Measurements were made of modulus and cycles to failure several times during the test.

Experimental Program

The experimental program was divided into two parts to develop an understanding of the fatigue performance of the material. The first part consisted of an investigation to determine the static properties of the material and determine how the modulus of the material changes as a function of maximum applied static strain. The second portion of the testing consisted of an investigation of the high cycle fatigue behavior of the material. The specimens were tested to failure or 1,000,000 cycles.

Tests were performed on two lay-ups of a C/SiC composite material system with a CVIP coating by DuPont. The coating provided protection from oxidation at high temperature. This composite system consisted of plain woven carbon fibers in a silicon carbide matrix produced in two 24 ply symmetric lay-ups. The test panels, produced with a [0/90] and a [0/±60] lay-up, both had a nominal thickness of 0.12 in. after manufacture. The panels were machined into 6 inch "dog-bone" specimens with a 2-inch long by approximately 0.50 inch wide gage section.

Test Procedure and Equipment

Static Properties

The static test used on the C/SiC was performed in load control at a loading rate of 5000 lb./min. on a 50 KIP servo-hydraulic test stand equipped with an automated data acquisition system. The load was increased by 150 lb, then decreased by 100 lb, in order to take the stiffness measurements. This loading series was repeated until the specimen failed. This loading and unloading provided the advantage of allowing the calculation of the modulus as a function of the maximum strain experienced by the specimen for each unloading segment during the test. A 25.4 mm (1 in.) gauge length extensometer was mounted on the edges and centered with respect to the gage section to provide strain data. Load and strain were recorded at half-second intervals during all of the tests. Property calculations consisted of ultimate strength and modulus. Ultimate stress was calculated using the area of the un-notched cross-section and the highest load carried by the laminate. Modulus was calculated using a least squares fit to the unloading segments of data during static tests.
Fatigue Properties

The tension fatigue tests were performed on a 50 KIP capacity servo-hydraulic test stand. Load, strain, actuator displacement and time were recorded using an automated digital data acquisition system. All fatigue tests were performed under load control at a frequency of 5 Hz (except during data cycles) with a min./max. stress ratio of R=0.1 and a sinusoidal wave form. Tests were concluded after 1,000,000 cycles if failure did not occur.

The strain and load were measured during specified cycles of the fatigue test as summarized in Table C-1 and allowed for determination of changes in modulus. For high temperature testing, a clamshell furnace maintained the test temperature over the entire gage length of the specimen. A high temperature ceramic extension rod extensometer with a 25.4 mm (1.0 in.) gage length was edge mounted on the gage section center to measure strain.

Applied load and measured strain were used to calculate an effective modulus for the gage section during fatigue tests. Stiffness calculations were performed by making a least squares linear fit to the stress versus strain data. Stress was calculated using the specimen gage section area and the load from the data acquisition system. Strain was taken from the extensometer attached to the gage section. The fit was performed on the loading and unloading data over the entire fatigue cycle.

Experimental Results

Static Tests

The static tests at 23°C and 1000°C were conducted for both lay-ups. The ultimate stress for each testing condition is shown in Table C-2. There was a 57% reduction in strength of the [0/90] lay-up between room temperature and 1000°C while the [0/+60] lay-up experienced a 35% reduction in strength. Results from the room temperature static tests are shown in Figure C-1 for both lay-ups. Elastic modulus as a function of maximum strain for these two static tests is shown in Figure C-2.

Fatigue Tests

The 23°C fatigue tests are summarized in Tables C-3 and C-4, and Figures C-3 and C-4. Tables C-3 and C-4 display the maximum stress and cycles to failure of individual [0/90] and [0/+60] specimens respectively during fatigue. The fatigue life diagrams for each material lay-up are shown in Figures C-3 and C-4. These figures allow visualization of both the fatigue data and the static results.

Similarly the 1000°C fatigue tests are summarized in Tables C-5 and C-6, and Figures C-5 and C-6. The fatigue data is tabulated in Tables C-5 and C-6 for [0, 90] and [0/+60] specimens respectively. The fatigue life diagrams for each material lay-up are shown in Figures 5 and 6. Notice that none of the fatigue tests lasted to run out.

The modulus is shown in Figures C-7 and C-8 as a function of cycle count during fatigue. Data for the [0/90] lay-up is shown in Figure C-7. Data for the [0/+60] is displayed in Figure C-8. These figures demonstrate how the modulus of both lay-ups decreases with fatigue loading.
Table C-1. Ultimate Stress for the C/SiC Static Tests

<table>
<thead>
<tr>
<th>Cycle Count Range</th>
<th>Reading Interval</th>
</tr>
</thead>
<tbody>
<tr>
<td>1-10</td>
<td>1 cycle</td>
</tr>
<tr>
<td>10-50</td>
<td>5 cycles</td>
</tr>
<tr>
<td>50-100</td>
<td>10 cycles</td>
</tr>
<tr>
<td>100-500</td>
<td>1,990 cycles</td>
</tr>
<tr>
<td>500-1,000</td>
<td>100 cycles</td>
</tr>
<tr>
<td>1,000-5,000</td>
<td>500 cycles</td>
</tr>
<tr>
<td>5,000-10,000</td>
<td>1,000 cycles</td>
</tr>
<tr>
<td>10,000-100,000</td>
<td>5,000 cycles</td>
</tr>
<tr>
<td>100,000-1,000,000</td>
<td>10,000 cycles</td>
</tr>
</tbody>
</table>

Table C-2. Ultimate Stress for the C/SiC Static Tests

<table>
<thead>
<tr>
<th>Lay-up</th>
<th>Temperature (°C)</th>
<th>Ultimate Stress (ksi)</th>
</tr>
</thead>
<tbody>
<tr>
<td>[0/90]</td>
<td>23</td>
<td>73.0</td>
</tr>
<tr>
<td>[0/90]</td>
<td>1000</td>
<td>31.3</td>
</tr>
<tr>
<td>[0/±60]</td>
<td>23</td>
<td>42.0</td>
</tr>
<tr>
<td>rdl[0/±60]</td>
<td>1000</td>
<td>27.1</td>
</tr>
</tbody>
</table>

Table C-3. Cycles to failure for the [0/90] C/SiC Fatigue Tests at 23°C  
(R = 0.1, f = 5Hz.)

<table>
<thead>
<tr>
<th>Maximum Stress</th>
<th>Cycles to Failure</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\sigma_{\text{max}}$ (ksi)</td>
<td>n</td>
</tr>
<tr>
<td>66.5</td>
<td>4369</td>
</tr>
<tr>
<td>62.7</td>
<td>37332</td>
</tr>
<tr>
<td>59.5</td>
<td>42334</td>
</tr>
<tr>
<td>56.0</td>
<td>515176</td>
</tr>
<tr>
<td>45.5</td>
<td>&gt; 1000000</td>
</tr>
</tbody>
</table>

Table C-4. Cycles to failure for the [0/±60] C/SiC Fatigue Tests at 23°C  
(R = 0.1, f = 5Hz.)

<table>
<thead>
<tr>
<th>Maximum Stress</th>
<th>Cycles to Failure</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\sigma_{\text{max}}$ (ksi)</td>
<td>n</td>
</tr>
<tr>
<td>40.53</td>
<td>423</td>
</tr>
<tr>
<td>39.9</td>
<td>2924</td>
</tr>
<tr>
<td>37.7</td>
<td>48754</td>
</tr>
<tr>
<td>36.0</td>
<td>546514</td>
</tr>
<tr>
<td>35.6</td>
<td>&gt; 1000000</td>
</tr>
</tbody>
</table>
Table C-5. Cycles to failure for the [0/90] C/SiC Fatigue Tests at 1000°C
(R = 0.1, f = 5Hz.)

<table>
<thead>
<tr>
<th>Maximum Stress $\sigma_{max}$ (ksi)</th>
<th>Cycles to Failure $n$</th>
</tr>
</thead>
<tbody>
<tr>
<td>28.0</td>
<td>1853</td>
</tr>
<tr>
<td>25.1</td>
<td>5417</td>
</tr>
<tr>
<td>21.7</td>
<td>8028</td>
</tr>
<tr>
<td>15.4</td>
<td>37751</td>
</tr>
<tr>
<td>rdrs9.1</td>
<td>95525</td>
</tr>
</tbody>
</table>

Table C-6. Cycles to failure for the [0/±60] C/SiC Fatigue Tests at 1000°C
(R = 0.1, f = 5Hz.)

<table>
<thead>
<tr>
<th>Maximum Stress $\sigma_{max}$ (ksi)</th>
<th>Cycles to Failure $n$</th>
</tr>
</thead>
<tbody>
<tr>
<td>21.8</td>
<td>6060</td>
</tr>
<tr>
<td>19.2</td>
<td>7861</td>
</tr>
<tr>
<td>16.2</td>
<td>33829</td>
</tr>
<tr>
<td>13.5</td>
<td>55309</td>
</tr>
<tr>
<td>10.8</td>
<td>114676</td>
</tr>
</tbody>
</table>
Fig. C-1 Static Results for C/SiC at 23°C

Fig. C-2 Modulus Shown as a Function of Maximum Strain for C/SiC Tests at 23°C
Fig. C-3 Fatigue Life Diagram for C/SiC at 23°C

Fig. C-4 Fatigue Life Diagram for C/SiC at 23°C
Fig. C-5  Fatigue Life Diagram for C/SiC at 1000°C

Fig. C-6  Fatigue Life Diagram for C/SiC at 1000°C
Fig. C-7 Modulus as a Function of Cycle Count Diagram for C/SiC at 23°C

Fig. C-8 Modulus as a Function of Cycle Count Diagram for C/SiC at 23°C
Part D. Publications

Two paper related to this project were published in technical conferences.


Part E. Commercialization Plan

MICROTEX and CPFail developed by Analytical Services and Material, Inc. are the user-friendly softwares which combined the degradation analysis of composite materials (textile and laminate) into ABAQUS structural analysis. The softwares have the capability to perform structural progressive failure analysis for laminate and textile composite structures.

Progressive failure analysis is very important to the structure safety, especially for aging aircraft structures. Comparing to the conventional structural analysis, the progressive failure analysis can more accurately predict the structural damage development and the structure life.

Many composite failure analysis theories have been developed, but not much of them actually been used in real structural analysis. This is due to either the theory is not applicable (theoretical research) or not available for commercial or practical users. MICROTEX and CPFail were developed under the consideration of practical application. They are combined with ABAQUS commercial structural analysis code, therefore they are available for the general structural analysis.

The following suggestions are for Phase III or commercialization MICROTEX and CPFail:

1) Provide free maintenance service to NASA Marshall and any buyer
2) Continuously update MICROTEX and CPFail, distribute updated version to current users
3) Provide consulting and training service for using MICROTEX and CPFail
4) Advertise MICROTEX and CPFail in technical conferences, seminars, internet, universities and research institutes.
5) Apply MICROTEX or CPFail technology to other projects, get the support from new projects to update MICROTEX and CPFail.
6) Find potential new users, keep in touch with the current users
7) Give free simple sample version to new user to test
8) Combine MICROTEX and CPFail to other commercial codes, for example, NASTRAN, COMET, ----.

The key point of Phase III is to find potential users. Composite structure manufacture company and airplane, automobile companies are the possible potential users.
Part F. Computer Codes

The software codes for CPFail, MicroTex, ABAQUS-CPF and ABAQUS-TEX are distributed in tar format file asm.tar. The file contains all the required data files, source files and java class files required to execute the programs. To install the software components copy change directory to the destination directory and extract the files using the command: tar -xvf asm.tar. This command creates the ASM main directory and sub directories to copy the CPFail, MicroTex and ABAQUS interface programs. After extracting the files the default executable file names and paths have to set before running the any sample problems. This can be set either manually editing the configuration files or by executing the programs and modifying through the GUI using the menu command Configuration under Options menu. The executable commands for software components are listed in the sections 'Installation and Execution' of each manual.

The CPFail is developed in C/C++ using X-Windows and Motif toolkits. Where as other components are developed using Sun’s JDK 1.1.4. To execute these components Java runtime environment of version 1.1.3 or higher is required. Refer the JDK or JRE documentation for installation of Java.

All the software codes currently have been installed in merlin.msfc.nasa.gov under the directory /xfs/merlin/lv00/home/xuedx:

1. /xfs/merlin/lv00/home/xuedx/ASM/CPF/ABAQUS-CPF
   Execution: java asm.abaqus.ABApp CMLIFE

2. /xfs/merlin/lv00/home/xuedx/ASM/CPF/CPFAIL
   Execution: ./cpfail

3. /xfs/merlin/lv00/home/xuedx/ASM/TEX/ABAQUS-TEX
   Execution: java asm.abaqus.ABApp TEXCAD

4. /xfs/merlin/lv00/home/xuedx/ASM/TEX/MICROTEx
   Execution: java asm.microtex.TXApp
**Progressive Failure and Life Prediction of Ceramic and Textile Composites**

**AUTHOR(S)**
David Y. Xue, Yucheng Shi, William M. Johnston, Madhu Katikala and Micheal P. Card

**PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)**
AS&M, Inc.
107 Research Drive
Hampton, VA 23666

**SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES)**
NASA
Marshall Space Flight Center
Huntsville, AL

**ABSTRACT (Maximum 200 words)**
An engineering approach to predict the fatigue life and progressive failure of multilayered composite and textile laminates is presented. Analytical models which account for matrix cracking, statistical fiber failures and nonlinear stress-strain behavior have been developed for both composites and textiles. The analysis methods use experimentally derived empirical coefficients to account for the interface of fiber and matrix, fiber strength, and fiber-matrix stiffness reductions. Similar considerations were applied to textiles using Repeating Unit Cells. Fatigue analyses included use of Walker's equation for cyclic matrix cracking and Heywood's formulation for stress degradation. The analysis has been compared with experiment with good agreement. Comparisons were made with Graphite-Epoxy, C/SiC and Nicalon/CAS composite materials. For textile materials, comparisons were made with tubular braided and plain weave materials under biaxial or uniaxial tension. Fatigue predictions were compared with test data obtained from plain weave C/SiC materials tested at AS&M. Several computer codes were developed to perform the analysis. Composite Progressive Failure Analysis for Laminates is contained in the code CPFail. Micromechanics Analysis for Textile Composites is contained in the code MicroTex. Both codes were adapted to run as subroutines for the nonlinear Finite Element code ABAQUS and MicroText-ABAQUS.

**SUBJECT TERMS**
Ceramics, Textiles, Progressive, failure