Trade names or manufacturers' names are used in this report for identification only. This usage does not constitute an official endorsement, either expressed or implied, by the National Aeronautics and Space Administration.
Micromechanics Analysis Code (MAC)
User Guide: Version 2.0

Developed by:

T. E. Wilt
The University of Akron
Department of Civil Engineering

S. M. Arnold
National Aeronautics and Space Administration
Lewis Research Center
1.0 Introduction

The ability to accurately predict the thermomechanical deformation response of advanced composite materials continues to play an important role in the development of these strategic materials. Analytical models that predict the effective behavior of composites are used not only by engineers performing structural analysis of large-scale composite components but also by material scientists in developing new material systems. For an analytical model to fulfill these two distinct functions it must be based on a micromechanics approach which utilizes physically based deformation and life constitutive models and allows one to generate the average (macro) response of a composite material given the properties of the individual constituents and their geometric arrangement. Only then can such a model be used by a material scientist to investigate the effect of different deformation mechanisms on the overall response of the composite in order to identify the appropriate constituents for a given application. However, if a micromechanical model is to be used in a large-scale structural analysis it must be 1) computationally efficient, 2) able to generate accurate displacement and stress fields at both the macro and the micro level and 3) be compatible with the finite element method. Additionally, new advancements in processing and fabrication techniques now make it possible to engineer the architectures of these advanced composite systems. Full utilization of these emerging manufacturing capabilities require the development of a computationally efficient micromechanics analysis tool capable of accurately predicting the effect of microstructural details on the internal and macroscopic behavior of composites. The above mentioned computational efficiency is required since 1) the large number of parameters that must be varied in the course of engineering (or designing) composite materials, and 2) the optimization of a material's microstructure will require the integration of the micromechanics model with optimization algorithms. From this perspective, analytical approaches that produce closed form expressions which describe the effect of a material's internal architecture on the overall material behavior are preferable to numerical methods such as the finite element or finite difference schemes.

A number of models presently exist that can fulfill some aspect of the aforementioned tasks. However, there are very few working models that are both computationally efficient and sufficiently accurate at the micro- as well as the macro-level. One such micromechanics model with the potential of fulfilling both tasks is the method of cells [1] and its generalization [2]. The comprehensive capabilities and efficiency of this method has been documented in references [4] and [5]. Consequently, the recently developed, computationally efficient and comprehensive micromechanics analysis code, MAC, who's predictive capability rests entirely upon the fully analytical micromechanics model, herein referred to as the generalized method of cells, GMC, [2 and 3] will now be described. MAC is a versatile form of research software that "drives" the double or triply periodic micromechanics constitutive models based upon GMC. GMC is capable of predicting the response of both continuous and discontinuous multi-phased composites with an arbitrary internal microstructure and reinforcement shape. GMC is a continuum based micromechanics model that provides closed-form expressions for the mac-
roscopic composite response in terms of the properties, size, shape, distribution, and response of the individual constituents or phases that make up the material. GMC also utilizes physically based viscoplastic deformation and life models for each constituent. Furthermore, expressions relating the internal stress and strain fields in the individual constituents in terms of the macroscopically applied stresses and strains are also available through strain or stress concentration factors. These expressions make possible the investigation of failure processes at the microscopic level at each step of an applied load history. Similarly, GMC provides the capability of studying the influence of bond strength at the fiber/matrix interface which recently has been shown to be an important damage mechanism.

**MAC** enhances the basic capabilities of GMC by providing a modular framework wherein 1) various thermal, mechanical (stress or strain control) and thermomechanical load histories can be imposed, 2) different integration algorithms may be selected, 3) a variety of constituent constitutive models may be utilized and/or implemented and 4) a variety of fiber architectures may be easily accessed through their corresponding representative volume elements. Figure 1 illustrates the basic flow diagram for this modular framework. The capabilities for this version of **MAC** are discussed in section 2, whereas theoretical and background information on the basic capabilities itemized above is given in section 3. Section 4 describes how one might use **MAC** and will be the most referred to section in the entire manual. Finally, section 5 gives some insights into the future modifications planned for **MAC**.
Input Phase

Load History Data

Material Model and Constituent Data

Time Increment Loop

Load History Control: Thermal, Mechanical or Thermomechanical

Integration Algorithm: Forward Euler or Predictor/Corrector

Output Data: X-Y Data and PATRAN

Load History Completed?

No

Yes

Stop

GMC: Double or Triple Periodic

Constituent Models: Bodner-Partom Robinson GVIPS

Figure 1: MAC Flowchart
2.0 Current Capabilities

In this section the current features/capabilities of MAC are itemized.

- **Load Types:**
  - Thermal
  - Mechanical
  - Thermomechanical

- **Aboudi GMC Models:**
  - Double Periodicity Model for continuous reinforcement
  - Triple Periodicity Model for discontinuous reinforcement

- **Graphical Output**
  - Up to 5 x-y data plot files may be generated for both macro and micro (sub-cell) quantities
  - PATRAN for subcell geometry and color results evaluation, e.g. stress, strains, inelastic strains, J2, etc.

- **Integration Options:**
  - Forward Euler
  - Predictor/Corrector

- **Constitutive Models**
  - Elastic orthotropic
  - Inelastic viscoplastic models:
    - Bodner-Partom
    - Robinson
    - GVIPS
    - User defined model using subroutine USRMAT

- **Fiber/Matrix Interface Layer**

- **Laminate Analysis**

- **Fiber/Matrix Debonding Conditions**

- **Free Format Input Data**
3.0 Background

3.1 Micromechanics Models

As stated in the introduction, MAC's predictive capabilities rest entirely upon the fully analytical micromechanics model known as GMC which is capable of predicting the inelastic response of both continuous (double periodicity) and discontinuous (triple periodicity) multi-phased composites with an arbitrary internal microstructure and reinforcement shape. Prior to describing the available architectures (Representative Volume Elements, RVE's) within MAC as discussed in section 4, a brief overview of the theoretical foundation behind the generalized method of cells follows. A more complete discussion of the theoretical formulation is given in [1] - [3].

In the original formulation of the method of cells, a continuously (or discontinuously) reinforced, unidirectional fibrous composite is modeled as a rectangular, double-periodic (or triply-periodic) array of fibers embedded in a matrix phase. The periodic character of the assemblage allows one to identify a repeating unit cell that can be used as a building block to construct the entire composite. The properties of the repeating cell are thus representative of the properties of the entire assemblage. The unit cell consists of a single fiber subcell surrounded by three matrix subcells for continuous and seven for discontinuous composites, hence the name method of cells. The rectangular geometry of the repeating unit cell allows one to obtain an approximate solution for the stresses and strains in the individual subcells given some macroscopically homogeneous state of strain or stress applied to the composite. The approximate solution to the posed boundary value problem is, in turn, used to determine macroscopic (average) or effective properties of the composite and the effective stress-strain response in the inelastic region.

In the generalized method of cells for continuous (or discontinuous) fibrous composites, the repeating unit cell can consist of an arbitrary number of phases. Hence the generalized method of cells is capable of modeling a multiphase composite. This generalization extends the modeling capability of the original method of cells to include the following: 1) inelastic thermomechanical response of multiphased metal matrix composite, 2) modeling of various fiber architectures (including both shape and packing arrangements), 3) modeling of porosities and damage, and 4) the modeling of interfacial regions around inclusions including interfacial degradation.

The basic homogenization approach taken in the micromechanical analysis consists essentially of four steps. First, the repeating volume element, RVE, of the periodic composite is identified. Second, the macroscopic or average stress and strain state in terms of the individual microscopic (subcell) stress and strain states is defined. Third, the continuity of tractions and displacements are imposed at the boundaries between the constituents. These three steps, in conjunction with
micro-equilibrium, establish the relationship between micro (subcell) total, thermal and inelastic strains and macro (composite) strains via the relevant concentration tensors. In the fourth and final step, the overall macro constitutive equations of the composite are determined. These four steps form the basis of the micro-to-macro-mechanics analysis which describe the behavior of heterogeneous media. The resulting micromechanical analysis establishes the overall (macro) behavior of the multi-phase composite and is expressed as a constitutive relation between the average stress, strain, thermal, and inelastic strains, in conjunction with the effective elastic stiffness tensor.

That is,

$$\bar{\varepsilon} = B^* (\bar{\varepsilon} - \bar{\varepsilon}^I - \bar{\varepsilon}^T) \quad \text{(EQ 1)}$$

where for the most general case of discontinuous reinforcement with \(N_a\) by \(N_p\) by \(N_\gamma\) number of subcells, the effective elastic stiffness tensor, \(B^*\), of the composite is given by,

$$B^* = \frac{1}{dhl} \sum_{a=1}^{N_a} \sum_{p=1}^{N_p} \sum_{\gamma=1}^{N_\gamma} d_a h_p l_\gamma \zeta^{(\alpha\beta\gamma)} \overline{A}^{(\alpha\beta\gamma)} \quad \text{(EQ 2)}$$

the composite inelastic strain tensor is defined as,

$$\bar{\varepsilon}^I = -B^* - 1 \frac{N_a}{dhl} \sum_{a=1}^{N_a} \sum_{p=1}^{N_p} \sum_{\gamma=1}^{N_\gamma} d_a h_p l_\gamma \zeta^{(\alpha\beta\gamma)} (D^{(\alpha\beta\gamma)}) \bar{\varepsilon}^I - \zeta^{(\alpha\beta\gamma)} \quad \text{(EQ 3)}$$

the average thermal strain tensor as,

$$\bar{\varepsilon}^T = -B^* - 1 \frac{N_a}{dhl} \sum_{a=1}^{N_a} \sum_{p=1}^{N_p} \sum_{\gamma=1}^{N_\gamma} d_a h_p l_\gamma \zeta^{(\alpha\beta\gamma)} (D^{(\alpha\beta\gamma)}) \bar{\varepsilon}^T - \zeta^{(\alpha\beta\gamma)} \quad \text{(EQ 4)}$$

and \(\bar{\varepsilon}\) is the uniform applied macro (composite) strain. For the case of continuous reinforcements with \(N_p\) by \(N_\gamma\) number of subcells, eq. (2) - (4) reduce to the following:

$$B^* = \frac{1}{h_l} \sum_{p=1}^{N_p} \sum_{\gamma=1}^{N_\gamma} h_p l_\gamma \zeta^{(\beta\gamma)} \overline{A}^{(\beta\gamma)} \quad \text{(EQ 5)}$$

$$\bar{\varepsilon}^I = \frac{-B^* - 1}{h_l} \sum_{p=1}^{N_p} \sum_{\gamma=1}^{N_\gamma} h_p l_\gamma \zeta^{(\beta\gamma)} (D^{(\beta\gamma)}) \bar{\varepsilon}^I - \zeta^{(\beta\gamma)} \quad \text{(EQ 6)}$$

$$\bar{\varepsilon}^T = \frac{-B^* - 1}{h_l} \sum_{p=1}^{N_p} \sum_{\gamma=1}^{N_\gamma} h_p l_\gamma \zeta^{(\beta\gamma)} (D^{(\beta\gamma)}) \bar{\varepsilon}^T - \zeta^{(\beta\gamma)} \quad \text{(EQ 7)}$$
In the above equations matrix notation is employed; where, for example, the average stress, $\bar{\sigma}$, average applied strain, $\bar{\varepsilon}$, and inelastic subcell strain, $\varepsilon_s$, vectors represent,

$$\bar{\sigma} = \{\sigma_{11}, \sigma_{22}, \sigma_{33}, \sigma_{12}, \sigma_{23}, \sigma_{13}\} \quad \text{(EQ 8)}$$

$$\bar{\varepsilon} = \{\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{33}, \varepsilon_{12}, \varepsilon_{23}, \varepsilon_{13}\} \quad \text{(EQ 9)}$$

$$\varepsilon^T_s = \{\varepsilon^T_s(111), ..., \varepsilon^T_s(N_\alpha N_\beta N_\gamma)\} \quad \text{(EQ 10)}$$

where the six components of the vector $\varepsilon^T_s(\alpha\beta\gamma)$ are arranged as in eq. (9). Similar definitions for $\varepsilon_s^T$, $\varepsilon_s^T(\alpha\beta\gamma)$ also exist. Note that the key ingredient in the construction of this macro constitutive law is the derivation of the appropriate concentration matrices, $A(\alpha\beta\gamma)$ and $D(\alpha\beta\gamma)$ having the dimensions $6 \times 6$ and $6 \times 6$ by $N_\alpha N_\beta N_\gamma$, respectively, at the micro (subcell) level. The definitions of $A$ and $D$, although not given here, may be found in references [2] and [5]. Finally, the matrix $C(\alpha\beta\gamma)$ represents the elastic stiffness tensor of each subcell $(\alpha\beta\gamma)$ and $d_\alpha$, $h_\beta$, $l_\gamma$ the respective subcell dimensions (see Fig. 2) wherein,

$$d = \sum_{\alpha=1}^{N_\alpha} d_\alpha \quad h = \sum_{\beta=1}^{N_\beta} h_\beta \quad l = \sum_{\gamma=1}^{N_\gamma} l_\gamma$$

Similarly, given the concentration matrices $A(\alpha\beta\gamma)$ and $D(\alpha\beta\gamma)$, expressions for the average strain in each subcell can be constructed, i.e.,

$$\bar{\varepsilon}(\alpha\beta\gamma) = A(\alpha\beta\gamma) \bar{\varepsilon} + D(\alpha\beta\gamma) \varepsilon^T_s + \varepsilon^T_s$$

as well as average stress,

$$\bar{\sigma}(\alpha\beta\gamma) = C(\alpha\beta\gamma) [A(\alpha\beta\gamma) \bar{\varepsilon} + D(\alpha\beta\gamma) \varepsilon^T_s + \varepsilon^T_s] - (\varepsilon^T(\alpha\beta\gamma) + \varepsilon^T(\alpha\beta\gamma))$$

The analytic constitutive law, see eq. 1, may be readily applied to investigate the behavior of various types of composites, given knowledge of the behavior of the individual phases. Numerous advantages can be stated regarding the current macro/micro constitutive laws as compared to the other numerical micromechanical approaches in the literature, e.g. the finite element unit cell approach. One advantage is that any type of simple or combined loading (multiaxial state of stress) can be applied irrespective of whether symmetry exists or not, as well as without resorting to different boundary condition application strategies as in the case of the finite element unit cell procedure. Another, advantage concerns the
Double Periodicity

\[ \beta = N_\beta \]
\[ \gamma = 1 \quad \gamma = 2 \quad \gamma = N_\gamma \]

\[ \alpha = 3 \quad d_3 \]
\[ \alpha = 2 \quad d_2 \]
\[ \alpha = 1 \quad d_1 \]

\[ \beta = 1 \quad \beta = 2 \quad \beta = 3 \quad \beta = 4 \]

Figure 2: Subcell Dimension Nomenclature
availability of an analytical expression representing the macro elastic-thermo-
inelastic constitutive law thus ensuring a reduction in memory requirements when implementing this formulation into a structural finite element analysis code. Furthermore, this formulation has been shown to predict accurate macro behavior given only a few subcells, within the repeating cell (see references [2], and [4]). Whereas, if one employs the finite element unit cell procedure, a significant number of finite elements are required within a given repeating unit cell to obtain the same level of accuracy as with the present formulation. Consequently, it is possible to utilize this formulation to efficiently analyze metal matrix composite structures subjected to complex thermomechanical load histories. This is particularly important when analyzing realistic structural components, since different loading conditions exist throughout the structure, thus necessitating the application of the macromechanical equations repeatedly at these locations.

3.2 Integration Algorithms

There are two integration algorithms currently available within MAC. The first is the standard, explicit Forward Euler algorithm, which can be expressed as,

\[ w_{i+1} = w_i + \Delta t f(t_i, w_i) \]

where

\[ w_i = \gamma(t_i) \]
\[ w_{i+1} = \gamma(t_{i+1}) \]

and \( f(t_i, w_i) \) is the rate of change with respect to time of the vector, \( \gamma(t_i) \), i.e.

\[ f(t_i, w_i) = \dot{\gamma} = \frac{d\gamma(t_i)}{dt} \]

The second is a predictor/corrector algorithm which uses:

1) a 4\textsuperscript{th} order Runge-Kutta starter:

\[ w_i = w_{i-1} + \frac{(K_1 + 2K_2 + 2K_3 + K_4)}{6} \]

where
\[ K_1 = \Delta t f(t_{i-1}, w_{i-1}) \]
\[ K_2 = \Delta t f\left(t_{i-1} + \frac{\Delta t}{2}, w_{i-1} + \frac{K_1}{2}\right) \]
\[ K_3 = \Delta t f\left(t_{i-1} + \frac{\Delta t}{2}, w_{i-1} + \frac{K_2}{2}\right) \]
\[ K_4 = \Delta t f(t_{i-1} + \Delta t, w_{i-1} + K_3) \]

2) with an Adams Bashforth four-step predictor:
\[ w_0 = \alpha_1 \quad w_1 = \alpha_2 \quad w_2 = \alpha_3 \quad w_3 = \alpha_4 \]
\[ w_{i+1}^p = w_i + \frac{\Delta t}{24} \left[ 55 f(t_i, w_i) - 59 f(t_{i-1}, w_{i-1}) + 37 f(t_{i-2}, w_{i-2}) - 9 f(t_{i-3}, w_{i-3}) \right] \]

and 3) an Adams Moulton four step corrector:
\[ w_{i+1}^c = w_i + \frac{\Delta t}{24} \left[ 9 f(t_{i+1}, w_{i+1}^p) + 19 f(t_i, w_i) - 5 f(t_{i-1}, w_{i-1}) + f(t_{i-2}, w_{i-2}) \right] \]

where the \( \alpha \)'s come from the 4\textsuperscript{th} order Runge-Kutta starter. Further details may be found in [6].

It has been found, based on experience, that for relatively rapid monotonic or cyclic loadings it may be more efficient to use the Forward Euler integrator since the predictor/corrector requires 5 evaluations per step, as shown above. However, in the case of creep, relaxation or slow monotonic or cyclic loading histories, significant increases in solution speeds can be obtained using the predictor/corrector algorithm with a self-adaptive time step.
Finally, within MAC the vector $\gamma$ as used above contains the following macro quantities:

<table>
<thead>
<tr>
<th>position</th>
<th>contents</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 - 6</td>
<td>Macro Total Strain</td>
</tr>
<tr>
<td>7 - 12</td>
<td>Macro Stress</td>
</tr>
<tr>
<td>13 - 18</td>
<td>Macro Inelastic Strain (currently empty space for 2 6x1 vectors)</td>
</tr>
<tr>
<td>19 - 30</td>
<td></td>
</tr>
<tr>
<td>31 - 36</td>
<td>Macro Thermal Strain</td>
</tr>
<tr>
<td>37</td>
<td>Current Temperature</td>
</tr>
</tbody>
</table>

and the quantities associated with each subcell are stored sequentially in $\gamma$, such that

<table>
<thead>
<tr>
<th>position</th>
<th>contents</th>
</tr>
</thead>
<tbody>
<tr>
<td>38 - 43</td>
<td>Micro Total Strain</td>
</tr>
<tr>
<td>44 - 49</td>
<td>Micro Stress</td>
</tr>
<tr>
<td>50 - 55</td>
<td>Micro Inelastic Strain (space for 2 6x1 vectors)</td>
</tr>
<tr>
<td>56 - 67</td>
<td>Micro Internal State Variables</td>
</tr>
<tr>
<td>68 - 73</td>
<td>Micro Thermal Strain</td>
</tr>
</tbody>
</table>

The above 36 positions are repeated for the total number of (N) subcells thus bringing the total length of the $\gamma$ vector to 37+36N. It follows that a second vector of similar length contains the corresponding macro and micro rates $\dot{\gamma}$.

### 3.3 Available Constituent Constitutive Models

Currently MAC provides one elastic and four inelastic constitutive models. These models have been selected purely based upon the availability of material parameters for the materials of interest. However, MAC is designed in a modular fashion thus allowing the implementation of additional constitutive models through a user defined subroutine. Two of the four available inelastic models are capable of representing transversely isotropic material behavior, thus allowing one to investigate the reinforcement of an anisotropic matrix or the idealization of an anisotropic fiber. In all four inelastic models a purely elastic response is possible by modifying a single material parameter for each model as noted below.
### 3.3.1 Elastic Model


The following transversely isotropic model is provided for those materials that have an elastic only response, such as the fiber constituent in a composite.

\[
\begin{bmatrix}
\sigma_{11} \\
\sigma_{22} \\
\sigma_{33} \\
\sigma_{23} \\
\sigma_{13} \\
\sigma_{12}
\end{bmatrix} =
\begin{bmatrix}
C_{11} & C_{12} & C_{12} & 0 & 0 & 0 \\
C_{12} & C_{22} & C_{23} & 0 & 0 & 0 \\
C_{12} & C_{23} & C_{22} & 0 & 0 & 0 \\
0 & 0 & 0 & C_{44} & 0 & 0 \\
0 & 0 & 0 & 0 & C_{44} & 0 \\
0 & 0 & 0 & 0 & 0 & C_{66}
\end{bmatrix}\begin{bmatrix}
\varepsilon_{11} \\
\varepsilon_{22} \\
\varepsilon_{33} \\
\gamma_{23} \\
\gamma_{13} \\
\gamma_{12}
\end{bmatrix}
\]

where components \( C_{ij} \) can be expressed in terms of five independent constants, \( E_A, E_T, \nu_A, \nu_T, G_A \)

thus,

\[
\begin{align*}
C_{11} &= E_A + 4\kappa\nu_A^2 \\
C_{12} &= 2\kappa\nu_A \\
C_{22} &= \kappa + \frac{0.5E_T}{1+\nu_T} \\
C_{23} &= \kappa - \frac{0.5E_T}{1+\nu_T} \\
C_{44} &= G_A \\
C_{66} &= \frac{(C_{22} - C_{23})}{2}
\end{align*}
\]
3.3.2 Bodner-Partom Model


This model represents the Bodner-Partom viscoplastic model with isotropic hardening, \( Z \), and can be used for an initially isotropic metallic material.

The flow law is given as:

\[
\dot{\varepsilon}_{ij} = \Lambda s_{ij}
\]

where

\[
\Lambda = \sqrt{\frac{D_2^{PL}}{J_2}}
\]

\[
D_2^{PL} = D_0^2 \exp \left[ -\left( \frac{A_2^2}{J_2} \right)^n \right]
\]

\[
A_2^2 = \frac{1}{3} Z_{eff} \left( \frac{n+1}{n} \right)
\]

\[
J_2 = \frac{1}{2} S_{ij} S_{ij}
\]

\[
S_{ij} = \sigma_{ij} - \frac{1}{3} \sigma_{kk} \delta_{ij}
\]

The evolution law for isotropic hardening is given as:

\[
\dot{Z} = m(Z_1 - Z_{eff}) \frac{W_{PL}}{Z_0}
\]

where \( Z_0 \), \( Z_1 \) and \( m \) are inelastic constants and the plastic work rate, \( W_{PL} \), is given by:

\[
W_{PL} = \sigma_{ij} \dot{\varepsilon}_{ij}
\]

\[
Z_{eff} = Z_0 + q \int_0^t \dot{Z}(\tau)d\tau + (1 - q) \sum_{i,j=1}^3 r_{ij} \int_0^t \dot{Z}(\tau) r_{ij}(\tau)d\tau
\]

\[
r_{ij}(t) = \sigma_{ij}(t) / [\sigma_{kl}(t) \sigma_{kl}(t)]^{1/2}
\]

An elastic only response may be obtained by setting the material parameter \( D_0 \) to zero.
3.3.3 Robinson Creep Model


This model represents a transversely isotropic material, wherein the vector of direction cosines, \( d_i \) defines the strong material direction along which no inelasticity occurs.

Flow Law:

\[
\frac{\dot{\varepsilon}_{ij}}{\dot{\varepsilon}_0} = 2F^n \left( \frac{\Gamma_{ij}}{\sigma_0 \sqrt{F}} \right)
\]

Evolution Law:

\[
\dot{\alpha}_{ij} = \frac{H}{G} \frac{1}{2} \frac{\dot{\varepsilon}_{ij}}{\dot{\varepsilon}_0} - R \frac{G_{mn} - \rho}{\sigma_0 \sqrt{G}} \Pi_{ij}
\]

where

\[
\Gamma_{ij} = \Sigma_{ij} - \frac{1}{2} l (3D_{ij} - \delta_{ij})
\]

\[
\Pi_{ij} = a_{ij} - \frac{1}{2} l (3D_{ij} - \delta_{ij})
\]

in which

\[
F = \frac{4}{3} \phi \quad \phi = J_2 - \frac{3}{4} l^2
\]

\[
G = \frac{4}{3} \psi \quad \psi = \tilde{J}_2 - \frac{3}{4} \tilde{l}^2
\]

and

\[
J_2 = \frac{1}{2} \Sigma_{ij} \Sigma_{ji} \quad l = D_{ij} \Sigma_{ji} \quad D_{ij} = a_{ij} \quad \Sigma_{ij} = S_{ij} - a_{ij}
\]

\[
\tilde{J}_2 = \frac{1}{2} \tilde{a}_{ij} \tilde{a}_{ji} \quad \tilde{l} = D_{ij} \tilde{a}_{ji} \quad \tilde{S}_{ij} = \sigma_{ij} - \frac{1}{3} \sigma_{kk} \delta_{ij}
\]

Special cases involving an isotropic material can be obtained by defining \( d_i = (1/3, 1/3, 1/3) \) and/or elastic only response by setting \( \sigma_0 \) to an extremely large number.
3.3.4 Robinson Viscoplastic Model

Reference: NASA TM 105819, 1992

This model represents a transversely isotropic material wherein the vector of direction cosines $d_i$ defines the preferred material direction. In this model the strength of anisotropy is specified by the parameters $\omega$ and $\eta$; where $\omega$ is the ratio of the normal longitudinal and transverse yield stress and $\eta$ is the ratio of longitudinal and transverse shear strengths.

Flow Law:

$$\dot{\varepsilon}_{ij} = \frac{F^n}{2\mu} \Gamma_{ij}$$

Evolution Law

$$\dot{\alpha}_{ij} = \frac{H}{G^8} \dot{\varepsilon}_{ij} - R G^m \beta \Pi_{ij}$$

where

$$\Gamma_{ij} = \Sigma_{ij} - \xi (D_{ki} \Sigma_{jk} + D_{jk} \Sigma_{ki} - 2 I_0 D_{ij}) - \frac{1}{2} \zeta I_0 (3 D_{ij} - \delta_{ij})$$

$$\Pi_{ij} = a_{ij} - \xi (D_{ki} a_{jk} + D_{jk} a_{ki} - 2 I_0 D_{ij}) - \frac{1}{2} \zeta I_0 (3 D_{ij} - \delta_{ij})$$

and

$$F = \frac{1}{\kappa^2} \left[ I_1 + \frac{1}{\eta^2} I_2 + \frac{9}{4(4\omega^2 - 1)} I_3 \right] - 1$$

$$\hat{G} = \frac{1}{\kappa^2} \left[ I_1 + \frac{1}{\eta^2} I_2 + \frac{9}{4(4\omega^2 - 1)} I_3 \right]$$

$$G = \langle \hat{G} - \hat{G}_0 \rangle H \nu [S_{ij} \pi_{ij}] + \hat{G}_0$$

$$I_1 = J_2 - I - \frac{1}{4} I_3 \quad I_2 = I - I_3 \quad I_3 = I_0^2$$

$$J_2 = \frac{1}{2} \Sigma_{ij} \Sigma_{ji} \quad I = D_{ij} \Sigma_{ji} \quad D_{ij} = d_i d_j \quad \Sigma_{ij} = S_{ij} - a_{ij}$$

$$\xi = \frac{\eta^2 - 1}{\eta^2} \quad \zeta = \frac{4(\omega^2 - 1)}{4\omega^2 - 1}$$
The invariants \( I_1, I_2, I_3 \) are the same as those given above but with \( \Sigma_{ij} \) replaced by \( a_{ij} \). Special cases involving an isotropic material and/or elastic only response can be obtained by defining \( \omega = \eta = 1 \) and/or by setting \( \kappa_T \) to an extremely large number.
3.3.5 Generalized Viscoplastic Potential Structure (GVIPS) Model

Reference: NASA TM 106609, 1994

This model is a fully associative, multiaxial, isothermal, nonlinear kinematic hardening viscoplastic model for use with initially isotropic metallic materials. A unique aspect of this model is the inclusion of non-linear hardening through the use of a compliance operator $Q_{ijkl}$ in the evolution law for the back stress. This non-linear tensorial operator is significant in that it allows both the flow and evolutionary laws to be fully associative (and therefore easily integrated) and greatly influences the multiaxial response under non-proportional loading paths.

Flow Law:

$$\dot{\varepsilon}_{ij} = \frac{3}{2} \dot{\varepsilon}_{ij} \frac{\Sigma_{ij}}{\sqrt{J_2}} \quad \text{if} \quad F \geq 0$$

where

$$\dot{\varepsilon}_{ij} = \sqrt{3} \dot{\varepsilon}_{ij} \dot{\varepsilon}_{ij} = \frac{\mu F^n}{\kappa}$$

Internal constitutive rate equation

$$\dot{a}_{ij} = L_{ijrs} A_{rs}$$

Evolution Law:

$$A_{rs} = \dot{\varepsilon}_{rs} - \frac{3\beta}{2\kappa} \dot{\varepsilon}_{ij} \left( \frac{a_{rs}}{\sqrt{G}} H_v[Y] - \frac{3R_{\alpha} B_0 G^q}{\kappa^2} a_{rs} \right) \quad \text{if} \quad a_{ij} \Sigma_{ij} \geq 0$$

$$A_{rs} = Q_{rstm} E_{imnp} \left( \dot{\varepsilon}_{np} - \frac{3\beta}{2\kappa} \dot{\varepsilon}_{ij} \frac{a_{np}}{\sqrt{G}} H_v[Y] - \frac{3R_{\alpha} B_0 G^q}{\kappa^2} a_{np} \right) \quad \text{if} \quad a_{ij} \Sigma_{ij} < 0$$

where
\[ F = \left\langle \frac{\sqrt{I_2}}{\kappa} - Y \right\rangle \]

\[ Y = \left\langle 1 - \beta_s \sqrt{G} \right\rangle \]

\[ G = \frac{I_2}{\kappa^2} \]

\[ L_{ijrs} = [Q_{ijrs}]^{-1} = \frac{\kappa^2}{3B_0(1 + B_1 p G^p - 1)} \left( I_{ijrs} - \frac{3B_1(p - 1)G^{p - 2}}{\kappa^2(1 + B_1 p G^p - 1)(6p - 5)} a_{rs} a_{ij} \right) \]

and

\[ I_2 = \frac{3}{2} a_{ij} a_{ij} \quad \Sigma_{ij} = S_{ij} - a_{ij} \]

\[ J_2 = \frac{3}{2} \Sigma_{ij} \Sigma_{ij} \quad S_{ij} = \sigma_{ij} - \frac{1}{3} \sigma_{kk} \delta_{ij} \]

The special case of an elastic only response maybe obtained by setting \( \kappa \) to an extremely large value.
3.4 Laminate Theory

Version 2.0 of MAC now includes the capability to analyze composite laminates. In-plane axial displacements and membrane forces may be applied to the laminate. That is, the global laminate stress-strain relation that is solved within MAC is expressed as,

\[
\begin{bmatrix}
N_{XX} \\
N_{YX} \\
N_{XY}
\end{bmatrix}
= \begin{bmatrix}
A_{11} & A_{12} & A_{13} \\
A_{21} & A_{22} & A_{23} \\
A_{31} & A_{32} & A_{33}
\end{bmatrix}
\begin{bmatrix}
\bar{e}_{xx} \\
\bar{e}_{xy} \\
\bar{e}_{yx}
\end{bmatrix}
- \begin{bmatrix}
N'_{XX} \\
N'_{XY} \\
N'_{YX}
\end{bmatrix}
- \begin{bmatrix}
N^T_{XX} \\
N^T_{XY} \\
N^T_{YX}
\end{bmatrix}
\]

(EQ 11)

or

\[
\bar{N} = A \bar{e} - \bar{N} - N^T
\]

(EQ 12)

where \( \bar{N} \), \( \bar{N}' \), and \( \bar{N}^T \) are the global laminate total, inelastic and thermal stress resultants, respectively. The matrix \( A \) is the global laminate stiffness and \( \bar{e} \) the global laminate strains.

In forming the laminate stiffness \( A \), the generalized method of cells model, GMC, is utilized to calculate the individual lamina properties. In this regard, the individual laminate stiffness, in lamina coordinates, \( Q \) is given by,

\[
Q = \begin{bmatrix}
Q_{11} & Q_{12} & 0 \\
Q_{21} & Q_{22} & 0 \\
0 & 0 & Q_{66}
\end{bmatrix}
\]

(EQ 13)

in which the components of \( Q \) are given as,

\[
Q_{11} = C_{11} - \frac{C_{13}C_{31}}{C_{33}} \quad Q_{12} = C_{12} - \frac{C_{13}C_{23}}{C_{33}} \\
Q_{22} = C_{22} - \frac{C_{23}C_{32}}{C_{33}} \quad Q_{33} = C_{66}
\]

(EQ 14)

The \( C_{ij} \) in the above are the effective macro properties for the unidirectional composite lamina obtained from GMC.

It follows that the lamina stress-strain relation in global (laminate) coordinates is given by the relation,
\[
\begin{bmatrix}
\sigma_x \\
\sigma_y \\
\sigma_{xy}
\end{bmatrix}_k =
\begin{bmatrix}
\bar{Q}_{11} & \bar{Q}_{12} & \bar{Q}_{13} \\
\bar{Q}_{12} & \bar{Q}_{22} & \bar{Q}_{23} \\
\bar{Q}_{13} & \bar{Q}_{23} & \bar{Q}_{33}
\end{bmatrix}_k
\begin{bmatrix}
\varepsilon_x \\
\varepsilon_y \\
\varepsilon_{xy}
\end{bmatrix}_k
\]
\] (EQ 15)

or

\[
\bar{\bar{e}} = \bar{Q} \bar{\bar{e}}
\] (EQ 16)

where

\[
\bar{Q} = R^{-1}QR
\] (EQ 17)

\[
R =
\begin{bmatrix}
\cos^2 \theta & \sin^2 \theta & 2 \sin \theta \cos \theta \\
\sin^2 \theta & \cos^2 \theta & (-2) \sin \theta \cos \theta \\
-\sin \theta \cos \theta & \sin \theta \cos \theta & \cos^2 \theta - \sin^2 \theta
\end{bmatrix}
\] (EQ 18)

and \( \theta \) is the orientation of the longitudinal lamina axis with respect to the global x direction, thus \( \bar{Q}_k \) is the transformed lamina stiffness, i.e. from local lamina to global laminate coordinates. In addition, \( \bar{\bar{e}} \) and \( \bar{\bar{e}} \) are the lamina stress and strain in laminate coordinates. It then follows that the global laminate stiffness \( \bar{A} \) is given by,

\[
\bar{A} = \sum_{k=1}^{nly} \bar{Q}_k t_k
\] (EQ 19)

in which \( nly \) is the total number of layers in the laminate and \( t_k \) is the thickness of the \( k^{th} \) lamina.

Returning to EQ. 12, the quantities \( \bar{N}_k^I \) and \( \bar{N}_k^T \) (the laminate inelastic and thermal stress resultants, respectively) are calculated from the individual lamina contributions through the following relations,

\[
\bar{\bar{N}}^I = \sum_{k=1}^{nly} \bar{N}_k^I t_k \quad \bar{\bar{N}}^T = \sum_{k=1}^{nly} \bar{N}_k^T t_k
\] (EQ 20)

The individual lamina stress resultants \( \bar{N}_k^I \) and \( \bar{N}_k^T \) are given by,

\[
\bar{N}_k^I = \bar{Q}_k \bar{\bar{e}}_k^I
\] (EQ 21)
for the inelastic strains and

\[ N^T_k = Q_k \xi_k^T \quad (EQ \ 22) \]

for the thermal strains.
### 3.5 User Defined Material Model

Another addition to version 2.0 of MAC, is the option for a user to implement their own constitutive model. This is accomplished by using the subroutine USRMAT into which the user writes the necessary FORTRAN code for the particular constitutive model being implemented. The USRMAT subroutine is as shown below.

#### USRMAT

```fortran
C******************************************************************************
SUBROUTINE USRMAT(DSA,SA,PE,PV,
     TIME,TSTEP,CTEMP,DTEMPR,TREF,NIO,NPE,NPV,NCE)
C
C purpose: user supplied constitutive model
C
C note: 1) in this subroutine, [sa] and [dsa] contain the
         "micro" quantities for aboudi's micromechanics model
C
C 2) arrangement of [dsa] & [sa] arrays are as follows
C
C variable       location
C
<table>
<thead>
<tr>
<th>strain rate</th>
<th>(1-6)</th>
</tr>
</thead>
<tbody>
<tr>
<td>stress rate</td>
<td>(7-12)</td>
</tr>
<tr>
<td>inelastic</td>
<td></td>
</tr>
<tr>
<td>strain rate</td>
<td>(13-18)</td>
</tr>
<tr>
<td>12 &quot;slots&quot;</td>
<td>(19-30)</td>
</tr>
<tr>
<td>for state variables</td>
<td></td>
</tr>
<tr>
<td>thermal strain rate</td>
<td>(31-36)</td>
</tr>
</tbody>
</table>

IMPLICIT DOUBLE PRECISION (A-H,O-Z)

DIMENSION DSA(36),SA(36),PE(NPE),PV(NPV)
```

---

23
A description of the input and output required for the USRMAT subroutine is as follows:

Program data available to USRMAT:

- **SA**: array containing current total quantities for all of the state variables
- **PE**: array containing elastic constants
- **PV**: array containing inelastic constants
- **TIME**: current time
- **TSTEP**: current time increment (step)
- **CTEMP**: current temperature
- **DTEMPR**: current temperature rate
- **TREF**: reference temperature
- **NPE**: total number of elastic constants
- **NPV**: total number of inelastic constants
- **NIO**: output file unit number
- **NCE**: current subcell number

Output expected from USRMAT:

- **DSA**: current increments in all state variables

The state variables are arranged in SA and DSA in the following order:

<table>
<thead>
<tr>
<th>Position</th>
<th>Quantity</th>
</tr>
</thead>
<tbody>
<tr>
<td>1-6</td>
<td>strain</td>
</tr>
<tr>
<td>7-12</td>
<td>stress</td>
</tr>
<tr>
<td>13-18</td>
<td>inelastic strain</td>
</tr>
<tr>
<td>19-30</td>
<td>available space for 2 6x1 vectors for model</td>
</tr>
<tr>
<td></td>
<td>dependent internal state variables</td>
</tr>
<tr>
<td>31-36</td>
<td>thermal strain</td>
</tr>
</tbody>
</table>

Again, SA contains the total quantities and DSA contains the rates.

**Note**: Appendix H contains a sample input file and the USRMAT subroutine containing an implementation of the Bodner-Partom viscoplastic material model.
3.6 Interface Modeling

Interfaces in composite materials play a major role in the determination of their mechanical and thermal properties. Consequently, it is important to have the ability to model interface behavior accurately. This is accomplished in MAC/GMC in one of two ways. The first is to define an actual interface region with its own constitutive behavior. In this way the influence of initial imperfections (flaws, voids, improper wetting, etc.) and induced interfacial damage (due to stress, environment, chemical reactions, etc.) may be incorporated into the micromechanical analysis of the overall behavior of the composite. The development of proper interfacial constitutive models is an active area of research and MAC, through the use of its USRMAT routine, provides the researcher with a convenient tool for testing new and existing interfacial constitutive models.

The second approach to modeling the effect of imperfect (weak) bonding between two phases (e.g. a fiber and matrix) is to assume a jump in the displacement field at a particle-matrix interface may occur given certain conditions, while still maintaining continuity of the traction vector. In the spirit of Jones and Whittier (see ref. 10) and Achenbach and Zhu (see ref. 11) we have assumed the following “elastic-perfectly plastic” flexible interface model.

\[
\begin{align*}
\begin{bmatrix}
\hat{u}_n^I = R_n \cdot \hat{\sigma}_n^I \\
\hat{u}_t^I = R_t \cdot \hat{\sigma}_t^I
\end{bmatrix} & \quad \text{if} \quad \begin{bmatrix}
\sigma_n^I \geq \sigma_{DBn} \\
\sigma_t^I \geq \sigma_{DBt}
\end{bmatrix} \\
\begin{bmatrix}
\hat{u}_n^I = 0 \\
\hat{u}_t^I = 0
\end{bmatrix} & \quad \text{if} \quad \begin{bmatrix}
\sigma_n^I < \sigma_{DBn} \\
\sigma_t^I < \sigma_{DBt}
\end{bmatrix}
\end{align*}
\]  

(EQ 23)  

(EQ 24)

where \( R_n, R_t, \sigma_{DBn}, \) and \( \sigma_{DBt} \) are the interfacial normal and shear, strength and debond stress, respectively. Implementation of this model impacts the definition of the concentration matrices, \( A^{(c)} \) discussed in section 3.1.
3.7 References


4.0 Running MAC

MAC has been developed for a Sun IPX workstation running Sun Solaris. Upon linking the associated object modules that comprise MAC, execution is begun by simply typing MAC (in upper case) and hitting return.

4.1 Input and Output Files

MAC requires that the names of the input and output files be specified. The user is prompted for the names of the input, infilename, and output file, outfilename, and the names of the input and output files maybe anything the user desires. As will be explained in the following sections, the user will also need to specify the names of the PATRAN output files and the files containing the X-Y data for plotting.

Note that a "debug" file may be generated depending on the "PRINT" level the user specifies in the input file (see 4.2.2). This file will have the same name as the output file but with the string "_debug" automatically appended at the end of the filename, i.e., outfilename_debug.

<table>
<thead>
<tr>
<th>Input</th>
<th>Output</th>
</tr>
</thead>
</table>
| infilename | outfilename
|            | debug file (outfilename__debug)     |
|            | PATRAN files (optional, see section 4.2.11) |
|            | plot data files (see section 4.2.15) |

4.2 Input Requirements

This section describes how the data should appear in the input file, infilename. In the following, each block of input data will have its own subsection and will typically contain the following information:

1) statement of purpose

2) declaration of input data block

3) example(s)

4) notes

A data block has the following general format:

1) *keyword beginning of data block

2) input line(s)

3) % end of data block
where

1) *keyword: (denotes beginning of data block)

Each input data block begins with the appropriate keyword, which starts with the * symbol. For example, the keyword for selecting the method of integration is *SOLVER. The input routine scans the input file and locates the appropriate keyword and then reads the corresponding input data.

2) Input line(s):

The input lines contain the necessary input data. The specific format of these input lines will be given in the next section.

Two special characters (& and #) are provided for entering the input.

The "&" (continuation) symbol:

For input data that is too long to fit on a single line, the "&" symbol is used as the continuation character. Thus, a line of data may be divided into a series of lines. For example, when entering the material properties all of the data cannot fit on a single line, thus the continuation character is required:

```
EL=11700.,11700.,0.365,0.365,4287.5,1.,1. &
VI=0.8E-8,0.1,0.1E-5,0.,0.85E-3,0.05,1.,1.,3.3,1.8,1.35,1.,0.01
```

The "#" (comment) symbol:

The input file may also contain "comments" for the users convenience. The "#" symbol is used to mark a comment line, with the requirement that the "#" symbol appears in the first column.

3) % (denotes end of data block)

Each input block ends with the "%" symbol. The % symbol must be included as it signals the input routine that a particular data block has been completed.

A sample data block is as follows,

```
*SOLVER
#   NTF=2 ISTM=0.0001 ERR=0.1E-3
   NTF=2 ISTM=0.000024 ERR=0.1E-2%
```

Note how the # symbol is used to comment out a line of input data. Thus, the user can change various parameters by simply commenting out input lines containing different input data. Also note how the data block is terminated with the % symbol and how it need not appear on a separate line.
**Note**: It is suggested that the order of keywords in a given input file follow that given in this manual.

### 4.2.1 Header Line:

**Purpose**: Define the title of this particular job (80A format).

**Note**: The **Header Line** is limited to one 80 character line, and is always taken as the first line in the input file.

*problem title*

**Example**: Transverse tensile response of 35% SCS6/Ti--6-4

### 4.2.2 Output Print Level:

**Purpose**: To control the output generated

\begin{verbatim}
*PRINT
   NPL=nplvl
%
\end{verbatim}

Where

*nplvl*:
-1 = print out macro stiffness matrix, engineering constants and macro thermal expansion coefficients
0 = minimal output (most commonly used)
7 = basic program trace
10 = program execution trace and all array data

**WARNING**: this generates a very large output file

**Example**: (minimal print out)

\begin{verbatim}
*PRINT
   NPL=0%
\end{verbatim}


4.2.3 Load Type:

Purpose: To select load type

*LOAD
   LCON=nsel   LOP=lop   LSS=iopt
%

Where

nsel:
1 = Thermal Load
2 = Mechanical Load
3 = Thermomechanical Load

lop: (for 2-D and 3-D GMC)
1 = axial load in 1-direction
2 = axial load in 2-direction
3 = axial load in 3-direction
4 = shear load 23-direction
5 = shear load 13-direction
6 = shear load 12-direction

For 2-D GMC:
7 = axial loads in 1 and 2 directions
8 = axial loads in 2 and 3 directions
9 = axial load in 1-direction and shear load in 23-direction
10 = axial load in 2-direction and shear load in 13-direction

For 3-D GMC:
7 = axial loads in 1 and 2 directions
8 = axial loads in 2 and 3 directions
9 = axial loads in 1 and 3 directions
10 = axial load in 1-direction and shear load in 23-direction
11 = axial load in 2-direction and shear load in 13-direction
12 = axial load in 3-direction and shear load in 12-direction

iopt:
1 = Strain control
2 = Stress control
Figure 3: Coordinate Systems
**4.2.4 Mechanical Load Control:**

Note: This block is only required if LCON = 2 or 3

Purpose: Select type of load control for mechanical load

*MECH

\[
\text{NPTW}=\text{nptw} \quad \text{T1}=t_1, t_2, ..., t_{nptw} \quad \text{LO}=l_1, l_2, ..., l_{nptw}
\%
\]

Where

- \( nptw \): number of points on load curve
- \( t_1, t_2, ..., t_{nptw} \): time values
- \( l_1, l_2, ..., l_{nptw} \): load curve values

Example:

*MECH

\[
\text{NPTW}=3 \quad \text{T1}=0., 1., 2. \quad \text{LO}=0., 20., 30. \%
\]

![Stress or Strain Graph](image)

Note: For LOP=7,8,...etc (i.e. two load/displacement components) the following format is required:

*MECH

\[
\begin{align*}
\text{NPTW}=2 & \quad \text{T1}=0., 1.5 & \text{LO}=0., 0.015 & \text{component 1 curve} \\
\text{NPTW}=2 & \quad \text{T1}=0., 1.5 & \text{LO}=0., 0.015 & \text{component 2 curve}
\end{align*}
\]

(See Appendix G for an example)
4.2.5 Temperature Control:

Note: This block is only required if LCON = 1 or 3

Purpose: Select control for temperature

*THERM
NPTT=nptt  Tl=t1, t2,...tnptw  TE= te1, te2,...te nptt

% 

Where

nptt: number of points on temperature curve

T1, T2, ...Tnptw: time values

TE1, TE2, ...TEnptt: temperature curve values

Example:

*THERM
NPTT=3  Tl=0., 1., 2.  TE=100., 200., 150. %
Note: For the Thermomechanical Load $t_{nptw} = t_{nptt}$, and both curves must have $t_1 = 0$. But the number and time value of the data points in-between maybe different, see figure 4.

**Figure 4: Load History Specification**
4.2.6 **Reference Temperature:**

Purpose: Define initial (starting) temperature

* TREF
  TREF=tref

%

Where

tref:
--- temperature

Example: (set reference temperature at 70.)

* TREF
  TREF=70. %

☞ **Note:** only important in thermal or thermomechanical load case when using secant values of coefficient of thermal expansion.

☞ **Note:** Units must be consistent with selected material model parameter units
4.2.7 Micromechanics Model Identification:

Purpose: Select desired GMC micromechanics model

*MODEL
MOD=modid  MATSYS=matsys

Note: If modid=3 (laminate option) you must enter the following line of data,
NLY=nly  THK=thk1, thk2, ... thknly  CON=c1, c2, ... cnly  SYS=s1, s2, ... sny
ANG=a1, a2, ... anly

Where

modid:
1 = double periodicity
2 = triple periodicity
3 = laminate option

matsys: - number of different material systems in the analysis
Note: matsys is required only for laminate option (MOD=3)

nly: - number of layers in the laminate

thk1, thk2, ... thknly: - thickness of each layer

c1, c2, ... cnly: - layer material type id: 1 = isotropic 2 = anisotropic

s1, s2, ... sny: - material system id number

a1, a2, ... anly: - angles of each layer

Example 1:
double periodicity model

*MODEL
MOD=1 %

Example 2:
laminate option with 1 material system

*MODEL
MOD=3 MATSYS=1 NLY=4 THK=0.25,0.25,0.25,0.25 CON=2,2,2,2 &
SYS=1,1,1,1 ANG=45.,-45.,-45.,45. %
4.2.8 PATRAN Output: (Optional)

Purpose: To select PATRAN output file generation.

\*PATRAN
  FN=prefix  STP=npstp  STO=s1,s2,...,snstp
%

Where

prefix:
  --- this will be the filename prefix that will be assigned to all PATRAN files.

npstp:
  the number of steps to be stored for plotting

s1,s2,...snstp:
  step numbers at which data is to be stored

\textbf{Note: This is an optional input line}. If no PATRAN files are desired, do not include this data block.

Example 1: (select PATRAN output, use file prefix “run1”)

\*PATRAN
   fn=run1  STP=3  STO=1,50,100  %

\textbf{Note: If this option is chosen, MAC generates the following PATRAN files:}
(all files are written in PATRAN 2.5 neutral file format in which the subcells are treated as “elements”)

1) \texttt{prefix.patgeo}: contains “geometry” information of the RVE.

2) \texttt{prefix.patstr}: contains micro-stress quantities
   (file is formatted as a PATRAN 2.5 element results file)

3) \texttt{prefix.patepsin}: contains micro-inelastic strain quantities

4) \texttt{prefix.patepsto}: contains micro-total strain quantities

For Example 1, the following files would be produced:

run1.patgeo
run1.patstr
run1.patepsin
run1.patepsto

and will contain data for load steps 1, 50, and 100
4.2.9 Integrator Identification
Purpose: Select type of integration scheme

*SOLVER
NTF=ntf

If NTF=1, enter the following line
NPTS=npts TIM=t1,t2,...tnpts STP=st1,sti,...stnpts

If NTF=2, enter the following line
ISTM=istpm ISTT=istpt ERR=errtol
%

Where

ntf:
1 = Forward Euler method
2 = Predictor/Corrector method

istpm:
= initial mechanical load time step
istpt:
= initial thermal load time step

Note: only istpm is required for a mechanical load (nse=2), only istpt is required for a thermal load (nse=1) and both istpm and istpt are required for a thermomechanical load (nse=3)

st:
= initial time step for that load increment
errtol:
= error tolerance for predictor/corrector

Note: errtol is only required when using predictor/corrector (ntf = 2)
Suggested errtol = 0.1

Example 1: for a mechanical load, select predictor/corrector, with initial time step = 0.001 and error tolerance = 0.01

*SOLVER
NTF=2
ISTM=0.001 ERR=0.01%
Example 2: select forward euler, time step = 0.001

*SOLVER
   NTF=1
   NPTS=3  TIM=0.,1.,2.  STP=0.01,0.02 %

\[ st_1=0.01 \quad st_2=0.02 \]
\[ t_1=0. \quad t_2=1. \quad t_3=2. \]

4.2.10 Thermal Conductivity Calculation: (Optional)

Purpose: Calculate thermal conductivity of the composite

*COND

\( \text{\textbf{Note:}} \) If the *COND keyword is not used, the thermal conductivity calculations are ignored.
4.2.11 Constituent Material Model Identification:

Purpose: To select the model for the fiber and matrix constituents

*FIBER

NFIBS=nfib

The following line is to be repeated for each fiber (nfibs):

\[ NF=n_{f_{f}} \quad MS=m_{s_{f}} \quad MF=n_{cmd_{f}} \quad NDPT=d_{pt} \quad MAT=m_{at_{f}} \]

IDB=id_{bf_{f}} \quad f=1,2,...nfibs

*MATRIX

NMATX=nmatx

The following line is to be repeated for each matrix (nmatx):

\[ NM=n_{m_{m}} \quad MS=m_{s_{m}} \quad MA=n_{cmd_{m}} \quad NDPT=d_{pt} \quad MAT=m_{at_{m}} \]

IDB=id_{bm_{m}} \quad m=1,2,...nmatx

*MONOL (available for laminate option only)

NMON=nmon

The following line is to be repeated for each isotropic material (nmon):

\[ NMO=n_{m_{iso}} \quad MS=m_{s_{iso}} \quad MMO=n_{cmd_{iso}} \quad NDPT=d_{pt} \quad MAT=m_{at_{iso}} \]

IDB=id_{biso_{iso}} \quad iso=1,2,...nmon

%

Where

nfibs:
number of different fibers

nmatx:
number of different matrices

nmon:
number of different isotropic layers

nf_{f}:
fiber material designation number running from 1 to nfibs, sequentially.

nm_{m}:
matrix material designation number running from 1 to nmatx, sequentially.

nmo_{iso}:
isotropic material designation number running from 1 to nmon.

ms_{f}:
fiber material system id (required only when using laminate option)
$m_{sm}$: matrix material system id. (required only when using laminate option)

$m_{iso}$: isotropic material system id (required only when using laminate option)

$ncmd$: material model identifier for either fiber, matrix or isotropic layer:
1 = Bodner-Partom Model
2 = Robinson Creep Model
3 = Robinson Non-normalized Viscoplastic Model
4 = Generalized Viscoplastic Potential Structure (GVIPS) Model
6 = Elastic Material Model
99 = User defined model (see note on page 44 for special format instructions)

$mat$: material identification letter for either fiber or matrix, selected from material database, see Table I or II depending upon.

$idb$: read material constants from the database (y) or read from input file (n)

$dpt$: (This flag is only required for the Bodner-Partom model)
flag indicating whether material constants should be temperature independent or temperature dependent
1 = Temperature Independent
2 = Temperature Dependent

**Table I: Isothermal Material Constants**

<table>
<thead>
<tr>
<th>Model</th>
<th>Material</th>
<th>Units</th>
<th>$mat$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Linear Elastic $ncmd = 6$</td>
<td>Boron</td>
<td>MPa</td>
<td>A</td>
</tr>
<tr>
<td></td>
<td>SCS-6</td>
<td>MPa</td>
<td>B</td>
</tr>
<tr>
<td></td>
<td>Tungsten</td>
<td>ksi</td>
<td>C</td>
</tr>
<tr>
<td></td>
<td>SCS-6</td>
<td>ksi</td>
<td>D</td>
</tr>
</tbody>
</table>
Example 1:
select 1 fiber, Boron, and 1 matrix material, Timetal21S, both read from database
the Boron Timetal21S composite

*FIBER
NFIB=1
NF=1 MF=6 MAT=A IDB=y%

*MATRIX
NMATX=1
NM=1 MA=4 MAT=A IDB=y%

« Note: See Section 3.3 for a mathematical description of each material model.

Example 2: (select 2 matrix materials; material 1: Boron, read from database, material 2: Aluminum, user supplied properties)

*MATRIX
NMATX=2
NM=1 MA=6 MAT=A IDB=y
NM=2 MA=1 MAT=A IDB=n EL=E, v, G, α &
VL=a, b, c, d, ... K=κ_A, κ_T %

« Note: The κ_A, κ_T data is only required if the *COND keyword has been used; where κ_A, κ_T are the axial and transverse thermal conductivities, respectively.
Note: Required Format for User Supplied Isothermal Material Constants:

Bodner-Partom: \( ncmd = 1 \)

Elastic:
\[ EL = E_L, \ E_T, \ \nu_A, \ \nu_T, \ G_A, \ \alpha_A, \ \alpha_T \]

Inelastic:
\[ n = D_0, \ Z_0, \ Z_1, \ m, \ n, q \]

Robinson Creep: \( ncmd = 2 \)

Elastic:
\[ EL = E_L, \ E_T, \ \nu_A, \ \nu_T, \ G_A, \ \alpha_A, \ \alpha_T \]

Inelastic:
\[ n = \sigma_0, \ \epsilon_0, \ n, \ \beta, \ m, \ R, \ H \]

Robinson Viscoplastic: \( ncmd = 3 \)

Elastic:
\[ EL = E_L, \ E_T, \ \nu_A, \ \nu_T, \ G_A, \ \alpha_A, \ \alpha_T \]

Inelastic:
\[ n = n, \ \mu, \ \kappa_T, \ \beta, \ R, \ H, \ \hat{G}_0, \ \eta, \ \omega \]

GVIPS: \( ncmd = 4 \)

Elastic:
\[ EL = E, \ \nu \]

Inelastic:
\[ n = \mu, \ \kappa, \ R_\alpha, \ R_\kappa, \ B_0, \ B_1, \ L_0, \ L_1, \ m, \ n, p, q, \ w, \ z_0 \]

Elastic Model: \( ncmd = 6 \)

\[ EL = E_L, \ E_T, \ \nu_A, \ \nu_T, \ G_A, \ \alpha_A, \ \alpha_T \]

Table II. Non-isothermal Material Constants

The following materials are available for a non-isothermal analysis:

\[ MAC \] Materials Database

<table>
<thead>
<tr>
<th>Model</th>
<th>Material</th>
<th>Units</th>
<th>mat</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bodner-Partom</td>
<td>Graphite</td>
<td>MPa, sec, °C</td>
<td>A</td>
</tr>
<tr>
<td>( ncmd = 1 )</td>
<td>Aluminum</td>
<td>&quot;</td>
<td></td>
</tr>
<tr>
<td>GVIPS</td>
<td>TIMETAL21S</td>
<td>ksi, sec</td>
<td>A</td>
</tr>
<tr>
<td>( ncmd = 4 )</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Required Format for User Supplied Non-Isothermal Material Constants:

Note: each of the following data statements are separate lines.
MAC: Micromechanics Analysis Code

NTP=ntpts
TEM=T₁, T₂, ..., Tₙtpts
EA=Eₘₐₜ₁, Eₘₐₜ₂, ..., Eₘₐₜₙtpts
ET=Eₜₜ₁, Eₜₜ₂, ..., Eₜₜₙtpts
NUA=νₘₐₜ₁, νₘₐₜ₂, ..., νₘₐₜₙtpts
NUT=νₜₜ₁, νₜₜ₂, ..., νₜₜₙtpts
GA=Gₘₐₜ₁, Gₘₐₜ₂, ..., Gₘₐₜₙtpts
ALPA=αₘₐₜ₁, αₘₐₜ₂, ..., αₘₐₜₙtpts
ALPT=αₜₜ₁, αₜₜ₂, ..., αₜₜₙtpts
D₀=D₀₁, D₀₂, ..., D₀ₙtpts
Z₀=Z₀₁, Z₀₂, ..., Z₀ₙtpts
Z₁=Z₁₁, Z₁₂, ..., Z₁ₙtpts
BM=m₁, m₂, ..., mₙtpts
AN=n₁, n₂, ..., nₙtpts
Q=q₁, q₂, ..., qₙtpts

Note: For MF=99 (user defined material model) the following special format is required

*FIBER
NFIBS=nfibs

The following line is to be repeated for each fiber (nfibs):
NF=nf MS=ms MF=99f NPE=npe EL=e₁, e₂, ..., e_npe &
NPV=npv VI=v₁, v₂, ..., v_npv K=k₁ k₂ %

*MATRIX
NMATX=nmatx

The following line is to be repeated for each matrix (nmatx):
NM=nm MS=ms MA=99m NPE=npe EL=e₁, e₂, ..., e_npe &
NPV=npv VI=v₁, v₂, ..., v_npv K=k₁ k₂ %
where:

\( npe \):
total number of elastic constants

\( \theta_1, \theta_2, \ldots \)
elastic constants

\( npv \):
total number of inelastic constants

\( v_1, v_2, \ldots \)
inelastic constants
4.2.12 RVE Data:

Purpose: Select RVE representing desired fiber packing arrangement/architecture

*MRVE
IDP=idp

**Note:** The following data is entered on the same line as idp (except where noted)

**Without Interface:**

- For IDP = 1, 2, or 3
  \[ VF=vf \]
- For IDP = 4
  \[ VF=vf \quad XA=xa \]
- For IDP = 6
  \[ VF=vf \quad R=R \]
- For IDP = 9
  \[ VF1=vf1 \quad RAD1=rad1 \quad VF2=vf2 \quad RAD2=rad2 \quad R=R \]
- For IDP = 11
  \[ VF=vf1 \quad RAD=rad1 \quad R=R \]

**With Interface:**

- For IDP = 1, 2, or 3
  \[ VF=vf \quad RAD=rad1 \quad CPER=cper \]
- For IDP = 4
  \[ \text{**Currently NOT Available**} \]
- For IDP = 6
  \[ \text{**Currently NOT Available**} \]
- For IDP = 9
  \[ VF1=vf1 \quad RAD1=rad1 \quad CPER1=cper1 \quad VF2=vf2 \quad RAD2=rad2 \quad CPER2=cper2 \quad R=R \]
- For IDP = 11
  \[ VF=vf1 \quad RAD=rad1 \quad R=R \quad CPER=cper \]

% where:

- \( vf, vf1, vf2 \) = the fiber volume ratios
- \( rad, rad1, rad2 \) = fiber radii
- \( cper, cper1, cper2 \) = ratios of interface thickness to fiber radius
- \( xa \) = length of the cross, see Fig. 6.
$R = \frac{X}{Y}$ which defines the ratio of distances between fibers within a “ply” and those between a “ply” (see figure 7)

$idp$: ID of RVE as given below

**Note: For Double Periodic RVE:**

<table>
<thead>
<tr>
<th>idp</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Square Fiber, Square Pack (original 4-cell model)</td>
</tr>
<tr>
<td></td>
<td>$V_f \leq \frac{1}{(1 + \Delta)^2}$</td>
</tr>
<tr>
<td></td>
<td>RVE shown in Fig. 5</td>
</tr>
<tr>
<td>2</td>
<td>Square Fiber, Triangular (hexagonal) Pack</td>
</tr>
<tr>
<td></td>
<td>$V_f \leq \frac{0.86602}{(1 + \Delta)^2}$</td>
</tr>
<tr>
<td></td>
<td>RVE shown in Fig. 5</td>
</tr>
<tr>
<td>3</td>
<td>Square Fiber, Square Diagonal Pack</td>
</tr>
<tr>
<td></td>
<td>$V_f \leq \frac{0.5}{(1 + \Delta)^2}$</td>
</tr>
<tr>
<td></td>
<td>RVE shown in Fig. 5</td>
</tr>
<tr>
<td>4</td>
<td>Cross Shaped Fiber, Square Pack</td>
</tr>
<tr>
<td></td>
<td>$V_f \leq 1 - 4(xa)^2$</td>
</tr>
<tr>
<td></td>
<td>RVE shown in Fig. 5</td>
</tr>
<tr>
<td>6</td>
<td>Circular Fiber Approximation Rectangular or Square Pack</td>
</tr>
<tr>
<td></td>
<td>$V_f \leq \frac{0.8125}{R(1 + \Delta)^2}$ if $R &gt; 1.0$</td>
</tr>
<tr>
<td></td>
<td>$V_f \leq \frac{R(0.8125)}{(1 + \Delta)^2}$ if $R &lt; 1.0$</td>
</tr>
<tr>
<td></td>
<td>RVE shown in Fig. 5</td>
</tr>
</tbody>
</table>
9 = Two Different Size Square Fibers, Rectangular or Square Pack

\[ V_{f_2} \leq \frac{1}{R \left[ (1 + \Delta_2) + (1 + \Delta_1) \frac{R_{f_1}}{R_{f_2}} \right]^2} \]

\[ V_{f_1} \leq \frac{2 \sqrt{R V_{f_2}}}{\left[ \left( \frac{R_{f_2}}{R_{f_1}} \right)^2 (1 + \Delta_2) + \left( \frac{R_{f_2}}{R_{f_1}} \right) (1 + \Delta_1) \right]} \]

RVE shown in Fig. 5

11 = Square Fiber, Rectangular Pack

\[ V_f \leq \frac{1}{R(1 + \Delta)^2} \quad \text{if} \quad R > 1.0 \]

\[ V_f \leq \frac{R}{(1 + \Delta)^2} \quad \text{if} \quad R < 1.0 \]

RVE shown in Fig. 5

99 = User Defined RVE

Example of RVE representing random packing shown in Fig. 8. Required input shown in Appendix F.

Note: For Triple Periodic RVE:

99 = User Defined RVE

Note: \( R, R_{f_1}, R_{f_2}, V_f, \Delta = \text{oper}, \Delta_1 = \text{oper1} \) and \( \Delta_2 = \text{oper2} \) are defined later in this subsection.

Note: The images of the RVE's shown in Figs.5 and 8 were generated using the PATRAN option (sec 4.2.8) available in MAC.

Example: Triangular packing with interface thickness 1% of fiber radius.

*MRVE

IDP=2 VF=35 RAD=0.07 CPER=0.01 %
Figure 5: RVE's Available in MAC
MAC: Micromechanics Analysis Code

Fig. 6 Cross Shaped Fiber Distance $x_a$

Fig. 7 Hybrid Composite RVE; IDP = 9. Fiber Spacing Ratio, $R = X/Y$
Note: Format for the user defined RVE’s, IDP = 99, see Fig. 8 for example.

1) For 2-D RVE: (Each line of data must be on a separate line)

NB=nb NG=ng
H=h₁, h₂, ..., hₙg
L=l₁, l₂, ..., lₙb
CM=ss₁,₁, ss₁,₂, ..., ss₁, ng

repeat data for the (nb x ng) 2-D RVE

CM=ss₁,₁, ss₁,₂, ..., ss₁, ng

2) For 3-D RVE: (Each line of data must be on a separate line)

NA=na NB=nb NG=ng
D=d₁, d₂, ..., dₙa
H=h₁, h₂, ..., hₙg
L=l₁, l₂, ..., lₙb
CM=ss₁,₁, ss₁,₂, ..., ss₁, ng

repeat data for the (na x nb x ng) 3-D RVE

CM=ss₁,₁, ng, ss₁,₂, ng, ..., ss₁, nb, ng

Where

ss₁,₁ and ss₁,i,k are the identifying material labels which are given in the following format:

first character: F - for a fiber or M - for matrix
second character: 1, 2, 3, ... for fiber/matrix number 1, 2, 3 ...

na - number of subcells in the alpha direction
nb - number of subcells in the beta direction
ng - number of subcells in the gama direction
h - the height of each subcell
l - the length of each subcell
d - the depth of each subcell
Figure 8: User defined RVE
Example: (2-D user defined RVE (as shown above)

*MRVE
  IDP=99
  NB=3   NG=3
  H=1.,1.,1.
  L=1.,1.,1.
  CM=M1,M1,M1
  CM=M1,M1,F1
  CM=F1,M1,M1  

*Note: the fiber id. is F1 and the matrix id. is M1
Example: A 3-D User defined RVE (as shown above), representing a [0/90] continuous reinforced laminate composite.

**Note:** the fiber is material number 1 and the matrix is material number 2.
*MRVE
IDP=99  ASC=1.
NA=4  NB=2  NG=2
D=1..1.,1.,1.
H=1.,1.
L=1..1.
CM=M1,M1
CM=M1,M1
CM=M1,M1
CM=F1,M1
CM=M1,M1
CM=F1,F1
CM=M1,M1
CM=F1,M1

rows 1-4 are for cross-section 1

rows 5-8 are for cross-section 2
4.2.13 Interface Data: (Optional, for IDP=1,2,3,9 or 11)

Purpose: Specify interface layer properties

*INTERFACE
  NINT=nint

The following line is to be repeated for each interface(nint):
  NM=nm, MI=ncmd, MAT=mat, IDB=idb

% Note: If idb=n see section 4.2.11 for format of additional input required.

Where

nint:
  number of different interfaces

nm:
  interface number

ncmd:
  material model identifier for the interface

mat:
  material id letter

idb:
  = y use database properties
  = n user specified properties

Note: Please refer to section 4.2.11 for ncmd and mat values

Example: (Assuming one interface with user supplied material constants)

*INTERFACE
  NINT=1
  NM=1 MI=4 MAT=C IDB=n &
  EL=1700.,1700.,0.365,0.365,4287.5,1.,1. &
  VI=0.8E-8,0.1,0.1E-5,0.,0.85E-3,0.05,1.,1.,1.,3.3,1.8,1.35,1.,0.01 %
4.2.14 Debond Data: (Optional)

Purpose: Specify the subcell faces where debond can take place (see section 3.6)

2-D GMC Model Format

*DEBOND
   NII=nii
   NBI=nbi NGI=ngi FACE=nfc RN=rn BDN=bdstrn RS=rs &
   BDS=bdstrs Tl=tmd
%

3-D GMC Model Format

*DEBOND
   NII=nii
   NAI=nai NBI=nbi NGI=ngi FACE=nfc RN=rn BDN=bdstrn RS=rs &
   BDS=bdstrs Tl=tmd
%

Where

nii:
   number of subcell interfaces with debond

nai, nbi and ngi:
   subcell indices \((\alpha, \beta, \gamma)\)

nfc:
   used specify which "face" of the subcell is to be debonded

For 2-D GMC
   = 1 Top Face
   = 2 Right Face
   = 3 Both Faces

For 3-D GMC
   = 1 Top Face
   = 4 Top & Right Face
   = 7 ALL Faces
   = 2 Right Face
   = 5 Top & Back Face
   = 6 Right & Back Face

rn:
   R_n value

bdstrn:
   debond stress, normal component (see Fig. 9)

rs:
   R_s value
bdstrs:
debond stress, shear component (see Fig. 9)

tmd:
time at which debond criteria becomes active
(defaults to tmd=0, if not specified)

Example 1:
debond subcell 1,1 top face normal component
NII=1
NBI=1 NGI=1 FACE=1 RN=1. BDN=100.

Example 2:
debond subcell 1,1 right face shear component
NII=1
NBI=1 NGI=1 FACE=2 RS=1. BDN=100.

Figure 9: Subcell Faces For 2-D GMC
Figure 10: Subcell Faces For 3-D GMC
4.2.15 Plot Point Information:
   Purpose: Specify the frequency at which data will be written to output files for both X-Y data
   *
   CURVE
   NP=nmax
   
   %
   
   Where

   nmax:
   plot point increment

   Example:: (print out every fifth data point)
   *
   CURVE
   NP=5  %
4.2.16 Curve Data:

**Note:** Currently a maximum of 5 curves may be specified per problem

**Specifying Curve Data For Macro (composite) Quantities:**

*MACRO

NT=nucuv

Repeat the following line nucuv times:

NC=nocu  X=maidx  Y=maidy  NAM=tname

% Where

nocu: curve number

maidx and maidy variable options are:

<table>
<thead>
<tr>
<th>i</th>
<th>j</th>
<th>k</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>e_{11}</td>
<td>7</td>
</tr>
<tr>
<td>2</td>
<td>e_{22}</td>
<td>8</td>
</tr>
<tr>
<td>3</td>
<td>e_{33}</td>
<td>9</td>
</tr>
<tr>
<td>4</td>
<td>\gamma_{23}</td>
<td>10</td>
</tr>
<tr>
<td>5</td>
<td>\gamma_{13}</td>
<td>11</td>
</tr>
<tr>
<td>6</td>
<td>\gamma_{12}</td>
<td>12</td>
</tr>
</tbody>
</table>

**Note:** The file(s) generated will be of the form tname_mac.data. If the user desires to use the same tname for files 2-5, a double quote, ", is entered for tname. Those files then will have the form:

| tname2_mac.data |
| tname3_mac.data |
| etc. |

(see the example for more details)
EXAMPLE:

*MACRO
   NT=3
   NC=1 X=1 Y=7 NAM=stress file created: stress_mac.data
   NC=2 X=22 Y=13 NAM=" file created: stress2_mac.data
   NC=3 X=23 Y=13 NAM= plot % file created: plot_mac.data

Specifying Curve Data for Micro (subcell) Quantities:

*MICRO
   NT=nucuv

Repeat the following line nucuv times:
   NC=nocu CELL=nsset X=maidx Y=maidy NAM=tnamex2
%

Where

nucuv: - curve number
nsset: - cell number
namex2: - name of plot file

miidx and miidy variable options are:
    1 - e_{11}    7 - \sigma_{11}    13 - e_{i1}    19 - \Phi_{11}    25 - \Psi_{11}    31 - Total Time
    2 - e_{22}    8 - \sigma_{22}    14 - e_{22}    20 - \Phi_{22}    26 - \Psi_{22}    32 - Creep Time
    3 - e_{33}    9 - \sigma_{33}    15 - e_{33}    21 - \Phi_{33}    27 - \Psi_{33}    33 - Temperature
    4 - \gamma_{23} 10 - \tau_{23}    16 - \gamma_{23} 22 - \Phi_{23}    28 - \Psi_{23}
    5 - \gamma_{13} 11 - \tau_{13}    17 - \gamma_{13} 23 - \Phi_{13}    29 - \Psi_{13}
    6 - \gamma_{12} 12 - \tau_{12}    18 - \gamma_{12} 24 - \Phi_{12}    30 - \Psi_{12}

**Note:** The quantities \Phi and \Psi are the possible internal state variables (constitutive model dependent).
Note: The file(s) generated will be of the form tname2_mic.data. If the user desires to use the same tname2 for files 2-5, a double quote, "", is entered for tname2. Those files then will have the form:

- tname2_2_mic.data
- tname2_3_mic.data
- etc.

(see the example for more details)

Note: Subcell numbering is assigned according to the following algorithm.

2-D case

\[
\text{Do } * \ IB=1, \ NB \\
\text{Do } * \ IG=1, \ NG \\
subcell \ number = NG*(IB-1)+IG \\
\text{* continue}
\]

3-D case

\[
\text{Do } * \ IA=1, \ NA \\
\text{Do } * \ IB=1, \ NB \\
\text{Do } * \ IG=1, \ NG \\
subcell \ number = IA+(IB-1)*NA+(IG-1)*NA*NB \\
\text{* continue}
\]

Note: Refer to numbers in upper left corners of subcells in Figures on page 52 and 54 for examples of subcell numbering scheme.

**EXAMPLE:**

*MICRO

NC=1  CELL=1  X=1  Y=7  NAM=cell  %

file created: cell_mic.data
5.0 Conclusion/Future Modifications

A computationally efficient, user-friendly, comprehensive, micromechanics analysis tool, MAC, has been presented that admits physically based viscoplastic deformation and life models, can analyze multiphased materials of interest in advanced propulsion systems, and can assist both the material scientist and structural analyst in developing, designing and analyzing strategic materials. However, the development of this tool is far from complete. A number of future enhancements that are planned and currently underway include:

- Provide an implicit integration algorithm to improve computational efficiency.
- Provide the ability to input a multiaxial state of loading, be it stress or strain.
- Provide the ability to input multiple runs within the same input data file.
- The incorporation of damage evolution laws and failure criteria so as to provide life estimates.

5.1 Acknowledgment

The authors would like to thank Mr. Robert Goldberg for his diligent efforts and assistance in checking the accuracy and consistency of this manual by running numerous test cases. Future users will be, as the authors are, grateful.
6.0 Appendix A

Sample Input File For A Mechanical Load Problem

The following example is used to explain the control blocks in more detail.

Problem Summary:

Load Type: Mechanical
Load Component: 22-direction (transverse to fiber)
Load History: Cyclic
Load Control: Strain
Load History Data: 
\[ \varepsilon = 8.333 \times 10^{-4}, \]
\[ \varepsilon_{\text{max}} = 0.015, \]
\[ \varepsilon_{\text{min}} = 0. \]
\[ \Delta t_{\text{initial}} = 0.0000024 \]

Micromechanics model: Double Periodicity
Fiber Packing Arrangement: Hexagonal Pack at 35% fiber volume ratio
Integration Algorithm: Predictor/Corrector
Constituent Material Model: GVIPS
Constituents: Fiber: SCS-6
Matrix: TIMETAL21S
Interface: fictitious weak interface for TIMETAL21S

Note: This example will take an extremely long time to run due to the elastic-perfectly viscoplastic definition of the fictitious weak interface, as the overall time step is limited to that of the allowable interface time step. This slow computational response is NOT indicative of GMC.
FOR MECHANICAL LOAD RUN
*PRINT
   NPL=0  %
*LOAD
   LCON=2  LOP=2  LSS=1  %
*MECH
   NPTW=2  T1=0.,18.  LO=0.,0.015  %
*TREF
   TREF=650.  %
*MODEL
   MOD=1  %
*SOLVER
   NTF=2  ISTM=0.0000024  ERR=0.1E-2  %
*FIBER
   NFIBS=1
   NF=1  MF=4  MAT=B  IDB=y  %
*MATRIX
   NMATX=1
   NM=1  NM=4  MAT=A  IDB=y  %
*MRVE
   IDP=2  VF=0.35  RAD=0.07  CPER=0.1  %
*INTERFACE
   NINT=1
   NI=1  MI=4  MAT=C  IDB=n  &
   EL=11700.,11700.,0.365,0.365,4287.5,1.,1.  &
   VI=0.8E-8,0.1,0.1E-5,0.,0.85E-3,0.05,1.,1.,3.3,1.8,1.35,1.,0.01  %
*CURVE
   NP=2  %
*MACRO
   NT=1
   NC=1  X=2  Y=8  NAM=apdxa  %
(END)
The following figure was obtained from the x-y plot data file produced by the present example.

![Graph](image)

Note: It is recommended that a new user construct a mac input file using the data given in this appendix and then check to see if the same result plot is obtained.
7.0 Appendix B

Sample Input File For A Thermal Load Problem

The following example is used to explain the control blocks in more detail.

Problem Summary:

Load Type: Thermal

Load History Data: \( T = 0.01, \)
\( T_{\text{max}} = 371.1, \)
\( T_{\text{min}} = 21.1 \)
\( \Delta t_{\text{initial}} = 17.505 \)

Micromechanics model: Double Periodicity

Fiber Packing Arrangement: Square Pack at 35% fiber volume ratio

Integration Algorithm: Predictor/Corrector

Constituent Material Model: Bodner-Partom

Constituents: Fiber: Graphite
Matrix: Aluminum

Note: This problem is taken from reference 1., pg.238
FOR THERMAL LOAD RUN
*PRINT
   NPL=0  %
*LOAD
   LCON=1  %
*THERM
   NPTT=3  T1=0.,35010.,70020. &
   TE=371.1,21.1,371.1  %
*TREF
   TREF=371.1  %
*MODEL
   MOD=1  %
*SOLVER
   NTF=2  ISTT=17.505  ERR=10.E-2  %
*FIBER
   NFIBS=1
   NF=1  MF=1  NDPT=2  MAT=A  IDB=y  %
*MATRIX
   NMATX=1
   NM=1  MM=1  NDPT=2  MAT=B  IDB=y  %
*MRVE
   IDP=1  VF=0.30  %
*CURVE
   NP=5  %
*MACRO
   NT=1
   NC=1  X=24  Y=1  NAM=thermal  %
*END

69
The following figure was obtained from the x-y plot data file produced by the present example.

**Note:** It is recommended that a new user construct a mac input file using the data given in this appendix and then check to see if the same result plot is obtained.
8.0 Appendix C

Sample Input File For A Laminate Problem

The following example is used to explain the control blocks in more detail.

Problem Summary:

Load Type: Strain Control
Load History Data: \[ \dot{\varepsilon} = 0.01, \]
\[ \varepsilon_{\text{max}} = 0.01, \]
\[ \Delta t_{\text{constant}} = 0.00025 \]

Micromechanics model: Laminate with Double Periodicity
Fiber Packing Arrangement: Square Pack at 46% fiber volume ratio
Integration Algorithm: Forward Euler
Constituent Material Model: Bodner-Partom
Constituents: Fiber: Boron
Matrix: Aluminum
test of laminate strain control

*PRINT
  NPL=0  %
*LOAD
  LCON=2  LOP=1  LSS=1  %
*MECH
  NPTW=2  TI=0.,1.  LO=0.,0.01  %
*MODEL
  MOD=3  MATSYS=1  NLY=1  THK=1.  ANG=0.  %
*SOLVER
  NTF=1  NPTS=2  TIM=0.,1.  STP=0.00025  %
*FIBER
  NFIBS=1
  NF=1  MS=1  MF=6  MAT=A  IDB=y  %
*MATRIX
  NMATX=1
  NM=1  MS=1  MM=1  MAT=A  IDB=y  %
*MRVE
  IDP=1  VF=0.46  %
*CURVE
  NP=10  %
*MACRO
  NT=1
  NC=1  X=1  Y=7  NAM=apdxc  %
*END
The following figures were obtained from the x-y plot data file produced by the present example.

From reference 1 Fig. 8-5, pg. 235

From reference 1 Fig. 8-7, pg. 237
9.0 Appendix D

Sample Input File For A Laminate Problem

The following example is used to explain the control blocks in more detail.

Problem Summary:

Load Type: Stress Control
Load History Data: \( \dot{\sigma} = 0.01 \text{ ksi/sec}, \)
\( \sigma_{\text{max}} = 45. \)
\( \Delta t_{\text{constant}} = 1.125 \)
Micromechanics model: \([\pm 45]_s\) Laminate with Double Periodicity
Fiber Packing Arrangement: Square Pack at 45% fiber volume ratio
Integration Algorithm: Forward Euler
Constituent Material Model: Bodner-Partom
Constituents: Fiber: Boron
Matrix: Aluminum

\*\* Note: This problem is taken from the reference:
Analysis of Metal-Matrix Composite Structures-II.
Laminate Analysis, Arenburg, R. T. and Reddy, J. N.,
test of [+45]2s laminate strain control

*PRINT
  NPL=0 %

*LOAD
  LCON=2 LOP=1 LSS=2 %

*MECH
  NPTW=2 T1=0.,4500. LO=0.,45. %

*TREF
  TREF=0. %

*MODEL
  MOD=3 MATSYS=1 NLY=4 THK=0.25,0.25,0.25,0.25 ANG=45.,-45.,-45.,45. %

*SOLVER
  NTF=1 NPTS=2 TIM=0.,4500. STP=1.125 %

*FIBER
  NFIBS=1
  NF=1 MS=1 MF=6 MAT=A IDB=n &
  EL=58.E3,58.E3,0.20,0.20,24.17E3,6.3E-6,8.28E-6 %

*MATRIX
  NMATX=1
  NM=1 MS=1 MM=1 MAT=A IDB=n &
  EL=9.53E3,9.53E3,0.33,0.33,3.58E3,21.06E-6,21.06E-6 &
  VI=1.E4,49.,63.,300.,4.,1. %

*MRVE
  IDP=1 VF=0.45 %

*CURVE
  NP=10 %

*MACRO
  NT=1
  NC=1 X=1 Y=7 NAM=apdxd %

*END
The following figure was obtained from the x-y plot data file produced by the present example.

From Arenburg and Reddy, Fig. 16, pg. 1382.
10.0 Appendix E

Sample Input File For Triple Periodic GMC

The following example is used to explain the control blocks in more detail.

Problem Summary:

Load Type: Strain Control
Load History Data: \( \dot{\varepsilon} = 0.01, \)
\( \varepsilon_{\text{max}} = 0.01, \)
\( \Delta t_{\text{constant}} = 0.00025 \)

Micromechanics model: Triple Periodic GMC
Fiber Packing Arrangement: Square Pack at 46% fiber volume ratio
Integration Algorithm: Forward Euler
Constituent Material Model: Bodner-Partom
Constituents: Fiber: Boron
Matrix: Aluminum

This problem uses the 3-D GMC model to simulate a [0/90] laminate as shown below.
test of gmc3d model 0/90 laminate simulation

*PRINT
  NPL=0 %

*LOAD
  LCON=2 LOP=2 LSS=1 %

*MECH
  NPTW=2 TI=0.,1. LO=0.,0.01 %

*MODEL
  MOD=2 %

*SOLVER
  NTF=1 NPTS=2 TIM=0.,1. STP=0.00025 %

*FIBER
  NFIBS=1
  NF=1 MF=6 MAT=A IDB=y %

*MATRIX
  NMATX=1
  NM=1 MM=1 MAT=A IDB=y %

*MRVE
  IDP=99 ASP=1.
  NA=4 NB=2 NG=2
  D=0.67823,0.32177,0.67823,0.32177
  H=0.67823,0.32177
  L=0.32177,0.67823
  CM=2,2
  CM=2,2
  CM=2,2
  CM=1,2
  CM=2,2
  CM=1,1
  CM=2,2
  CM=1,2 %

*CURVE
  NP=10 %

*MACRO
  NT=1
  NC=1 X=2 Y=8 NAM=apdxe %

*END
The following figure was obtained from the x-y plot data file produced by the present example.

![Graph showing stress-strain relationship](image)

**Note:** It is recommended that a new user construct a mac input file using the data given in this appendix and then check to see if the same result plot is obtained.
11.0 Appendix F

Sample Input File For A User Defined RVE

The following example is used to explain the control blocks in more detail.

Problem Summary:

Load Type: Strain Control
Load History Data: \[ \dot{\varepsilon} = 0.01, \]
\[ \varepsilon_{\text{max}} = 0.001, \]
\[ \Delta t_{\text{constant}} = 0.1 \]

Micromechanics model: Double periodic
Fiber Packing Arrangement: Random composite (see figure 8)
Integration Algorithm: Forward Euler
Constituent Material Model: Bodner-Partom
Constituents: Fiber: Boron
Matrix: Aluminum
random composite rve using 2-d gmc

*PRINT
  NPL=0  

*LOAD
  LCON=2 LOP=1 LSS=2  

*MECH
  NPTW=2 TI=0.,0.1 LO=0.,0.001  

*MODEL
  MOD=1  

*SOLVER
  NTF=1 NPTS=2 TIM=0.,0.1 STP=0.1 

*FIBER
  NFIBS=1
  NF=1 MF=6 MAT=A IDB=y  

*MATRIX
  NMATX=1
  NM=1 MM=1 MAT=A IDB=y  

*MRVE
  IDP=99
  NB=14 NG=14
  H=1.,1.,1.,1.,1.,1.,1.,1.,1.,1.,1.
  L=1.,1.,1.,1.,1.,1.,1.,1.,1.,1.,1.,1.
  CM=F1,M1,M1,M1,M1,F1,M1,M1,F1,M1,F1,M1,F1
  CM=M1,M1,F1,F1,M1,M1,F1,M1,F1,M1,F1,M1,F1
  CM=M1,M1,M1,M1,M1,M1,M1,M1,M1,F1,F1,M1,F1
  CM=M1,M1,F1,F1,M1,M1,F1,M1,F1,M1,F1,M1,F1
  CM=M1,M1,F1,F1,M1,M1,F1,M1,F1,M1,F1,M1,F1
  CM=M1,M1,F1,F1,M1,M1,F1,M1,F1,M1,F1,M1,F1
  CM=F1,M1,M1,M1,M1,M1,M1,M1,M1,M1,F1,F1,M1,F1
  CM=M1,M1,M1,M1,M1,F1,F1,M1,M1,F1,F1,M1,F1
  CM=M1,M1,F1,F1,M1,M1,F1,F1,M1,F1,F1,M1,F1
  CM=M1,M1,F1,F1,M1,M1,F1,F1,M1,F1,F1,M1,F1
  CM=M1,M1,F1,F1,M1,M1,F1,F1,M1,F1,F1,M1,F1
  CM=M1,M1,F1,F1,M1,M1,F1,F1,M1,F1,F1,M1,F1
  CM=M1,M1,F1,F1,M1,M1,F1,F1,M1,F1,F1,M1,F1
  CM=M1,M1,F1,F1,M1,M1,F1,F1,M1,F1,F1,M1,F1
  CM=F1,F1,M1,M1,M1,M1,M1,M1,M1,F1,F1,M1,M1
  CM=M1,M1,F1,F1,M1,M1,F1,F1,M1,F1,F1,M1,F1
  CM=F1,F1,M1,M1,M1,M1,M1,M1,M1,F1,F1,M1,M1
  CM=M1,M1,M1,M1,M1,M1,M1,M1,M1,F1,F1,M1,F1
  CM=M1,M1,M1,M1,M1,M1,M1,M1,M1,F1,F1,M1,F1
  CM=F1,F1,M1,M1,M1,M1,M1,F1,F1,M1,M1,F1,F1
  CM=F1,F1,M1,M1,M1,M1,M1,F1,F1,M1,M1,F1,F1
  CM=M1,M1,M1,M1,M1,M1,M1,M1,M1,F1,F1,M1,F1
  CM=M1,M1,M1,M1,M1,M1,M1,M1,M1,F1,F1,M1,F1
  CM=F1,F1,M1,M1,M1,M1,M1,F1,F1,M1,M1,F1,F1
  CM=F1,F1,M1,M1,M1,M1,M1,F1,F1,M1,M1,F1,F1
  CM=F1,F1,M1,M1,M1,M1,M1,F1,F1,M1,M1,F1,F1
  CM=F1,F1,M1,M1,M1,M1,M1,F1,F1,M1,M1,F1,F1
  CM=F1,F1,M1,M1,M1,M1,M1,F1,F1,M1,M1,F1,F1
  CM=F1,F1,M1,M1,M1,M1,M1,F1,F1,M1,M1,F1,F1
  CM=F1,F1,M1,M1,M1,M1,M1,F1,F1,M1,M1,F1,F1
  CM=F1,F1,M1,M1,M1,M1,M1,F1,F1,M1,M1,F1,F1
  CM=F1,F1,M1,M1,M1,M1,M1,F1,F1,M1,M1,F1,F1
  CM=F1,F1,M1,M1,M1,M1,M1,F1,F1,M1,M1,F1,F1
  CM=F1,F1,M1,M1,M1,M1,M1,F1,F1,M1,M1,F1,F1
  CM=F1,F1,M1,M1,M1,M1,M1,F1,F1,M1,M1,F1,F1

*CURVE
  NP=1 

*MACRO
  NT=1
  NC=1 X=1 Y=7 NAM=apdxf  

*END
12.0 Appendix G

Sample Input File For A Biaxial Load

The following example is used to explain the control blocks in more detail.

Problem Summary:

<table>
<thead>
<tr>
<th>Load Type:</th>
<th>Strain Control</th>
</tr>
</thead>
<tbody>
<tr>
<td>Load History Data:</td>
<td>( \varepsilon_{22} = 0.01 ) ( \dot{\varepsilon}_{33} = 0.0067 )</td>
</tr>
<tr>
<td></td>
<td>( \varepsilon_{22\text{max}} = 0.015 ) ( \varepsilon_{33\text{max}} = 0.01 )</td>
</tr>
<tr>
<td></td>
<td>( \Delta t_{\text{constant}} = 0.015 )</td>
</tr>
<tr>
<td>Micromechanics model:</td>
<td>Double Periodic</td>
</tr>
<tr>
<td>Fiber Packing Arrangement:</td>
<td>Square Pack at 35% fiber volume ratio</td>
</tr>
<tr>
<td>Integration Algorithm:</td>
<td>Forward Euler</td>
</tr>
<tr>
<td>Constituent Material Model:</td>
<td>Elastic</td>
</tr>
</tbody>
</table>
MAC: Micromechanics Analysis Code

test of biaxial load
*PRINT
   NPL=7 %
*LOAD
   LCON=2 LOP=7 LSS=1 %
*MECH
   NPTW=2 Tl=0.,1.5 LO=0.,0.015
   NPTW=2 Tl=0.,1.5 LO=0.,0.010 %
*TREF
   TREF=900. %
*MODEL
   MOD=1 %
*SOLVER
   NTF=1 NPTS=2 TIM=0.,1.5 STP=0.015 %
*FIBER
   NFIBS=1
   NF=1 MF=4 MAT=B IDB=n &
   EL=58.E3,58.E3,0.32,0.32,23.2E3,2.1E-06,2.1E-06 &
   VI=0.8E-9,1.E20,0.1E-5,0.,0.85E-4,0.05,1.,1.,3.3,1.35,1.,0.01 %
*MATRIX
   NMATX=1
   NM=1 MM=4 MAT=A IDB=y %
*MRVE
   IDP=1 VF=0.35 %
*CURVE
   NP=1 %
*MACRO
   NT=2
   NC=1 X=1 Y=7 NAM=biaxl
   NC=2 X=2 Y=8 NAM=biaxt %
*END
13.0 Appendix H

Sample Input File For A User Defined Material Model

The following example is used to explain the control blocks in more detail.

Problem Summary:

Load Type: Stress Control

Load History Data:

\[ \sigma = 0.01 \text{ ksi/sec}, \]
\[ \sigma_{\text{max}} = 45., \]
\[ \Delta t_{\text{constant}} = 1.125 \]

Micromechanics model: \([\pm 45]_s\) Laminate with Double Periodicity

Fiber Packing Arrangement: Square Pack at 45% fiber volume ratio

Integration Algorithm: Forward Euler

Constituent Material Model: Bodner-Partom using user subroutine USRMAT

Constituents:

Fiber: Boron
Matrix: Aluminum

*Note: This problem is the same as that given in Appendix D. The purpose here is to demonstrate how to use the USRMAT option.
use of usrmat for a \([\pm 45]2s\) laminate strain control problem

*PRINT
  NPL=0 %
*LOAD
  LCON=2 LOP=1 LSS=2 %
*MECH
  NPTW=2 TI=0.,4500. LO=0.,45. %
*TREF
  TREF=0. %
*MODEL
  MOD=3 MATSYS=1 NLY=4 THK=0.25,0.25,0.25,0.25 ANG=45.,-45.,-45.,45. %
*SOLVER
  NTF=1 NPTS=2 TIM=0.,4500. STP=1.125 %
*FIBER
  NFIBS=1
  NF=1 MS=1 MF=99 NPE=7 EL=58.E3,58.E3,0.2,0.2,24.17E3,6.3E-6,8.28E-6 &
  NPV=6 VI=0.,1.,1.,1.,1.,1. %
*MATRIX
  NMATX=1
  NM=1 MS=1 MM=99 NPE=7 &
  EL=9.53E3,9.53E3,0.33,0.33,3.58E3,21.06E-6,21.06E-6 &
  NPV=6 VI=1.E4,49.,63.,300.,4.,1. %
*MRVE
  IDP=1 VF=0.45 %
*CURVE
  NP=10 %
*MACRO
  NT=1
  NC=1 X=1 Y=7 NAM=apdxg %
*END
The USRMAT subroutine

The USRMAT subroutine is used here to implement the Bodner-Partom Visco-plastic model currently available in MAC.

```
SUBROUTINE USRMAT(DSA,SA,PE,PV,
& TIME,TSTEP,CTEMP,DTEMP,TREF,NIO,NPE,NPV,NCE)

purpose: user material model

note: 1) in this subroutine, [sa] and [dsa] contain the "micro" quantities for aboudi's micromechanics model

2) arrangement of [dsa] & [sa] arrays

variable location

<table>
<thead>
<tr>
<th>l strain rate (1-6)</th>
</tr>
</thead>
<tbody>
<tr>
<td>l stress rate (7-12)</td>
</tr>
<tr>
<td>l inelastic</td>
</tr>
<tr>
<td>l strain rate (13-18)</td>
</tr>
<tr>
<td>l 12 &quot;slots&quot; (19-30)</td>
</tr>
<tr>
<td>l for state variables</td>
</tr>
<tr>
<td>l thermal strain rate (31-36)</td>
</tr>
</tbody>
</table>

IMPLICIT DOUBLE PRECISION (A-H,O-Z)

DIMENSION DSA(36),SA(36),PE(NPE),PV(NPV)

default user supplied material routine

dimension SS(6),S(6),R(6)
```
MAC: Micromechanics Analysis Code

c write(nio,*) 'bodner, ctemp, ctemp, time, time

------------------------------------------------------------------------------------------------------------------------
c extract appropriate viscoplastic material constants
------------------------------------------------------------------------------------------------------------------------
D0 = PV(1)
Z0 = PV(2)
Z1 = PV(3)
BM = PV(4)
AN = PV(5)
Q = PV(6)

------------------------------------------------------------------------------------------------------------------------
c copy stress from [sa] to [s]
------------------------------------------------------------------------------------------------------------------------
SS(1) = SA(7)
SS(2) = SA(8)
SS(3) = SA(9)
SS(4) = SA(10)
SS(5) = SA(11)
SS(6) = SA(12)

------------------------------------------------------------------------------------------------------------------------
c compute the deviatoric stress [s] in the subcell
------------------------------------------------------------------------------------------------------------------------
TEMP = (SS(1) + SS(2) + SS(3))/3.
S(1) = SS(1) - TEMP
S(2) = SS(2) - TEMP
S(3) = SS(3) - TEMP
S(4) = SS(4)
S(5) = SS(5)
S(6) = SS(6)

------------------------------------------------------------------------------------------------------------------------
c

AJ2=0.5*(S(1)**2+S(2)**2+S(3)**2)+S(4)**2+S(5)**2+S(6)**2
SQ3AJ = DSQRT( SS(1)**2 + SS(2)**2 + SS(3)**2 +
& 2*(SS(4)**2+SS(5)**2+SS(6)**2) )
SQ2=1.414215
IF(SQ3AJ.EQ.0.) THEN
CALL ZEROR(R,6)
ELSE
R(1) = SS(1)/SQ3AJ
R(2) = SS(2)/SQ3AJ
R(3) = SS(3)/SQ3AJ
R(4) = SQ2*SS(4)/SQ3AJ
R(5) = SQ2*SS(5)/SQ3AJ
R(6) = SQ2*SS(6)/SQ3AJ
ENDIF

87
c if d0=0 then assume elastic and zero-out

c [dsa(13-30)] (inelastic strain rate and internal variable rates), then return

\begin{verbatim}
C IF(D0.EQ.0) THEN
  DO 100 JJ=13,30
  DSA(JJ) = 0.0
  CONTINUE
  RETURN
C......................................................
C......................................................
C......................................................
C......................................................
C......................................................
C......................................................
C......................................................
C......................................................

C ELSE

ZEF = Z0 + Q*SA(20) +
   & (1-Q)*(R(1)*SA(21)+R(2)*SA(22)+R(3)*SA(23)+
        & R(4)*SA(24)+R(5)*SA(25)+R(6)*SA(26))
C

C IF(AJ2.EQ.0.) THEN
  AL=0.0
ELSE
  ARG1=ZEF**2.0/(3.*AJ2)
  CON=.5*(AN+1.)/AN
  ARG=CON*(ARG1)**AN
  AL=D0/(DEXP(ARG)*DSQRT(AI2))
ENDIF

C inelastic strain rates

DSA(13) = AL*S(1)
DSA(14) = AL*S(2)
DSA(15) = AL*S(3)
DSA(16) = 2*AL*S(4)
DSA(17) = 2*AL*S(5)
DSA(18) = 2*AL*S(6)

C plastic work rate

WPD = S(1)*DSA(13) + S(2)*DSA(14) + S(3)*DSA(15) +
   & S(4)*DSA(16) + S(5)*DSA(17) + S(6)*DSA(18)

C state variable rates
\end{verbatim}
MAC: Micromechanics Analysis Code

DSA(19) = WPD
Z0M = BM/Z0
ZD = Z0M * (Z1 - ZEF) * WPD
DSA(20) = ZD

C

DSA(21) = ZD * R(1)
DSA(22) = ZD * R(2)
DSA(23) = ZD * R(3)
DSA(24) = ZD * R(4)
DSA(25) = ZD * R(5)
DSA(26) = ZD * R(6)
ENDIF

C

RETURN
END

C

C#******************************************************************************
The ability to accurately predict the thermomechanical deformation response of advanced composite materials continues to play an important role in the development of these strategic materials. Analytical models that predict the effective behavior of composites are used not only by engineers performing structural analysis of large-scale composite components but also by material scientists in developing new material systems. For an analytical model to fulfill these two distinct functions it must be based on a micromechanics approach which utilizes physically based deformation and life constitutive models and allows one to generate the average (macro) response of a composite material given the properties of the individual constituents and their geometric arrangement. Here the user guide for the recently developed, computationally efficient and comprehensive micromechanics analysis code, MAC, who's predictive capability rests entirely upon the fully analytical generalized method of cells, GMC, micromechanics model is described. MAC is a versatile form of research software that "drives" the double or triply periodic micromechanics constitutive models based upon GMC. MAC enhances the basic capabilities of GMC by providing a modular framework wherein 1) various thermal, mechanical (stress or strain control) and thermomechanical load histories can be imposed, 2) different integration algorithms may be selected, 3) a variety of constituent constitutive models may be utilized and/or implemented, and 4) a variety of fiber and laminate architectures may be easily accessed through their corresponding representative volume elements.