NASA/CR-2008-215528

User-Defined Material Model for Thermo-Mechanical Progressive Failure Analysis

Norman F. Knight, Jr.
General Dynamics – Advanced Information Systems, Chantilly, Virginia

November 2008

The NASA STI Program Office . . . in Profile

Since its founding, NASA has been dedicated to the advancement of aeronautics and space science. The NASA Scientific and Technical Information (STI) Program Office plays a key part in helping NASA maintain this important role.

The NASA STI Program Office is operated by Langley Research Center, the lead center for NASA's scientific and technical information. The NASA STI Program Office provides access to the NASA STI Database, the largest collection of aeronautical and space science STI in the world. The Program Office is also NASA's institutional mechanism for disseminating the results of its research and development activities. These results are published by NASA in the NASA STI Report Series, which includes the following report types:

- TECHNICAL PUBLICATION. Reports of completed research or a major significant phase of research that present the results of NASA programs and include extensive data or theoretical analysis. Includes compilations of significant scientific and technical data and information deemed to be of continuing reference value. NASA counterpart of peer-reviewed formal professional papers, but having less stringent limitations on manuscript length and extent of graphic presentations.

- TECHNICAL MEMORANDUM. Scientific and technical findings that are preliminary or of specialized interest, e.g., quick release reports, working papers, and bibliographies that contain minimal annotation. Does not contain extensive analysis.

- CONTRACTOR REPORT. Scientific and technical findings by NASA-sponsored contractors and grantees.

- CONFERENCE PUBLICATION. Collected papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or co-sponsored by NASA.

- SPECIAL PUBLICATION. Scientific, technical, or historical information from NASA programs, projects, and missions, often concerned with subjects having substantial public interest.

- TECHNICAL TRANSLATION. English-language translations of foreign scientific and technical material pertinent to NASA's mission.

Specialized services that complement the STI Program Office's diverse offerings include creating custom thesauri, building customized databases, organizing and publishing research results ... even providing videos.

For more information about the NASA STI Program Office, see the following:

- Access the NASA STI Program Home Page at ***http://www.sti.nasa.gov***

- E-mail your question via the Internet to help@sti.nasa.gov

- Fax your question to the NASA STI Help Desk at (301) 621-0134

- Phone the NASA STI Help Desk at (301) 621-0390

- Write to:
  NASA STI Help Desk
  NASA Center for AeroSpace Information
  7115 Standard Drive
  Hanover, MD 21076-1320

NASA/CR-2008-215528

User-Defined Material Model for Thermo-Mechanical Progressive Failure Analysis

Norman F. Knight, Jr.
General Dynamics – Advanced Information Systems, Chantilly, Virginia

National Aeronautics and
Space Administration

Langley Research Center
Hampton, Virginia 23681-2199

Prepared for Langley Research Center
under Contract NNL07AA00B
Subcontract S70509

November 2008

Available from:

NASA Center for AeroSpace Information (CASI)
7115 Standard Drive
Hanover, MD 21076-1320
(301) 621-0390

National Technical Information Service (NTIS)
5285 Port Royal Road
Springfield, VA 22161-2171
(703) 605-6000

User-Defined Material Model for Thermo-Mechanical Progressive Failure Analysis

Norman F. Knight, Jr.

General Dynamics – Advanced Information Systems, Chantilly, VA

Abstract

Previously a user-defined material model for orthotropic bimodulus materials was developed for linear and nonlinear stress analysis of composite structures using either shell or solid finite elements within a nonlinear finite element analysis tool. Extensions of this user-defined material model to thermo-mechanical progressive failure analysis are described, and the required input data are documented. The extensions include providing for temperature-dependent material properties, archival of the elastic strains, and a thermal strain calculation for materials exhibiting a stress-free temperature.

Introduction

The analysis of advanced materials in a structural design necessitates having a capability to incorporate a user-defined material model within the overall stress analysis. For high-temperature applications, the thermo-mechanical response needs to be investigated. Previously, a user-defined material model was developed for the progressive failure analysis of a bimodulus orthotropic material [1]. This material model assumed that only mechanical loading was present, and the material properties were independent of temperature. Extensions to this user-defined material model to accommodate thermal loading and temperature-dependent material properties have been incorporated.

The present report describes the extensions made to the previous user-defined material model (or UMAT subroutine for ABAQUS/Standard[1]), the resulting modifications required for the input data, and the corresponding changes to the output response parameters available for archiving in the computational database (i.e., saving the numerical results for later post-processing and display). With these extensions, the present UMAT subroutine is applicable to thermo-mechanical stress analyses with temperature-dependent material properties.

The present report is organized in the following way. First, temperature-dependent material properties are defined in terms of describing the temperature dependence and interpolation for intermediate temperature values. Second, the thermal strain calculation is presented. Next, the specific extensions required for the previous UMAT subroutine [1] are documented. A sample input data file is presented to illustrate the required input data preparation. The final section summarizes these UMAT subroutine extensions.

---

[1] ABAQUS/Standard is a trademark of ABAQUS, Inc.

Temperature-Dependent Material Properties

Mechanical properties of a material typically exhibit temperature dependence. The temperature dependence is usually described assuming a piecewise linear interpolation given a set of tabular values for each material property (say $P(T)$) at different temperatures T. A set of N temperature-property data pairs are defined as (T_1, P_1), (T_2, P_2), (T_3, P_3), …,(T_N, P_N) as shown in Figure 1. These data pairs are 'ordered' data pairs (i.e., $T_1 < T_2 < T_3 < ... < T_N$) for the present UMAT subroutine implementation.

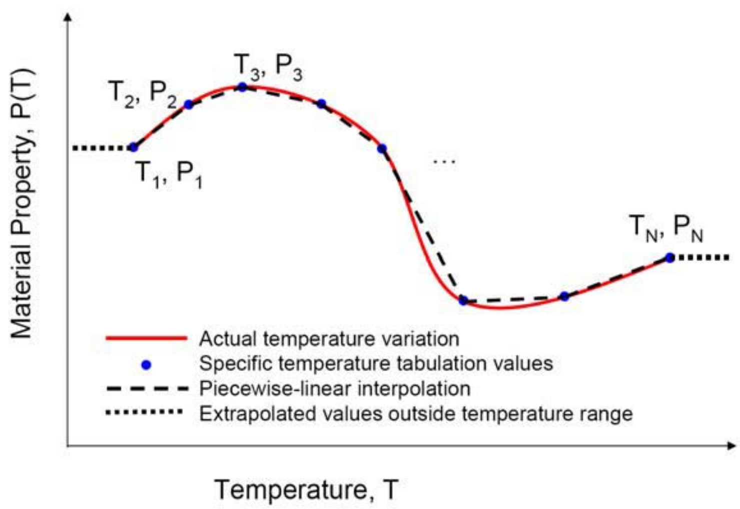

Figure 1. Illustrative sketch of representative temperature-dependent property description and interpolation process.

These N data pairs may be tabulated from measured data or from design data sheets. The tabulated values are then used with piecewise linear interpolation between data pairs to obtain a property value at an intermediate temperature value. For temperature values outside the temperature range specified in the data pairs, the property value is assumed to maintain a constant value equal to the first (or last) data pair. That is, for temperatures lower than T_1 or higher than T_N, the property value is held constant at P_1 or P_N, respectively.

The coefficient of thermal expansion (CTE) is the only material property used by this user-defined material model that requires a specific temperature input value T_{REF}. Typically, secant data are provided for a temperature-dependent coefficient of thermal expansion $\alpha(T)$ as indicated in Figure 2. During a thermal expansion test, specimen elongation or change in length ($L-L_0$) as a function of temperature is measured and the thermal strain computed. Secant-based CTE values as a function of temperature are then determined as illustrated in Figure 2. The secant-based CTE values are determined using a 'reference' temperature T_{REF}, which must be included as part of the UMAT material property input data, and explicitly used in the thermal strain calculation. In many instances, the reference temperature is equal to ambient room temperature. For materials that do not exhibit a temperature-dependent CTE, the reference temperature is still required as an input value; however, it can be any arbitrary value and does not affect the thermal strain calculation.

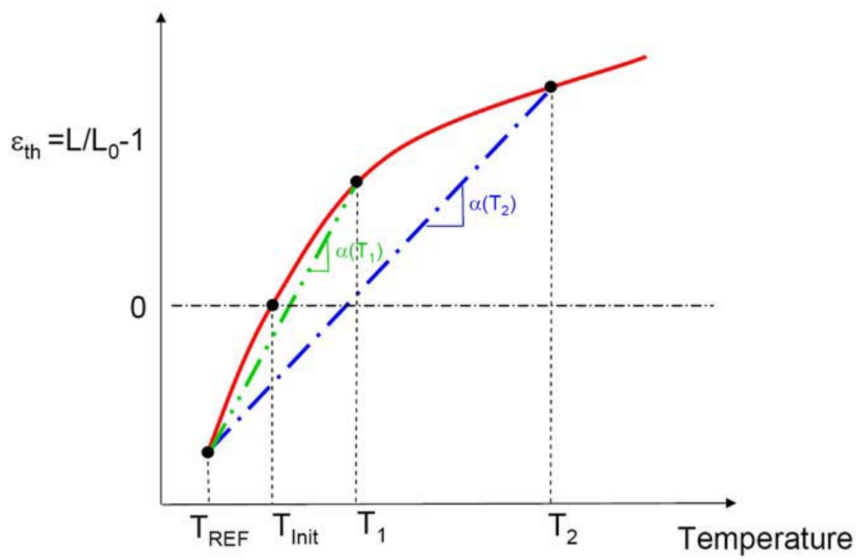

Figure 2. Sketch illustrating a one-dimensional secant-based coefficient of thermal expansion.

Thermal Strains

Thermal strains are readily computed within a finite element analysis given a temperature T value and material data as a function of temperature. For the present UMAT subroutine implementation, ABAQUS/Standard transfers the total strains to the UMAT subroutine through the subroutine calling argument list. The total strain is a sum of the mechanical strain and the thermal strain as given by:

$$\left(\varepsilon_{ij}\right)^{Total} = \left(\varepsilon_{ij}\right)^{Mechanical} + \left(\varepsilon_{ij}\right)^{Thermal} \quad (1)$$

where i and j range from one to three for three-dimensional problems and from one to two for two-dimensional problems. The normal strain components are denoted when $i=j$, and the shear strain components are denoted when $i \neq j$. The UMAT subroutine requires the mechanical strains for the local stress analysis and subsequent evaluation of failure criteria and material degradation, if desired. Hence, the present UMAT subroutine internally calculates the thermal strains for a temperature-dependent material as defined by:

$$\left(\varepsilon_{ij}\right)^{Thermal} = \begin{cases} 0 \text{ for } i \neq j \\ \alpha_i(T)\left(T - T_{\mathrm{REF}}\right) - \alpha_i(T_{Init})\left(T_{Init} - T_{\mathrm{REF}}\right) \text{ for } i = j \end{cases} \quad (2)$$

where only the normal strain components have non-zero values, $\alpha_i(T)$ are the temperature-dependent coefficients of thermal expansion, T is the current temperature, T_{Init} is the initial temperature at which no thermal strains exist, and T_{REF} is the reference temperature for a secant-based definition of the coefficients of thermal expansion (see Figure 2).

If the CTE is temperature dependent and the initial temperature is identical to the reference temperature (i.e., T_{Init}=T_{REF}), then the components of the thermal strain given in Eq. 2 simplify considerably and are:

$$\left(\varepsilon_{ij}\right)^{Thermal} = \begin{cases} 0 \ \text{ for } i \neq j \\ \alpha_i(T)\left(T - T_{\text{Init}}\right) \ \text{ for } i = j \end{cases} \tag{3}$$

where $\alpha_i(T)$ is the temperature-dependent value of the CTE.

If the CTE is temperature independent, then the components of the thermal strain are given by:

$$\left(\varepsilon_{ij}\right)^{Thermal} = \begin{cases} 0 \ \text{ for } i \neq j \\ \alpha_i^0\left(T - T_{\text{Init}}\right) \ \text{ for } i = j \end{cases} \tag{4}$$

where α_i^0 is the temperature-independent CTE value and the initial temperature is independent of the reference temperature. Note that the reference temperature T_{REF} does not explicitly appear in the thermal strain calculations of Eq. 3 or 4.

Having calculated the thermal strains, the mechanical or elastic strains are then calculated by subtracting the thermal strains from the total strains:

$$\left(\varepsilon_{ij}\right)^{Mechanical} = \left(\varepsilon_{ij}\right)^{Total} - \left(\varepsilon_{ij}\right)^{Thermal} \tag{5}$$

and used subsequently in other calculations (i.e., for stress calculations and failure criteria evaluations as defined in Ref. 1) within the UMAT subroutine.

ABAQUS/Standard Usage for Thermo-Mechanical Analysis

ABAQUS/Standard can be used for thermo-mechanical stress analysis by defining temperature conditions and selecting a material model. The present approach for thermo-mechanical stress analysis is described. ABAQUS/Standard uses the concept of analysis steps and increments within an analysis step. ABAQUS/Standard also uses the concept of defining groups of nodes or elements as "named" sets. These sets could be all the nodes (or elements) in the finite element model or nodes (or elements) from different regions on the finite element model. Initial and final temperatures are then defined for each node[2] using ABAQUS keyword commands. For example, the initial temperature for the beginning (or zeroth increment) of the first solution step is `*initial conditions, type=temperature`. The final temperature for a solution step is defined by the keyword `*temperature`. For the second solution step, the final temperature of the first solution step is treated as the initial temperature for the second solution step.

ABAQUS Material Models

Most of the material models available within ABAQUS/Standard provide for temperature-dependent material properties and thermal stress analysis. Various strains (total, elastic, and thermal) may be selected as element output variables by the user and written to the computational database (`*.odb` file).

[2] Element temperatures can not be defined.

For temperature-dependent materials, the keyword command `*expansion, zero=TREF` defines the reference temperature for secant-based CTE values. In addition, the `*initial conditions, type=temperature` keyword command defines the starting temperature for solution step 1, which is assumed to correspond to a stress-free state.

User-Defined Material Models

User-defined material models implemented as UMAT subroutines can be developed in different forms. A common approach is to omit the keyword command `*expansion, zero=TREF`, and as a result, the total strains are transferred to the UMAT subroutine. The mechanical strains must therefore be calculated within the UMAT subroutine using the approach just presented (i.e., based on Eq. 5). To post-process either the thermal or mechanical strains, they must be identified within the UMAT subroutine as solution-dependent variables to be archived in the computational database (`*.odb` file); otherwise, only the total strains are available as element output variables, if selected by the user.

User-Defined Material Model Extensions

Extensions to the previous UMAT subroutine [1] are required in three areas for thermo-mechanical stress analysis: input data extensions, addition of thermal strain calculations, and extension of the solution data archived for subsequent post-processing (i.e., increase the number of solution-dependent variables). The capabilities of the previous UMAT subroutine are preserved with these extensions; however, modifications to the input data are required even for temperature-independent material assumptions.

Input Data Extensions

Input data for a UMAT subroutine are provided through the subroutine calling argument array `PROPS` with `NPROPS` as the total number of entries in that array. Input data preparation for the present UMAT subroutine for thermo-mechanical progressive failure analysis parallels the preparation of the required input data for the previous UMAT subroutine [1] listed in Table 1. Previously, 55 entries were required as input to the UMAT subroutine for each material definition. With the extensions to include thermo-mechanical stress analysis with temperature-dependent material properties, each material definition requires 61 entries (an additional six entries) for the first temperature value (e.g., room-temperature values). The six variables beyond those required for the previous UMAT subroutine are the values for three coefficients of thermal expansion (CTE), the reference temperature for the secant-based CTE, the initial or stress-free temperature, and the temperature value for the material properties given in these first 61 entries. These data entries are usually the room-temperature values when elevated temperature cases are to be analyzed. However, the first 61 entries are reserved for the material properties corresponding to lowest temperature value provided as input (e.g., cryogenic temperatures).

Additional material properties for other temperature values are appended to these first 61 entries. For each additional temperature value, 34 entries are needed (a temperature value plus 33 property values at that temperature). The input data for the present UMAT subroutine are described in Table 2 for the first 61 entries and in Table 3 for the 34 entries for each subsequent temperature value. The total number of entries (`NPROPS`) to be defined for the `PROPS` input data array in the present UMAT subroutine for `NTEMPS` sets of temperature-dependent properties is equal to:

$$NPROPS = 61 + 34 \times (NTEMPS - 1) \tag{6}$$

That is, if material properties are to be defined over a temperature range using seven specific temperature values (i.e., NTEMPS=7), then the number of entries that need to be defined for the PROPS array is 265 (i.e., 61+34×(7-1) entries). The present implementation is limited to a maximum of ten 'ordered' data pairs for each material property (i.e., maximum value for NTEMPS is 10); however, this limitation of ten is easily increased. These 'ordered' data pairs are defined from the lowest temperature value to the highest temperature value. In addition, all material properties are required for each temperature value being provided rather than providing specific tabular data for each property. Consequently, all material properties have values at each input temperature value.

Thermal Strain Calculations

Two extensions are required for extending the previous UMAT subroutine to provide a thermo-mechanical stress analysis capability. The first extension generates the material properties at the current temperature through piecewise-linear interpolation of the input data pairs (*i=1,2,…,N*). As indicated in Figure 1, this interpolation process for property *P* at current temperature *T* has the form:

$$P(T)=\begin{cases} P_1 & \text{for } T \le T_1 \\ P_i+\dfrac{P_{i+1}-P_i}{T_{i+1}-T_i}\left(T-T_i\right) & \text{for } T_i \le T \le T_{i+1} \quad 1 \le i \le N \\ P_N & \text{for } T \ge T_N \end{cases} \tag{7}$$

where *i* ranges from one to *N*, which equals NTEMPS. ABAQUS/Standard passes the temperature at the beginning of the solution increment as well as the increment of temperature to the UMAT subroutine as subroutine calling arguments, and thereby the current temperature at that physical point in the structure is known. The present UMAT subroutine then determines the temperature interval that contains the current temperature *T* using the tabulated input data pairs (i.e., $T_i \le T \le T_{i+1}$). Given the current temperature *T* and its temperature interval, the value of each material property corresponding to that physical point and at that temperature is interpolated using Eq. 7. Once each property value at the current temperature is determined, the thermal strains can be calculated using Eq. 2.

The second extension is the addition of the calculation of the thermal strains using Eq. 2, since the previous UMAT subroutine ignored the thermal strains. Once the thermal strains are calculated within the present UMAT subroutine, then the mechanical or elastic strains are obtained, as indicated by Eq. 5, by subtracting these thermal strains from the total strains passed into the present UMAT subroutine from ABAQUS/Standard,. For the present UMAT implementation, the initial temperature T_{Init} is taken as the stress-free temperature (SFT or T_{SF}) of the material rather than the initial temperature for the start of the solution increment. The stress-free temperature value is entry 57 in the PROPS array given in Table 2.

Solution-Dependent Variables

Output data from within a UMAT subroutine is provided through the subroutine calling argument array STATEV with NSTATV as the total number of entries in that array. These solution-dependent variables listed in Table 4 and stored in the STATEV array for the present UMAT subroutine are similar to the output variables for the previous UMAT subroutine [1]. These values are element values that may be requested by the user for output to the computational database (*.odb file) for post-processing. The number of solution-dependent variables (NSTATV) listed in Table 4 for shell and solid elements increased from 8 and 14 to 12 and 20, respectively, for the present UMAT subroutine. The increase in the number of solution-dependent variables permits archiving the mechanical and thermal normal strains (Eq. 5) for a thermo-mechanical stress analysis. The number of solution-dependent variables is defined using the keyword `*DEPVAR` in the ABAQUS/Standard input for each material set included.

Sample Application

To illustrate these two approaches (i.e., existing ABAQUS/Standard material models and the present UMAT subroutine), a sample thermo-elastic stress analysis problem is posed. Here, the temperature-dependent properties of sintered silicon carbide based on material property data listed in Ref. 3 are used for illustration. For this material, the tension and compression elastic moduli are the same, and the material is assumed to be isotropic. The material properties are temperature dependent and the reference temperature for the secant-based CTE values is 70°F. Table 5 lists the temperature-dependent properties needed for a linear thermo-elastic stress analysis.

Assume that a simple three-dimensional configuration is represented by a finite element model with nodes and elements defined. The structure has an initial or starting temperature for the simulation of 10°F (cold condition) and a final temperature of 2500°F (hot condition). For illustrative purposes only, the stress-free temperature (i.e., temperature at which there is no thermal strain as indicated in Figure 2) of the material is assumed to be 500°F.[3] The analysis steps and keyword commands for ABAQUS/Standard finite element analysis code for these two material modeling approaches are somewhat different and are illustrated next.

Sample Application using ABAQUS Material Models

For an analysis using an ABAQUS material model, the temperature-dependent material properties are defined for the selected material model within ABAQUS/Standard. For the selected material model, the reference temperature for a secant-based CTE is 70°F (i.e., using the `*expansion` keyword command). The input data for the `*ELASTIC` ABAQUS material model of SiC is illustrated in Table 6. Note that the mechanical properties and the CTE values are defined using different keyword commands.

The analysis process to obtain solutions at both the cold and hot conditions involves two solution steps. The basic ABAQUS/Standard input for this process is illustrated in Table 7. Step 1 is a static analysis that uses the initial (stress free) temperature of 500°F as the starting (initial) temperature for Step 1 using the `*initial conditions, type=temperature` keyword command and increments the temperature loading to the final temperature of Step 1 (i.e., the starting cold condition of 10°F) using the `*temperature` keyword command. The result at the end of Step 1 is the linear elastic thermo-mechanical response at the cold condition. The second step is a continuation of the static analysis that uses the final temperature from Step 1 (i.e., 10°F) as the starting temperature for Step 2 and increments the temperature loading to the final temperature of Step 2 (i.e., the hot condition of 2500°F). The result at the end of Step 2 is the linear elastic thermo-mechanical response at the hot condition. If the cold condition response was not desired, then a single analysis step could have been performed where the starting (or initial) temperature is again the stress-free temperature and the final temperature is the hot condition.

Sample Application using Present UMAT Material Model

For an analysis using the present UMAT subroutine, the basic ABAQUS/Standard input is illustrated in

---

[3] Again this is only assumed to illustrate the analysis process and is NOT indicative of sintered silicon carbide.

Table 8. The reference temperature (70°F) and the initial stress-free temperature (500°F) are defined as input variables and used internally by the present UMAT subroutine to calculate the thermal strains using Eq. 2. The input data set for six temperature values (NTEMPS=6) is illustrated in Table 9 for SiC with the six temperature values being underlined (i.e., the temperatures values are 75, 1000, 1500, 2000, 2500, 2732; all degrees Fahrenheit). A total of 231 entries in the PROPS array are needed for this case (i.e., 61+34×(6-1) entries). The preparation of the material input data for the present UMAT subroutine is somewhat tedious – especially for a linear elastic isotropic material. In some cases, a separate FORTRAN computer program can be developed to simplify the data preparation step.

The number of solution-dependent variables for this sample data set is defined using the ABAQUS keyword command *DEPVAR. The sample data set shown in Table 9 contains a value of 20 for the keyword command *DEPVAR implying the data set is for a three-dimensional problem. For a two-dimensional plane stress problem, the value would be 12. The input data set is unchanged whether a two- or three-dimensional problem is to be solved.

Implicit in the present UMAT subroutine is the assumption that the 3-direction of the material system is defined as the through-the-thickness, transverse, or interlaminar normal direction, while the 1- and 2-directions are the in-plane directions for the material system. Orientation of the element and material coordinate systems needs to be performed by the user through the *orientation keyword command in ABAQUS/Standard.

The keyword command *expansion is omitted as part of the analysis input data for ABAQUS/Standard, and therefore the total strains are passed to the present UMAT subroutine. The analysis process involves two static analysis steps. The starting temperature for the analysis step is defined using the *initial conditions, type=temperature keyword command (e.g., assume the default value of 0°F as the initial value) and the final temperature for Step 1 is defined using the *temperature keyword command (i.e., the cold condition of 10°F). Then Step 2 is a continuation of the static analysis that uses the final temperature from Step 1 (i.e., 10°F) as the starting temperature for Step 2 and increments the temperature loading to the final temperature of Step 2 (i.e., the hot condition of 2500°F).

Summary

The present report describes the extensions implemented within a previous user-defined material model [1] in order to analyze temperature-dependent, bimodulus orthotropic materials subjected to thermo-mechanical loading. Extensions for the present UMAT subroutine include treatment of the temperature-dependent material properties, thermal strain calculations, and archiving mechanical or elastic strains for post-processing. The input material property data are described and the preparation of the present UMAT subroutine input data is illustrated.

References

1. Knight, N. F., Jr., *User-Defined Material Model for Progressive Failure Analysis*, NASA CR-2006-214526, December 2006.

2. Anon., *ABAQUS/Standard User's Manual*, Version 6.6, ABAQUS, Inc., Providence, RI, 2006.

3. Anon., National Institute of Standards and Technology, NIST Structural Ceramics Database, SRD Database Number 30, http://www.ceramics.nist.gov/srd/summary/scdscs.html, accessed July 29, 2008

Table 1. User-defined property data for the previous UMAT subroutine [1].

`PROPS` array entry, i	Variable name	Description
1,2,3	Et(i)	Initial elastic tension moduli: E_{11t}, E_{22t}, E_{33t}
4,5,6	Ec(i)	Initial elastic compression moduli: E_{11c}, E_{22c}, E_{33c}
7,8,9	G0(i)	Initial elastic shear moduli: G_{12}, G_{13}, G_{23}
10,11,12	Anu(i)	Poisson's ratios: ν_{12}, ν_{13}, ν_{23}
13,14,15	Xt(i)	Ultimate tension stress allowable values in the 1-, 2-, 3-directions
16,17,18	Xc(i)	Ultimate compression stress allowable values in the 1-, 2-, 3-directions
19,20,21	S(i)	Ultimate shear (12-, 13-, 23-planes) stress allowable values
22,23,24	EpsT(i)	Ultimate normal tension strain allowable values in the 1-, 2-, 3-directions
25,26,27	EpsC(i)	Ultimate normal compression strain allowable values in the 1-, 2-, 3-directions
28,29,30	GamS(i)	Ultimate shear (12-, 13-, 23-planes) strain allowable values
31,32,33	EpsTx(i)	Maximum normal tension strain allowable value in the 1-, 2-, 3-directions; *currently not used*
34,35,36	EpsCx(i)	Maximum normal compression strain allowable values in the 1-, 2-, 3-directions; *currently not used*
37,38,39	GamSx(i)	Maximum shear (12-, 13-, 23-planes) strain allowable values; *currently not used*
40	GIc	Critical strain energy release rate for Mode I fracture
41	FPZ	Width of the fracture process zone
42	SlimT	Stress limit factor for tension behavior; *currently not used*
43	SlimC	Stress limit factor for compression behavior; *currently not used*
44	SlimS	Stress limit factor for in-plane shear behavior; *currently not used*
45, 46, 47	Weibl(i)	Weibull parameter of MLT model for normal stress components (i=1, 2, 3); *currently not used*
48, 49, 50	Weibl(j)	Weibull parameter of MLT model for shear stress components (j=4, 5, 6); *currently not used*
51	Dgrd(1)	Material degradation factor for tension failures, non-zero values result in degradation after failure initiation; active only when `PDA=1` to `4`
52	Dgrd(2)	Material degradation factor for compression failures, non-zero values result in degradation after failure initiation; active only when `PDA=1` to `4`
53	Dgrd(3)	Material degradation factor for shear failures, non-zero values result in degradation after failure initiation; active only when `PDA=1` to `4`
54	RECURS	Flag for selecting the type of material degradation: 0=instantaneous, 1=recursive when `PDA=1` to `4`
55	PDA	Progressive failure analysis option (0=linear, elastic, bimodulus response, 1=use maximum stress criteria, 2=use maximum strain criteria, 3=use Tsai-Wu polynomial, 4=use Hashin criteria)

Table 2. First 61 values of the user-defined property data for the present extended UMAT subroutine.

`PROPS` array entry, i	Variable name	Description
1,2,3	Et(i)	Initial elastic tension moduli: E_{11t}, E_{22t}, E_{33t} at temperature T_1
4,5,6	Ec(i)	Initial elastic compression moduli: E_{11c}, E_{22c}, E_{33c} at T_1
7,8,9	G0(i)	Initial elastic shear moduli: G_{12}, G_{13}, G_{23} at T_1
10,11,12	Anu(i)	Poisson's ratios: ν_{12}, ν_{13}, ν_{23} at T_1
13,14,15	Xt(i)	Ultimate tension stress allowable values in the 1-, 2-, 3-directions at T_1
16,17,18	Xc(i)	Ultimate compression stress allowable values in the 1-, 2-, 3-directions at T_1
19,20,21	S(i)	Ultimate shear (12-, 13-, 23-planes) stress allowable values at T_1
22,23,24	EpsT(i)	Ultimate normal tension strain allowable values in the 1-, 2-, 3-directions at T_1
25,26,27	EpsC(i)	Ultimate normal compression strain allowable values in the 1-, 2-, 3-directions at T_1
28,29,30	GamS(i)	Ultimate shear (12-, 13-, 23-planes) strain allowable values at T_1
31,32,33	EpsTx(i)	Maximum normal tension strain allowable value in the 1-, 2-, 3-directions at T_1
34,35,36	EpsCx(i)	Maximum normal compression strain allowable values in the 1-, 2-, 3-directions at T_1
37,38,39	GamSx(i)	Maximum shear (12-, 13-, 23-planes) strain allowable values
40	GIc	Critical strain energy release rate for Mode I fracture
41	FPZ	Width of the fracture process zone
42	SlimT	Stress limit factor for tension behavior; *currently not used*
43	SlimC	Stress limit factor for compression behavior; *currently not used*
44	SlimS	Stress limit factor for in-plane shear behavior; *currently not used*
45, 46, 47	Weibl(i)	Weibull parameter of MLT model for normal stress components (i=1, 2, 3); *currently not used*
48, 49, 50	Weibl(j)	Weibull parameter of MLT model for shear stress components (j=4, 5, 6); *currently not used*
51	Dgrd(1)	Material degradation factor for tension failures, non-zero values result in degradation after failure initiation; active only when `PDA=1` to `4`
52	Dgrd(2)	Material degradation factor for compression failures, non-zero values result in degradation after failure initiation; active only when `PDA=1` to `4`
53	Dgrd(3)	Material degradation factor for shear failures, non-zero values result in degradation after failure initiation; active only when `PDA=1` to `4`
54	RECURS	Flag for selecting the type of material degradation: 0=instantaneous, 1=recursive when `PDA=1` to `4`
55	PDA	Progressive failure analysis option (0=linear, elastic, bimodulus response, 1=use maximum stress criteria, 2=use maximum strain criteria, 3=use Tsai-Wu polynomial, 4=use Hashin criteria)
56	Tref	Reference temperature for secant-based CTE values
57	Tinit	Initial (or stress free) temperature
58	Temp1	Temperature for the first set of property values, T_1
59, 60, 61	CTE(i)	Secant-based coefficients of thermal expansion in the 1-, 2-, 3-directions at T_1

Table 3. Repeating sets of the user-defined property data for each specific temperature value beyond the first value (*j=2* through NTEMPS) for the present extended UMAT subroutine.

Additional PROPS array entries, *i**	Variable name	Description
1	T(j)	Temperature for the j-th set of properties (values at T_j)
2,3,4	Et(i)	Initial elastic tension moduli: E_{11t}, E_{22t}, E_{33t} at temperature T_j
5,6,7	Ec(i)	Initial elastic compression moduli: E_{11c}, E_{22c}, E_{33c} at T_j
8,9,10	G0(i)	Initial elastic shear moduli: G_{12}, G_{13}, G_{23} at T_j
11,12,13	Anu(i)	Poisson's ratios: ν_{12}, ν_{13}, ν_{23} at T_j
14,15,16	Xt(i)	Ultimate tension stress allowable values in the 1-, 2-, 3-directions at T_j
17,18,19	Xc(i)	Ultimate compression stress allowable values in the 1-, 2-, 3-directions at T_j
20,21,22	S(i)	Ultimate shear (12-, 13-, 23-planes) stress allowable values at T_j
23,24,25	EpsT(i)	Ultimate normal tension strain allowable values in the 1-, 2-, 3-directions at T_j
26,27,28	EpsC(i)	Ultimate normal compression strain allowable values in the 1-, 2-, 3-directions at T_j
29,30,31	GamS(i)	Ultimate shear (12-, 13-, 23-planes) strain allowable values at T_j
32,33,34	CTE(i)	Secant-based coefficients of thermal expansion in the 1-, 2-, 3-directions at T_j

PROPS(61+34(j-2)+i) where j is the temperature set number and i is the entry for the j-th set.

Table 4. UMAT-defined solution-dependent variables.

STATEV array entry *i*	Solution-Dependent Variable Name	Description of Solution-Dependent Variables
Two-dimensional shell elements, `NDV=12`		
1	dmg(1)	Degradation factor for the σ_{11} stress component
2	dmg(2)	Degradation factor for the σ_{22} stress component
3	dmg(3)	Degradation factor for the σ_{12} stress component
4	fflags(1)	Failure flag for first failure mode
5	fflags(2)	Failure flag for second failure mode
6	fflags(3)	Failure flag for third failure mode
7	SEDtot	Total strain-energy density
8	Damage	Damage estimate based on energy lost (total minus recoverable)/(fracture toughness)
9	strain(1)	Mechanical normal strain component ε_{11}
10	strain(2)	Mechanical normal strain component ε_{22}
11	thstrain(1)	Thermal normal strain component ε_{11}
12	thstrain(2)	Thermal normal strain component ε_{22}
Three-dimensional solid elements, `NDV=20`		
1	dmg(1)	Degradation factor for the σ_{11} stress component
2	dmg(2)	Degradation factor for the σ_{22} stress component
3	dmg(3)	Degradation factor for the σ_{33} stress component
4	dmg(4)	Degradation factor for the σ_{12} stress component
5	dmg(5)	Degradation factor for the σ_{13} stress component
6	dmg(6)	Degradation factor for the σ_{23} stress component
7	fflags(1)	Failure flag for first failure mode
8	fflags(2)	Failure flag for second failure mode
9	fflags(3)	Failure flag for third failure mode
10	fflags(4)	Failure flag for fourth failure mode
11	fflags(5)	Failure flag for fifth failure mode
12	fflags(6)	Failure flag for sixth failure mode
13	SEDtot	Total strain-energy density
14	Damage	Damage estimate based on energy lost (total minus recoverable)/(fracture toughness)
15	strain(1)	Mechanical normal strain component ε_{11}
16	strain(2)	Mechanical normal strain component ε_{22}
17	strain(3)	Mechanical normal strain component ε_{33}
18	thstrain(1)	Thermal normal strain component ε_{11}
19	thstrain(2)	Thermal normal strain component ε_{22}
20	thstrain(3)	Thermal normal strain component ε_{33}

Note: The degradation solution-dependent variables (SDVs) should be zero until failure initiation is detected. Once failure initiation has been detected, the degradation SDVs will be non-zero and approach a value of unity (*i.e.,* complete degradation at that material point). The failure flag SDVs are the solution increment number when failure initiation at that material point and for that stress component is detected. Contour plots of the failure flag SDVs can be used to give an indication of the evolution of the damage progression.

Table 5. Typical material properties for sintered silicon carbide (SiC) based on Ref. 3.

	Temperature, °F					
	75	1000	1500	2000	2500	2732
Modulus of elasticity, Msi	13.6	13.2	13.0	12.8	12.6	12.4
Poisson's ratio	0.16	0.16	0.16	0.16	0.16	0.16
Coefficient of thermal expansion, in./in./°F (with T_{REF}=70°F)	2.4×10^{-6}	2.56×10^{-6}	2.65×10^{-6}	2.75×10^{-6}	2.85×10^{-6}	2.9×10^{-6}

Table 6. Sample ABAQUS/Standard *ELASTIC material input data for representative linear elastic isotropic material.

These input records are assumed to be contained in a files named `sic-elastic.dat` and are read into the ABAQUS/Standard input file by using the `*INCLUDE` keyword command.

```
** Material Properties Definition
*Material, name=SIC
*Elastic, type=isotropic
**  Modulus, Poisson's ratio, temperature
13.6e6, 0.16, 75.0
13.2e6, 0.16, 1000.0
13.0e6, 0.16, 1500.0
12.8e6, 0.16, 2000.0
12.6e6, 0.16, 2500.0
12.4e6, 0.16, 2732.0
*Expansion, type=iso, zero=70
** CTE, temperature
2.4e-6, 75.0
2.56e-6, 1000.0
2.65e-6, 1500.0
2.75e-6, 2000.0
2.85e-6, 2500.0
2.90e-6, 2732.0
**
```

Table 7. Selected ABAQUS/Standard input commands for sample thermo-mechanical problem using `*ELASTIC` material model.

```
** Basic ABAQUS Finite Element Model definitions and input
** Assumes FE model has NN nodes numbered sequentially from 1 to NN
** and NE elements numbered sequentially from 1 to NE
**
...
** Define element coordinate frame to have '3-direction' in the
** thickness direction
*orientation, definition=offset to nodes, name=elemframe
   2, 3, 1
** Note this orientation definition is illustrative ONLY.
**
** Assign solid section properties to named element set
*solid section, elset=ALL-Elements, material=SIC, orientation=elemframe
   1.,
**
** Material Properties Definition
*INCLUDE, input=sic-elastic.dat
**
**  Define element set
*ELSET, elset=ALL-Elements
** where [NE] is the number of elements
   1,[NE],1
** Define node set
*NSET, nset=ALL-Nodes
** where [NN] is the number of nodes in the finite element model
   1,[NN],1
**
**  Assign initial AND stress-free temperature
**  to named node set
*Initial conditions, type=temperature
    ALL-Nodes, 500.
**
** ============
** STEP: Step-1
** ============
*Step, name=Step-1, nlgeom=yes
*Static
   .1, 1., 1e-05, 1.
**
** Assign Step 1 final temperature of 10-deg. F to named node set
*Temperature
   ALL-Nodes, 10.
**
*OUTPUT, FIELD, FREQ=5
**output nodal displacements and temperatures
*NODE OUTPUT
   U, NT
** output element stresses, total, elastic, and thermal strains
*ELEMENT OUTPUT
   S,
   E,EE,THE
*END STEP
```

Table 7. Concluded.

```
** ============
** STEP: Step-2
** ============
*Step, name=Step-2, nlgeom=yes
*Static
   .1, 1., 1e-05, 1.
**
** Assign Step 2 final temperature of 2500-deg. F to named node set
*Temperature
   ALL-Nodes, 2500.
**
*OUTPUT, FIELD, FREQ=5
** output nodal displacements and temperatures
*NODE OUTPUT
   U, NT
** output element stresses, total, elastic, and thermal strains
*ELEMENT OUTPUT
   S,
   E,EE,THE
*END STEP
```

Table 8. Selected ABAQUS/Standard input commands for sample thermo-mechanical problem using the present UMAT subroutine.

```
** Basic ABAQUS Finite Element Model definitions and input
** Assumes FE model has NN nodes numbered sequentially from 1 to NN
** and NE elements numbered sequentially from 1 to NE
**
...
** Define element coordinate frame to have '3-direction' in the
** thickness direction (required for present UMAT)
*orientation,  definition=offset to nodes, name=elemframe
   2, 3, 1
** Note this orientation definition is illustrative ONLY.
**
** Assign solid section properties to named element set
*solid section, elset=ALL-Elements, material=UserSIC, orientation=elemframe
  1.,
**
** Material Properties Definition
*INCLUDE, input=sic-umat.dat
** (input data given in Table 9)
...
**
**  Define element set
*ELSET, elset=ALL-Elements
** where [NE] is the number of elements
   1,[NE],1
** Define node set
*NSET, nset=ALL-Nodes
** where [NN] is the number of nodes in the finite element model
   1,[NN],1
**
**  Assign initial temperature (NOT a stress-free temperature)
**  to named node set
*Initial conditions, type=temperature
    ALL-Nodes, 0.
**
** ============
** STEP: Step-1
** ============
*Step, name=Step-1, nlgeom=yes
*Static
   .1, 1., 1e-05, 1.
**
** Assign Step 1 final temperature of 10-deg. F to named node set
*Temperature
   ALL-Nodes, 10.
**
*OUTPUT, FIELD, FREQ=5
**output nodal displacements and temperatures
*NODE OUTPUT
   U, NT
** output element stresses, solution-dependent variables, total strains
*ELEMENT OUTPUT
   S,SDV,
   E
*END STEP
```

Table 8. Concluded.

```
** ============
** STEP: Step-2
** ============
*Step, name=Step-2, nlgeom=yes
*Static
   .1, 1., 1e-05, 1.
**
** Assign Step 2 final temperature of 2500-deg. F to named node set
*Temperature
   ALL-Nodes, 2500.
**
*OUTPUT, FIELD, FREQ=5
** output nodal displacements and temperatures
*NODE OUTPUT
   U, NT
** output element stresses, solution-dependent variables, total strains
*ELEMENT OUTPUT
   S,SDV,
   E
*END STEP
```

Table 9. Sample extended UMAT input data for representative linear elastic isotropic material with temperature-dependent properties.

These input records are assumed to be contained in a files named `sic-umat.dat` and are read into the ABAQUS/Standard input file by using the `*INCLUDE` keyword command.

```
** ==========================================================================================
** ==========================================================================================
** UMAT Property Data Definitions
** props(1-8):E11t,E22t,E33t,E11c,E22c,E33c,G12,G13,
** props(9-16):G23,nu12,nu13,nu23, Xt, Yt, Zt, Xc,
** props(17-24):Yc,Zc,S12,S13,S23,Eps11T,Eps22T,Eps33T,
** props(25-32):Eps11C,Eps22C,Eps33C,Gam12,Gam13,Gam23,Eps11Tmx,Eps22Tmx,
** props(33-40):Eps33Tmx,Eps11Cmx,Eps22Cmx,Eps33Cmx,Gam12mx,Gam13mx,Gam23mx,GIc,
** props(41-48):FPZ,SlimT,SlimC,SlimS,weibull(1),weibull(2),weibull(3),weibull(4),
** props(49-56):weibull(5),weibull(6),Dgrd(1),Dgrd(2),Dgrd(3),RECURS,PDA,Tref
** props(57-61):Tinit=SFT,First Temp Value,cte(1),cte(2),cte(3)
** plus 34 entries for each additional temperature value
** ==========================================================================================
** Sintered Silicon Carbide       data for     6 temperature values
*MATERIAL, NAME=UserSIC
*USER MATERIAL,  CONSTANTS=231
 1.360E+07, 1.360E+07, 1.360E+07, 1.360E+07, 1.360E+07, 1.360E+07, 5.900E+06, 5.900E+06,
 5.900E+06, 1.600E-01, 1.600E-01, 1.600E-01, 5.500E+04, 5.500E+04, 3.300E+04, 5.500E+04,
 5.500E+04, 5.000E+04, 5.500E+04, 1.920E+03, 1.920E+03, 4.044E-03, 4.044E-03, 2.426E-03,
 4.044E-03, 4.044E-03, 3.676E-03, 9.323E-03, 3.254E-04, 3.254E-04, 1.000E-01, 1.000E-01,
 1.000E-01, 1.000E-01, 1.000E-01, 1.000E-01, 1.000E-01, 1.000E-01, 1.000E-01, 3.080E+01,
 2.000E-01, 0.000E+00, 0.000E+00, 0.000E+00, 1.000E+00, 1.000E+00, 1.000E+00, 1.000E+00,
 1.000E+00, 1.000E+00, 5.000E-01, 5.000E-01, 5.000E-01, 1.000E+00, 0.000E+00, 7.000E+01,
 0.500E+03, 7.500E+01, 2.400E-06, 2.400E-06, 2.400E-06, 1.000E+03, 1.320E+04, 1.320E+04,
 1.320E+07, 1.320E+07, 1.320E+07, 1.320E+07, 5.700E+06, 5.700E+06, 5.700E+06, 1.600E-01,
 1.600E-01, 1.600E-01, 4.199E+04, 4.199E+04, 3.144E+04, 4.199E+04, 4.199E+04, 4.199E+04,
 5.500E+04, 1.920E+03, 1.920E+03, 3.181E+00, 3.181E+00, 2.382E-03, 3.181E-03, 3.181E-03,
 3.181E-03, 9.650E-03, 3.368E-04, 3.368E-04, 2.560E-06, 2.560E-06, 2.560E-06, 1.500E+03,
 1.300E+04, 1.300E+04, 1.300E+07, 1.300E+07, 1.300E+07, 1.300E+07, 5.600E+06, 5.600E+06,
 5.600E+06, 1.600E-01, 1.600E-01, 1.600E-01, 3.766E+04, 3.766E+04, 3.060E+04, 3.766E+04,
 3.766E+04, 3.766E+04, 5.500E+04, 1.920E+03, 1.920E+03, 2.897E+00, 2.897E+00, 2.354E-03,
 2.897E-03, 2.897E-03, 2.897E-03, 9.822E-03, 3.429E-04, 3.429E-04, 2.650E-06, 2.650E-06,
 2.650E-06, 2.000E+03, 1.280E+04, 1.280E+04, 1.280E+07, 1.280E+07, 1.280E+07, 1.280E+07,
 5.500E+06, 5.500E+06, 5.500E+06, 1.600E-01, 1.600E-01, 1.600E-01, 3.333E+04, 3.333E+04,
 2.975E+04, 3.333E+04, 3.333E+04, 3.333E+04, 5.500E+04, 1.920E+03, 1.920E+03, 2.604E+00,
 2.604E+00, 2.325E-03, 2.604E-03, 2.604E-03, 2.604E-03, 1.000E-02, 3.491E-04, 3.491E-04,
 2.750E-06, 2.750E-06, 2.750E-06, 2.500E+03, 1.260E+04, 1.260E+04, 1.260E+07, 1.260E+07,
 1.260E+07, 1.260E+07, 5.400E+06, 5.400E+06, 5.400E+06, 1.600E-01, 1.600E-01, 1.600E-01,
 2.900E+04, 2.900E+04, 1.623E+04, 2.900E+04, 2.900E+04, 2.900E+04, 5.500E+04, 1.920E+03,
 1.920E+03, 2.302E+00, 2.302E+00, 1.288E-03, 2.302E-03, 2.302E-03, 2.302E-03, 1.019E-02,
 3.556E-04, 3.556E-04, 2.850E-06, 2.850E-06, 2.850E-06, 2.732E+03, 1.240E+07, 1.240E+07,
 1.240E+07, 1.240E+07, 1.240E+07, 1.240E+07, 5.300E+06, 5.400E+06, 5.400E+06, 1.600E-01,
 1.600E-01, 1.600E-01, 1.450E+04, 1.450E+04, 8.000E+03, 1.450E+04, 1.450E+04, 1.450E+04,
 5.500E+04, 1.920E+03, 1.920E+03, 1.169E-03, 1.169E-03, 6.452E-04, 1.169E-03, 1.169E-03,
 1.169E-03, 1.038E-02, 3.556E-04, 3.556E-04, 2.900E-06, 2.900E-06, 2.900E-06
*DEPVAR
  20
** ==========================================================================================
** ==========================================================================================
```

REPORT DOCUMENTATION PAGE	*Form Approved OMB No. 0704-0188*

1. REPORT DATE *(DD-MM-YYYY)*	**2. REPORT TYPE**	**3. DATES COVERED** *(From - To)*
01-11-2008	Contractor Report	August 2008 - September 2008

4. TITLE AND SUBTITLE	
User-Defined Material Model for Thermo-Mechanical Progressive Failure Analysis	**5a. CONTRACT NUMBER** NNL07AA00B
	5b. GRANT NUMBER
	5c. PROGRAM ELEMENT NUMBER
6. AUTHOR(S) Knight, Norman, F., Jr.	**5d. PROJECT NUMBER**
	5e. TASK NUMBER
	5f. WORK UNIT NUMBER 510505.03.07.01.11
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) NASA Langley Research Center Hampton, VA 23681-2199	**8. PERFORMING ORGANIZATION REPORT NUMBER** General Dynamics Project No. S70509
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Washington, DC 20546-0001	**10. SPONSORING/MONITOR'S ACRONYM(S)** NASA
	11. SPONSORING/MONITORING REPORT NUMBER NASA/CR-2008-215528

12. DISTRIBUTION/AVAILABILITY STATEMENT

Unclassified - Unlimited
Subject Category 39 - Structural Mechanics
Availability: NASA CASI (301) 621-0390

13. SUPPLEMENTARY NOTES

Langley Technical Monitor: I. S. Raju

14. ABSTRACT

Previously a user-defined material model for orthotropic bimodulus materials was developed for linear and nonlinear stress analysis of composite structures using either shell or solid finite elements within a nonlinear finite element analysis tool. Extensions of this user-defined material model to thermo-mechanical progressive failure analysis are described, and the required input data are documented. The extensions include providing for temperature-dependent material properties, archival of the elastic strains, and a thermal strain calculation for materials exhibiting a stress-free temperature.

15. SUBJECT TERMS

Stress analysis, structures, material model, failure analysis, data, material properties, elastic strains

16. SECURITY CLASSIFICATION OF:			**17. LIMITATION OF ABSTRACT**	**18. NUMBER OF PAGES**	**19a. NAME OF RESPONSIBLE PERSON**
a. REPORT	**b. ABSTRACT**	**c. THIS PAGE**			STI Help Desk (email: help@sti.nasa.gov)
UU	UU	UU	UU	23	**19b. TELEPHONE NUMBER** *(Include area code)* (301) 621-0390