IN-24
97781

NASA Contractor Report 198500

Optimization of Residual Stresses in MMC's Through Process Parameter Control and the Use of Heterogeneous Compensating/Compliant Interfacial Layers

OPTCOMP2 User's Guide

Marek-Jerzy Pindera and Robert S. Salzar
University of Virginia
Charlottesville, Virginia

September 1996

Prepared for
Lewis Research Center
Under Contract NAS3–26571

National Aeronautics and
Space Administration

PREFACE

A user's guide for the computer program **OPTCOMP2** is presented in this report. This program provides a capability to optimize the fabrication or service-induced residual stresses in unidirectional metal matrix composites subjected to combined thermomechanical axisymmetric loading by altering the processing history, as well as through the microstructural design of interfacial fiber coatings. The user specifies the initial architecture of the composite and the load history, with the constituent materials being elastic, plastic, viscoplastic, or as defined by the "user-defined" constitutive model, in addition to the objective function and constraints, through a user-friendly data input interface. The optimization procedure is based on an efficient solution methodology for the inelastic response of a fiber/interface layer(s)/matrix concentric cylinder model where the interface layers can be either homogeneous or heterogeneous. The response of heterogeneous layers is modeled using Aboudi's three-dimensional *method of cells* micromechanics model. The commercial optimization package **DOT** is used for the nonlinear optimization problem. The solution methodology for the arbitrarily layered cylinder is based on the *local-global stiffness matrix formulation* and Mendelson's iterative technique of *successive elastic solutions* developed for elastoplastic boundary-value problems. The optimization algorithm employed in **DOT** is based on the *method of feasible directions.*

Notice: The **OPTCOMP2** code is being made available strictly as a research tool. Neither the authors of the code nor NASA-Lewis Research Center assume liability for application of the code beyond research needs. Any questions or related items concerning this computer code can be directed to either Professor Marek-Jerzy Pindera at the Civil Engineering & Applied Mechanics Department, University of Virginia, Charlottesville, VA 22903 (Tel: 804-924-1040, e-mail: marek@virginia.edu) or Dr. Robert S. Salzar, an NRC Fellow, at the Structural Fatigue Branch, NASA-Lewis Research center, Cleveland, OH 44135 (Tel: 216-433-3262).

Acknowledgements: The support for this work was provided by the NASA-Lewis Research Center through the contract **NAS3-26571**. The authors thank Dr. Steven M. Arnold of the NASA-Lewis Research Center, the technical monitor of this contract, for his valuable suggestions and comments in the course of this investigation and the preparation of this user's guide.

TABLE OF CONTENTS

1.0 INTRODUCTION

This user's guide provides a description of the operation and use of an efficient, computer-based algorithm for optimizing residual thermal stresses in metal matrix composites (MMC's) through the tailoring of the composite's processing history, as well as through the use of elastic and/or inelastic layers at the fiber/matrix interface. The development of the optimization algorithm **OPTCOMP2** was motivated by the need to reduce high residual stresses, and thus the potential for cracking, in advanced MMC's such as SiC/Ti that arise due to the large mismatch in the thermal expansion coefficients of the fiber and matrix phases, the lack of matrix ductility, and the high processing temperature [1]. The goal is to tailor the process used to fabricate these composites, as well as the microstructure of the interfacial region. This is accomplished by varying processing parameters such as temperature, pressure, axial load and time, and by using single or multiple interfacial layers with heterogeneous, two-phase microstructures inserted between the fibers and the surrounding matrix which act as compliant/compensating layers (Arnold et al. [2,3]).

The computer program **OPTCOMP2** enables the user to identify those processing parameters and microstructural details of the interfacial layers, herein called design variables, that optimize (i.e., minimize or maximize) residual thermal stresses or some other objective function describing the response of the composite under combined axisymmetric thermomechanical loading for the specified set of constraint variables. The definitions for the optimization terminology employed throughout this report are given below.

Objective function: An expression for the dependent variable such as a stress or strain component, or a combination of these components (e.g., strain energy density function) that is to be minimized or maximized by the optimization algorithm.

Design variable(s): The independent variable(s), such as processing parameters given in terms of temperature and external pressure or inclusion volume fraction in heterogeneous interfacial layers, used in determining an improved (optimum) design.

Constraint: A limiting value placed on a dependent variable, which is not an objective function, necessary to achieve a feasible (physically meaningful) design.

Side constraint(s): Upper and lower bounds placed on an independent design variable necessary for maintaining it within physically meaningful values.

The calculation of residual stresses is based on a micromechanics multiple concentric cylinder model, Figure 1, which consists of a fiber, an interfacial layer region and a matrix region. The fiber and interface regions may exhibit layered morphologies, with the individual

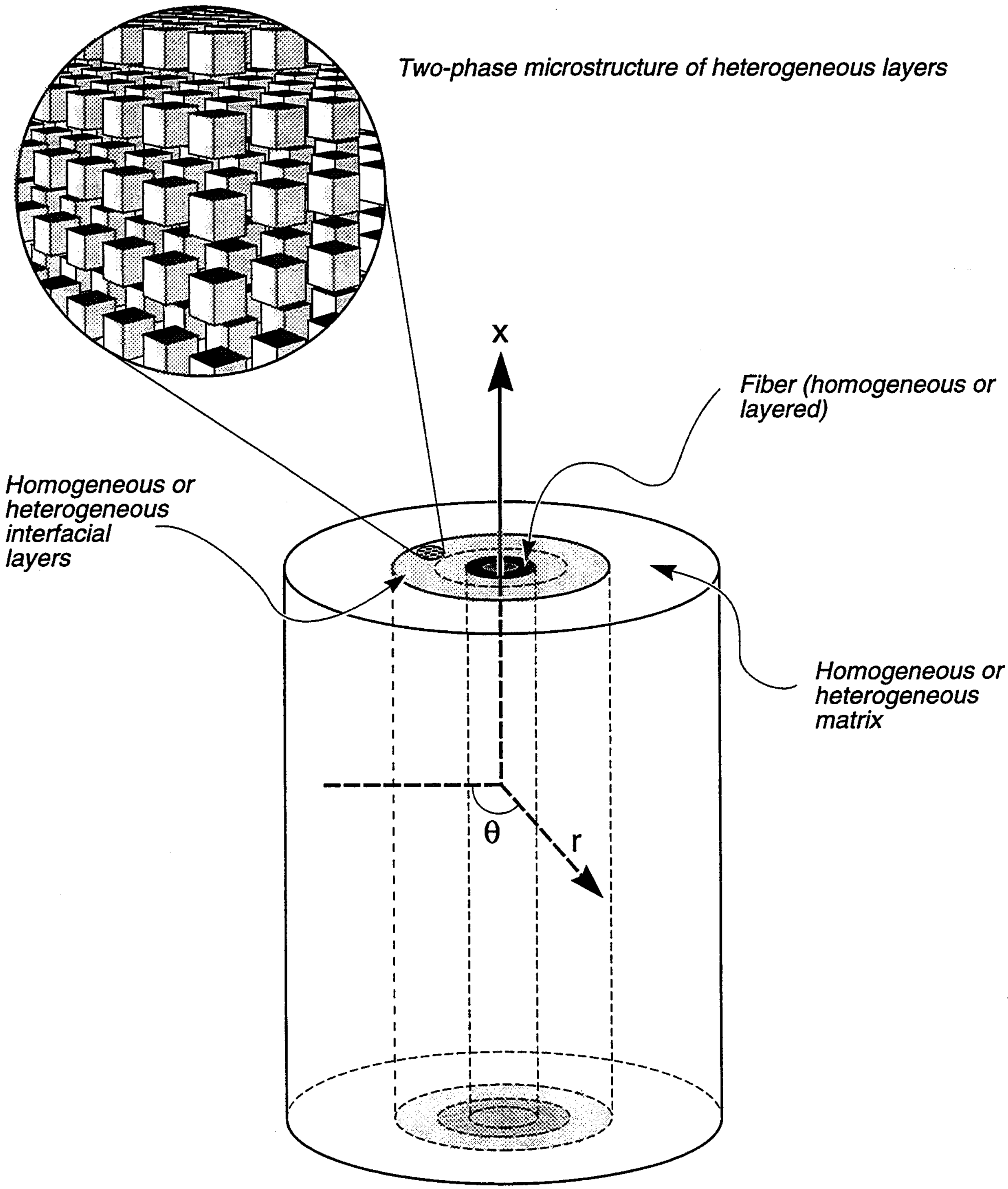

Figure 1. Multiple concentric cylinder model with heterogeneous microstructures

sublayers possessing either homogeneous or heterogeneous microstructures. The matrix region also admits an heterogeneous morphology. The heterogeneous regions are two-phase regions, consisting of an inclusion phase embedded in a matrix phase. The inclusion phase forms a triply-periodic array in the x–r–θ coordinate system, and is assumed to be sufficiently small relative to an heterogeneous layer's thickness so that the layer's macroscopic response can be calculated using a micromechanics model. It has a square cross section in the r–θ plane and an arbitrary length along the x-axis so that the inclusion aspect ratio, defined as the ratio of the inclusion's length to its witdth, is a variable parameter. Finally, with the exception of the fiber core, the fiber, interface and matrix layers exhibit inelastic, temperature-dependent behavior which is modeled using either the classical incremental plasticity theory, Bodner-Partom viscoplasticity theory or a user-defined constitutive model (see Appendix I). To simulate actual processing conditions, the multiple concentric cylinder is subjected to axisymmetric thermomechanical loading, consisting of spatially uniform temperature change, external pressure and axial stress or strain, applied simultaneously or sequentially in a monotonic or cyclic manner.

The calculation of residual stresses within each layer of the concentric cylinder assemblage subjected to the specified thermomechanical loading utilizes a novel analytical technique for the solution of axisymmetric, elastoplastic boundary-value problems recently developed by Pindera and co-workers [4-16]. This solution technique combines elements of the *local/global stiffness matrix formulation* originally developed for efficient analysis of elastic multi-layered media [17,18], and Mendelson's iterative method of *successive elastic solutions* for elastoplastic boundary-value problems [19]. The macroscopic response of the heterogeneous layers is obtained using the well-established method of cells micromechanics model developed by Aboudi [20]. The actual optimization algorithm is based on the *method of feasible directions* and utilizes the commercially-available package **DOT** [21].

In addition to the optimization capability, the user has the option of generating the response of a given composite system subject to specified axisymmetric thermomechanical loading for the chosen geometry and constituent materials. This is achieved by employing a subset of **OPTCOMP2**, called **RTSHELL2**, which is a separate program with the same menu-driven, user-friendly interface employed in the former but without the optimization subroutines and related control statements. **RTSHELL2** allows analytical characterization and evaluation of different composite material systems for applications in a wide temperature range. Specifically, it is possible to evaluate the effects of new heterogeneous coating systems for existing fibers and different processing histories on the internal stress and strain fields with a minimum of effort. An outline of the analytical solution procedure and the optimization procedure employed in **RTSHELL2** and **OPTCOMP2** is given in the supplied references.

2.0 PROGRAM DESCRIPTION

OPTCOMP2 is an executable file that is created by compiling and linking groups of subroutines that comprise the total design package for identifying an optimal processing history or an optimal microstructure of the interfacial region in unidirectional metal matrix composites. These groups of subroutines, including a brief description of their functions, are listed below.

- **shell.f**: menu-driven, user-friendly interface
- **micro.f**: analysis source code adapted for use with **dot.f**
- **dot.f**: **DOT**[1] source code consisting of **DOT1.FOR**, ..., **DOT5.FOR** files
- **user.f**: file containing user-defined constitutive model subroutine
- **objective.f**: file containing user-defined objective function subroutine
- **constraint.f**: file containing user-defined constraint function subroutine

The flow chart outlining the logical organization and execution of these subroutines within the executable **OPTCOMP2** file is given in Figure 2. In essence, the optimization algorithm is based on three modules, namely the user interface **shell.f** which provides a menu-driven, user-friendly data input environment described in Section 3.0, the analysis code **micro.f** which, in addition to generating the inelastic solution to the concentric cylinder assemblage subjected to specified loading, also controls the execution of the optimization procedure, and the optimization package **DOT** contained in the subroutine **dot.f**. The user defines the optimization problem by responding to a sequence of menu-driven instructions executed by **shell.f**. This involves specification of the concentric cylinder geometry, microstructure, materials and properties of the individual regions, processing history (and subsequent thermomechanical loading, if any), and the type of optimization problem selected (processing history or interfacial region's microstructure optimization) together with the design variables, objective function and imposed constraints. The response (i.e., properties) of the individual regions can be modeled using either the "built-in" classical incremental plasticity theory, the Bodner-Partom unified viscoplasticity theory [22,23], or a user-defined constitutive theory that has to be programmed by the user into the subroutine USERVP residing in the **user.f** file (see Appendix I). The user can select from fourteen "built-in" objective functions and eleven "built-in constraint functions. The two additional

[1]License for the **DOT** source code must be purchased separately from VMA Engineering (Vanderplaats, Miura & Associates, Inc.), 5960 Mandarin Ave., Suite F, Goleta, CA 93117. Phone: (805) 967-0058.

subroutines EXTOBJ and EXTCONST located in the **objective.f** and **constraint.f** files allow the user to construct his or her own objective function and associated constraints if so desired (see Appendix II). When the above user-defined files are employed for the given optimization problem, the user has to compile and link the subroutines residing in these files. The data provided during the problem definition stage is subsequently used to generate a solution to the defined inelastic boundary-value problem which, in turn, is used as input in the collection of optimization subroutines **dot.f**. The features and presently available capabilities of the subroutine **micro.f** and the optimization algorithm **OPTCOMP2** are summarized in Table I.

2.1 Memory Allocation

The analysis code **micro.f** contains the INCLUDE statement which utilizes the file **paraccm.v2.h** where several fundamental parameters reside which dimension the program's arrays. These parameters are:

MAXNRING	= maximum number of rings (layers) in the concentric cylinder assemblage
MAXNMT	= maximum number of materials for which properties are specified
MAXNTEMP	= maximum number of temperatures at which material properties are specified
MAXNCOLPT	= maximum number of collocation points in each layer at which field variables are calculated

MAXNRING, MAXNMT and MAXNTEMP are pre-set to 25, and MAXNCOLPT is pre-set to 250. The analysis code **micro.f** must be recompiled and relinked with the remaining subroutines if these parameters are reset by the user.

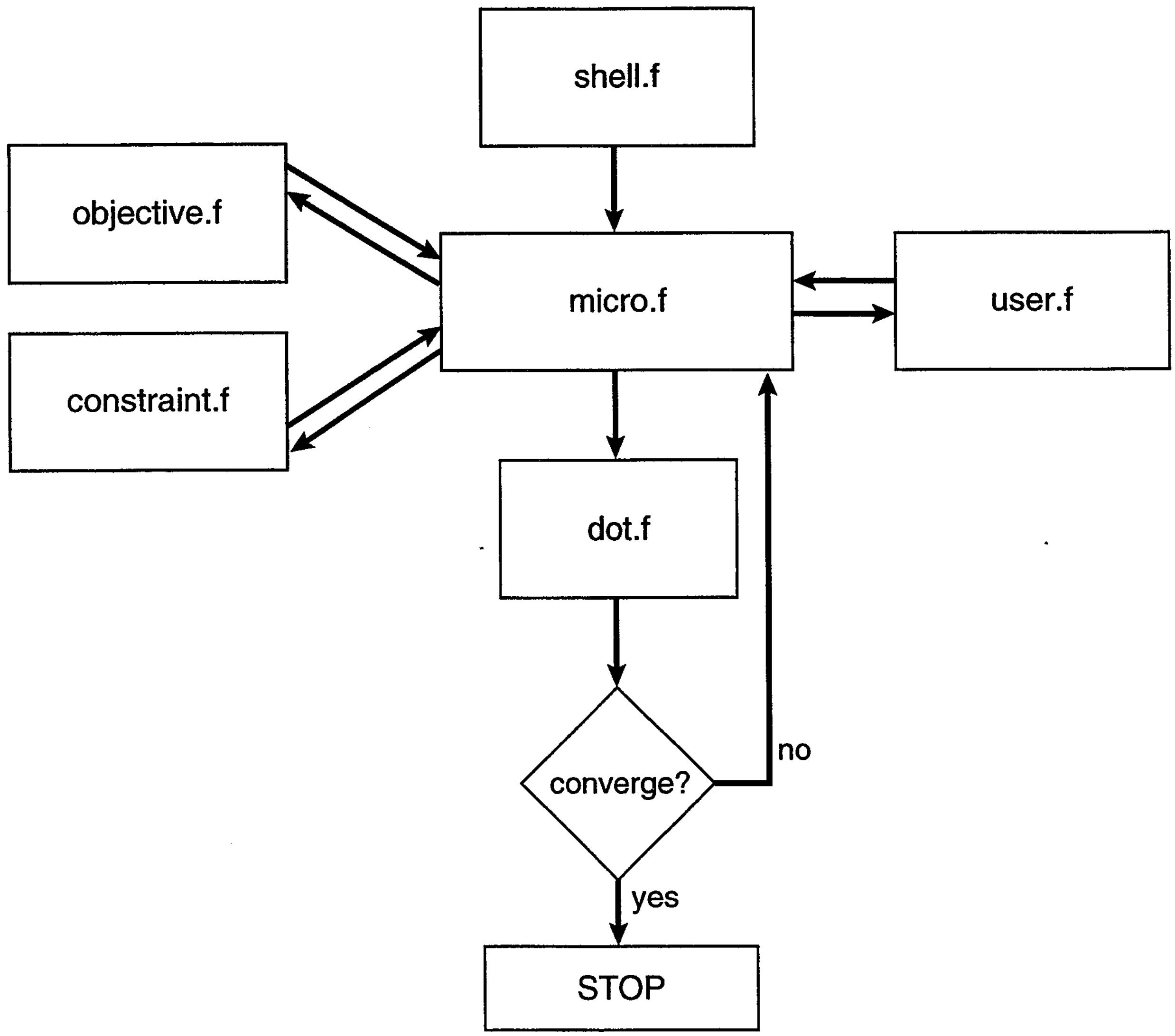

Figure 2. Flow chart for the computer program **OPTCOMP2**

Table I. Current available capabilities within **OPTCOMP2**.

Concentric Cylinder Geometry & Materials	
Constituents' Morphology	Constitutive model
fiber: layered, homogeneous or heterogeneous	core: elastic and (transversely) isotropic shells: elastic or inelastic and isotropic, or elastic and orthotropic
interfaces: layered, homogeneous or heterogeneous	elastic: (transversely) isotropic, orthotropic inelastic: isotropic
matrix: homogeneous or heterogeneous	elastic: (transversely) isotropic, orthotropic inelastic: isotropic

Loading Capabilities	
Type	Mode and History
Thermal	Monotonic or cyclic, spatially uniform ΔT
Mechanical	Monotonic or cyclic external pressure + axial tension/compression
Combined	Monotonic or cyclic ΔT + external pressure + axial tension/compression

Optimization Features	
Design variables	Interfacial layers' inclusion volume fraction, temperature history, time history, external pressure history, and axial loading history
Objective functions	Fiber radial stress; interfacial layer axial, hoop, hydrostatic and radial stress; matrix axial, hoop, hydrostatic and radial stress, and axial strain; composite axial strain; user-defined objective function
Constraints	Interfacial layer axial, hoop and radial stress; matrix axial, hoop, hydrostatic and radial stress; composite axial strain; user-defined constraint function

2.2 Input/Output Files

During execution of **OPTCOMP2**, additional files are either employed or created. These files are listed and briefly described below according to the order in which they are created or employed.

- **optcomp2.data**: internally created data file for the execution of **OPTCOMP2**
- **optcomp2.review**: review of information contained in the data file **optcomp2.data**
- **class.data**: classical plasticity material databank
- **visco.data**: Bodner-Partom viscoplasticity material databank
- **user.data**: user-defined constitutive theory material databank
- **class.int**: internally created direct-access file
- **visco.int**: internally created direct-access file
- **user.int**: internally created direct-access file
- **optcomp2.history**: an execution history of **OPTCOMP2**
- **optcomp2.out**: material properties, initial and final geometry of the concentric cylinder assemblage (including optimum microstructure of the interfacial region), initial and final (optimum) processing history, and stresses and inelastic strains
- **optcomp2.conv**: information on the convergence of the iterative solution

The data file **optcomp2.data** is created during the definition of the optimization problem through the user-friendly, menu-driven interface **shell.f**. This is done by selecting option 1 (CREATE NEW DATA FILE) from the main menu when the execution of the program is initiated by typing the command *optcomp2* as described in Section 3.0. A sequence of commands is then displayed which prompts the user to define the optimization problem. The resulting data file contains all the information needed to execute a complete optimization run. Included in this file is the information on the geometry of the concentric cylinder, microstructure and material properties of the fiber, matrix and the interfacial layer(s), processing and subsequent loading history, design variables, objective functions and constraints. This file is stored so that it can be executed either immediately after its creation, or at a later time.

At the end of the **optcomp2.data** creation process, the file **optcomp2.review** is created which summarizes the optimization problem contained in **optcomp2.data**. Unless the user has directly altered the **optcomp2.data** file using an editor, the **optcomp2.review** file will always reflect the problem stored in **optcomp2.data**.

The data files **class.data**, **visco.data** and **user.data** contain the name and properties of materials used in constructing a composite cylinder assemblage. The data file **class.data** contains materials whose inelastic response is modeled using the classical incremental plasticity theory, the data file **visco.data** contains materials whose inelastic response is modeled using the Bodner-Partom unified viscoplasticity theory, and the data file **user.data** contains materials whose inelastic response is modeled using a user-defined inelastic constitutive theory. The material properties of all materials used in constructing a given concentric cylinder assemblage, including fiber materials, must be stored in each of these files. In the case of materials which exhibit purely elastic behavior, such as many ceramic or graphite fibers, the material parameters used in describing the inelastic response of the materials in the given data file must be set appropriately so as to produce elastic response. Thus the response of the individual regions comprising a given concentric cylinder assemblage can presently be modeled using a single inelastic constitutive theory, which includes elastic behavior as a special case through appropriate adjustment of the inelastic material parameters. The three material property data files can be created and/or modified using the three alternative methods described in Section 2.3.

The files **class.int**, **visco.int** and **user.int** are automatically generated "direct-access" internal files which are read from the corresponding files with the extension ".data". These files are necessary for the execution of **OPTCOMP2**, and are re-created every time the **OPTCOMP2** program is executed. Consequently, they may be deleted between optimization runs without erasing the databanks. **However, if the files with the extension ".data" are deleted, the material databanks will be lost**. Thus it is recommended to make back-up copies of these files.

Output generated by **OPTCOMP2** upon selection of option 2 (RUN EXISTING DATA FILE) is written to three files, namely **optcomp2.history**, **optcomp2.out** and **optcomp2.conv**. The file **optcomp2.history** contains the entire history of a given optimization run that includes, at each iteration of the optimization procedure, the values of the chosen design variables, their lower and upper bounds, and the specified constraints and objective function. This information can also be written to the screen during execution of the optimization procedure at the user's discretion. The information written to the file **optcomp2.out** includes the material properties of the individual layers, followed by the initial and final geometry of the concentric cylinder assemblage and microstructure of the interfacial region, processing history, stresses and inelastic strains. Finally, the file **optcomp2.conv** contains information on the convergence of the iterative solution in the form of messages which advise the user whether or not convergence of the iterative solution has been achieved at the given optimization iteration, as explained in Sections 3.1.2 and 3.2. The user has the option to suppress these convergence messages.

2.3 Entering New Material Data

The three material property databanks residing in the files **class.data**, **visco.data** and **user.data**, supplied with the standard version of **OPTCOMP2**, contain several types of fibers and matrices, which are listed in Table II. Although these properties have been entered in SI units (i.e., MPa, °C and sec), English units can also be chosen. The material parameters for the fibers residing in each of the three data banks, which represent three different inelastic constitutive models, have been set so as to suppress the inelastic behavior. Properties of fibers with layered morphologies which exhibit inelastic behavior can also be entered into any of the three data banks if required for the given application.

Table II. Materials residing in the three data banks.

Material Data Bank	Constitutive Theory	Fiber	Matrix
class.data	Classical incremental plasticity	SiC (SCS-6) Al_2O_3 Gr	Ti-24Al-11Nb Ti-6Al-4V NiAl FeAl FeAl1 Cu
visco.data	Bodner-Partom viscoplasticity	SiC (SCS-6)	Ti-6Al-4V
user.data	Power-law creep model	SiC (SCS-6)	Ti-6Al-4V

Additional fiber and matrix materials, and their properties, can be entered into the corresponding databanks in the three ways described below.

The first way one can enter new material properties into the databanks is during the creation of the **optcomp2.data** data file, after option 1 (CREATE NEW DATA FILE) is chosen from the main menu during the execution of **OPTCOMP2**, as described in Section 3.1. During the data file creation, the user will be prompted to select the fiber/interfacial layer(s)/matrix material combination for the given problem. If the desired material for a given problem is not listed under the material selection menu, the user has the option to enter the new material interactively by selecting the appropriate option. Once this is completed, the user will automatically re-enter the material selection menu, with the newly entered material now available for selection. The material properties entered into the databanks in the manner described are stored in the respective files (i.e., **class.data**, **visco.data**, and **user.data**), and will be available for selection in subsequent optimization procedures as well.

The second, and most direct way to enter new material properties is to do so before creation of an executable data file. This is done by selecting option 3 (ENTER NEW MATERIALS INTO DATA BANKS) from the main menu when the execution of the program is initiated as described in Section 3.3. By selecting this option, the user will be directed through additional menus to the appropriate constitutive model material property databank. As the user is prompted for the material name and properties, the data is stored in the respective files. The third way of updating the material property databanks is to edit directly the files **class.data**, **visco.data** and **user.data** using a text editor as described in Section 4.4 and Appendix VI.

3.0 EXECUTING OPTCOMP2

OPTCOMP2 is executed by typing the command *optcomp2* after the unix system prompt. At this point, execution of the subroutine **shell.f** is initiated, providing the user with the menu given below as the first step in a sequence of commands:

1. CREATE NEW DATA FILE
2. RUN EXISTING DATA FILE
3. ENTER NEW MATERIALS INTO DATABANK
4. EXIT SHELL

The user chooses the appropriate option which prompts the sequence of events outlined in Figure 3. As indicated in the preceding section, option 1 creates a new data file that defines a given optimization problem. This file can be executed immediately, or stored for later use. If a file defining the optimization problem already exists (i.e., it has been constructed at an earlier time), then the user can execute it by choosing option 2. Choosing option 3 allows the user to enter new material properties into the appropriate material databanks for use at some later time in an optimization problem. The execution of **OPTCOMP2** is terminated when option 4 is selected. The sequence of commands initiated when the above options are selected is described in the following sections. Since the data input is accomplished through a menu-driven interface, only a general outline of the above options is given. Examples are provided in the following section and in the appendices, based on actual runs, that clearly illustrate the step-by-step data input which the user is interactively prompted to supply.

3.1 Option 1: Creating A New Data File

Selection of option 1 (CREATE NEW DATA FILE) initiates a sequence of input commands that define the given optimization problem in terms of: the concentric cylinder geometry, microstructure and material properties corresponding to each region (fiber, interfacial layer(s) and surrounding matrix); the processing history (and subsequent thermomechanical loading, if any) and the parameters that control the accuracy of the inelastic solution procedure for the concentric cylinder as well as the field variable (i.e., stress and strain) output; and finally the type of the optimization problem (processing history or interfacial microstructure optimization), choice of design variables, objective function and associated constraints. The sequence of input commands is logically divided into three distinct data input blocks that describe the geometry, loading history, and optimization parameters for the specified optimization problem. This information is used to create the file **optcomp2.data**. Appendices III through V provide examples that illustrate the step-by-step construction of an **optcomp2.data** file for two processing history optimization problems and one interfacial microstructure optimization problem.

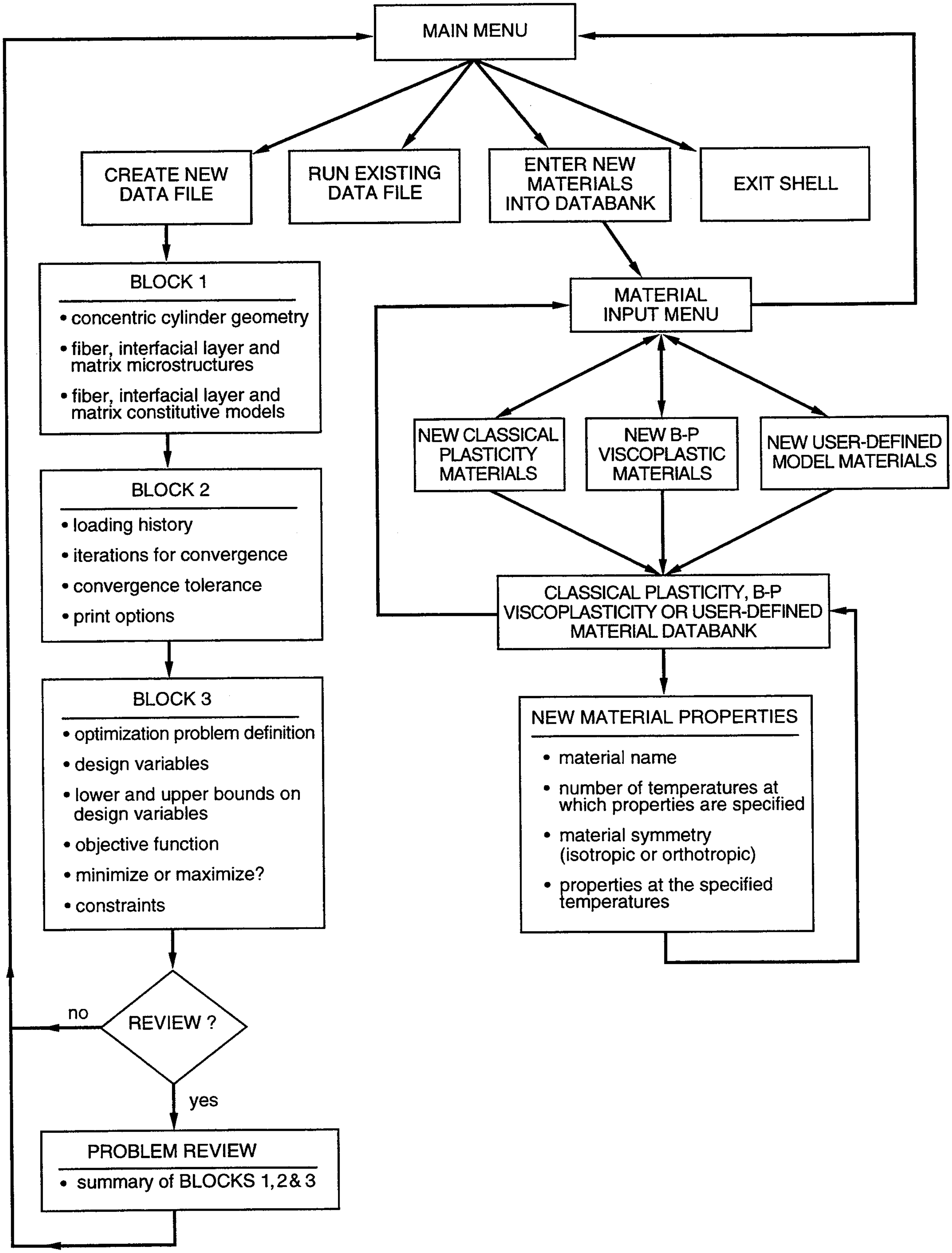

Figure 3. Flow chart for the menu-driven user interface **shell.f**

3.1.1 Block 1: geometry/microstructure/material data

First, the user specifies the concentric cylinder geometry, starting from the fiber and progressing outwards. The concentric cylinder consists of a circular fiber surrounded by a multi-layer interfacial region which, in turn, is surrounded by a matrix annulus. The fiber can be either homogeneous or layered, in order to realistically model the microstructure of certain advanced ceramic fibers (e.g., SCS-6 SiC fiber). The maximum number of layers used to model the response of the concentric cylinder assemblage is 25. The specification of the concentric cylinder geometry consists of providing the number of layers used to model the fiber and interface layer(s) and the dimensions of each of these regions. The dimensions of the fiber and interfacial layer(s) are entered in two different ways. If the fiber consists of a single core, then only the outer radius of the core is specified. In the case of a layered fiber, the outer radius of the core is specified first, followed by the actual thickness of each layer, after the number of regions used to model the fiber is provided. Then, the total fiber volume fraction is requested. For the interfacial region, the actual thickness of each of the interfacial layers, is specified. After the user specifies the number of layers required to model the fiber and interfacial regions, the outer matrix region's thickness is automatically calculated based on the fiber and interfacial layer dimensions and the fiber volume fraction. For numerical implementation reasons, the outer radius of the concentric cylinder assemblage is subsequently normalized to 1 and the dimensions of the individual regions rescaled accordingly.

The specification of the concentric cylinder geometry is followed by the specification of the microstructure of each of the fiber, interfacial layer, and matrix regions. Each of the regions can be either homogeneous or heterogeneous. Heterogeneous regions possess two-phase microstructures whose instantaneous response is calculated using the three-dimensional method of cells.

After the concentric cylinder geometry and microstructural details of the individual regions are specified and verified, the user selects one of the three constitutive models that will be employed to generate the response of each of the regions. As previously mentioned, the user can choose either the classical incremental plasticity theory, the Bodner-Partom unified viscoplasticity theory, or a user-defined constitutive theory. Subsequently, materials are selected for the fiber, interfacial layer, and matrix regions (whose properties are stored in three databanks). If a region is heterogeneous, the user specifies the materials for the inclusion and matrix phases, inclusion volume fraction and aspect ratio (i.e., inclusion's length/width). As explained previously, new materials and their properties can be entered at this stage if so desired. The program automatically sets the constitutive model for the fiber core to be elastic, suppressing the elastoplastic, viscoplastic or user-defined inelastic properties. If a layered fiber is employed, each of the fiber sublayers is treated as a different material that has to be present in the chosen databank. In entering the fiber morphology, it is important to remember that the core of the fiber must be isotropic whereas the remaining fiber sublayers may be specified as either isotropic (elastic or inelastic) or orthotropic (elastic), as summarized in Table I. The user has the option of specifying the interfacial layers and the surrounding matrix as either elastic or inelastic.

3.1.2 Block 2: applied load history

Next, the user defines the load history in terms of initial and final temperature, pressure, axial stress or axial strain, and load duration. First, the number of load segments involving simultaneous application of these quantities is specified, followed by the type of imposed axial loading (i.e., whether axial stress or axial strain is specified). This is followed by the initial values of the applied temperature, external pressure, and axial stress or axial strain for the given load segment. The specification of an arbitrary number of load segments allows application of cyclic loading. Subsequently, the duration of the load step and the number of load increments is specified by the user in order to define the size of the temperature, pressure and axial stress or strain increment, and their rates, used in generating the solution to the inelastic boundary-value problem of a multiple concentric cylinder. This is then followed by specification of the final values of temperature, pressure and axial stress or axial strain for the given load segment.

The last sequence of instructions involves specification of the parameters that control the accuracy with which the solution to the concentric cylinder assemblage is generated, and the number of points within each layer at which the field variables will be printed to the **optcomp2.out** file. Since the solution procedure is an iterative one at every step of applied loading (defined by the number of load increments), the user specifies the maximum number of iterations allowed for convergence at every load increment, together with the error tolerance imposed on the differences in the effective inelastic strain increments between successive iterations. The default values for the maximum number of iterations and the error tolerance are 10 and 0.01 (or 1%), respectively. Iteration is terminated at each load increment after the specified maximum number of iterations is reached regardless of whether the solution has converged or not, and the next load increment is applied. Intimately related to the accuracy of the solution is the number of (equally-spaced) integration points within each layer and the related number of points at which the field variables are written to the **optcomp2.file** that are specified next. The number of points within each layer at which the field variables are written to the above file must be chosen such that these points coincide with the specified integration points within the given layer. The integration points are employed to determine the inelastic strain distributions that are needed in generating the solution. Thus their number within a given layer is chosen such that inelastic strain distributions are determined with sufficient accuracy. The above options have been included in the menu to allow the user a certain amount of control over the accuracy of the solution versus the execution time for a given inelastic boundary-value problem, and thus the associated optimization problem. Information on the convergence of the iterative solution in the form of messages described in Section 3.2 can be written to the **optcomp2.conv** file if specified by the user. Finally, the user specifies whether the data recorded in the **optcomp2.history** file during the actual execution of the optimization procedure is to be simultaneously written to the screen.

The convergence of the iterative solution technique employed within **micro.f** depends on six factors, namely: 1) the constitutive model employed; 2) the size of the load increment for a given loading segment; 3) the number of iterations at a given load increment; 4) the magnitude of error that can be tolerated between successive values of the inelastic strain increments during a sequence of iterations for a given load increment; 5) the number of integration points within each layer that the program employs to calculate the inelastic strain distributions; and 6) the layer's microstructure.

When the classical incremental plasticity theory is employed to calculate the inelastic strain distributions within each layer of the concentric cylinder assemblage, the method of successive elastic solutions used in **micro.f** can be thought of as a purely spatial integration scheme for the rate-independent Prandtl-Reuss constitutive equations. Williams and Pindera [10] investigated the rates of convergence of this numerical technique for thermal loading situations when the individual layers within the concentric cylinder assemblage are homogeneous and found the method to be robust even for large loading steps (50°F increments). In general, large loading increments require small error tolerances (on the order of 0.01 or 1%) and sufficiently large numbers of iterations at each load increment (on the order of 10). For smaller temperature increments, rapid convergence was achieved with as few as 4 iterations. The convergence rate also depends on the degree of plasticity exhibited by the given material. Thus, materials with small rates of hardening and low yield stresses generally require smaller load increments and larger numbers of iterations. Further, the number of integration points used to calculate the inelastic strain distributions within each layer depends on the thickness of the layer and the nonuniformity of the inelastic strain distributions. Thin interfacial layers require fewer integration points than a thick matrix region that undergoes substantially nonuniform inelastic deformations. Finally, the convergence rate tends to be substantially slower for heterogeneous than for homogeneous layers. Heterogeneous layers may require as many as 50 iterations for convergence to be to achieved at every point within the concentric cylinder assemblage at every load increment. It should be mentioned, however, that essentially the same results (field variable distributions) are typically obtained in the presence of heterogeneous layers with, say, ten iterations as with fifty, even though convergence is not achieved at a few points within the heterogeneous layers during the thermal loading history. The nonconvergence typically occurs at elevated temperatures and can be eliminated by increasing the number of iterations. The employed iterative scheme is sufficiently rugged to produce convergence at the lower temperatures despite occurrence of nonconvergence at a few integration points at the higher temperatures.

When a rate-dependent viscoplasticity theory is employed to calculate the inelastic strain distributions, both point-wise time integration and spatial integration aspects have to be considered. In general, unified viscoplasticity theories are formulated in terms of first-order ordinary differential equations for the evolution of the inelastic strains that typically exhibit very stiff behavior. This stiff behavior requires that the time integration of these equations be carried out with great care, using a sufficiently small time increment that depends on the actual time integration scheme. A time-integration scheme coupled with the method of successive elastic solutions can be thought of as a predictor-corrector scheme, with each iteration for the given load increment providing a correction to the initial values of field variables at each point within the cylinder assemblage. In the current **OPTCOMP2**, an explicit forward Euler integration scheme, based on **a priori** known field quantities at the begining of a load increment, is employed to estimate the viscoplastic strain distributions that result from the imposed thermomechanical load increment at every point within the assemblage on the first iteration. The initial estimates of the viscoplastic strain distributions are then used in a Runge-Kutta integration scheme to generate better, and final, estimates of these distributions on the second iteration. Therefore, the iterative scheme does not provide new information on the current field quantities on the third iteration and so the number of iterations at each load increment should be set to 3 for convergence to be automatically satisfied. If the number of iterations is set to 2, non-convergence message(s) may be produced since the differences in the estimates of the viscoplastic strain distributions obtained from the first and second iterations may exceed the set error tolerance. In order to ensure convergence at the local level, and thus the global level, with the presently available viscoplastic models and the integration scheme, very small time increments must be employed as illustrated in the provided examples.

3.1.3 Block 3: specification of optimization problem

Finally, the user selects the type of optimization problem desired. The choices at this point are either to optimize the processing history (including variables such as temperature, external pressure, axial stress or strain, and load duration), or to optimize the inclusion volume fraction in the heterogeneous interface layers. If the Bodner-Partom or user-defined time-dependent constitutive model is selected for the response of the individual layers in the first block of the

optcomp2.data file's construction, the load history will include load duration if optimization of the processing history option is selected. Once the selection of the optimization problem is made, the user is prompted for the desired design variables that will be varied and their upper and lower bounds.

The selection of an objective function is next, with the user being presented with a list of possible objective goals, including the function in the user-defined subroutine EXTOBJ that resides in the file **objective.f**. After making this selection, the user has a choice to either minimize or maximize the objective function.

Finally, the selection of constraints is made from a list of pre-programmed constraints and constraints programmed in the user-defined subroutine EXTCONST that resides in the file **constraint.f**. Unlike the selection of an objective function, any number and combination of constraints can be applied to the problem.

The data creation process is terminated with an option to review the input data just entered.

3.2 Option 2: Running An Existing Data File

After the data file that defines the optimization problem has been created by executing option 1, the program returns to the main menu. At this point the user may initiate execution of the optimization procedure by selecting option 2 (RUN EXISTING DATA FILE) from the main menu. During execution of the optimization procedure, the current values of the design variables, their lower and upper bounds, and the current values of the objective function and constraints are written to the file **optcomp2.history** at every iteration on the design variables. This information is also written to the screen, if so specified, thus allowing the user to both record and monitor the optimization process.

When the search for an optimum value of the objective function is completed, the information on the values of the material properties in each layer of the concentric cylinder assemblage, and initial and final geometry and microstructure, loading history, stresses and inelastic strains is written to the file **optcomp2.out**. The material properties of each layer within the concentric cylinder assemblage, associated with the cylindrical coordinate system $x - r - \theta$, are recorded at each of the specified temperatures according to the constitutive model-dependent format:

MATERIAL # ***

TEMPERATURE = *.********

EXX	ETT	ERR			Elastic Young's moduli E_{xx}, $E_{\theta\theta}$, E_{rr}
VXR	VXT	VRT			Poisson's ratios ν_{xr}, $\nu_{x\theta}$, $\nu_{r\theta}$
ALFXX	ALFTT	ALFRR			Thermal expansion coefficients α_{xx}, $\alpha_{\theta\theta}$, α_{rr}
either					
Y	HS				(Classical incremental plasticity theory parameters)
or					
Z0	Z1	D0	N	M	(Bodner-Partom viscoplasticity theory parameters)
or					
D1	D2	D3	D4	D5	(User-defined constitutive model parameters)
D6	D7	D8	D9	D10	(User-defined constitutive model parameters)

where the variable Y is the yield stress and HS is the hardening slope based on a bilinear representation of an elastoplastic material described by the classical incremental plasticity theory.

The stress, radial displacement, and plastic strain distributions are given at the number of locations within each ring specified by the user in the second block of the data creation procedure, starting with the core and progressing outward. The program prints the following results according to the format:

RING NO.	RADIUS	STRXX	STRRR	STRTT	W
1	...	...	...	...	...
1	...	...	...	...	...
.	...	...	...	...	...
.	...	...	...	...	...
.	...	...	...	...	...
N	...	...	...	...	...
N	...	...	...	...	...

RING NO.	EPXXP	EPRRP	EPTTP	EPEFF	STREFF	SIGEFF
1	...	...	...	...	...	...
1	...	...	...	...	...	...
.	...	...	...	...	...	...
.	...	...	...	...	...	...
.	...	...	...	...	...	...
N	...	...	...	...	...	...
N	...	...	...	...	...	...

where the variables appearing in the format headers have the following meaning:

STRXX	-	axial stress $\sigma_{xx}(r)$
STRRR	-	radial stress $\sigma_{rr}(r)$
STRTT	-	circumferential stress $\sigma_{\theta\theta}(r)$
W	-	radial displacement w(r)
EPXXP	-	inelastic axial strain $\varepsilon^p_{xx}(r)$
EPRRP	-	inelastic radial strain $\varepsilon^p_{rr}(r)$
EPTTP	-	inelastic circumferential strain $\varepsilon^p_{\theta\theta}(r)$
EPEFF	-	effective plastic strain calculated from the actual plastic strain fields
STREFF	-	effective stress calculated from the actual stress fields
SIGEFF	-	effective stress calculated from the effective stress-plastic strain curve

There are two ways of calculating the effective stress when the incremental plasticity model is chosen. STREFF is calculated according to the formula: $\bar{\sigma} = \sqrt{3/2 s_{ij} s_{ij}}$, normalized to the yield stress in uniaxial tension, where s_{ij} are the deviatoric stress components, determined directly from the solution of the elastoplastic boundary-value problem for the specified concentric cylinder configuration and loading history. SIGEFF, on the other hand, is calculated from the effective stress-plastic strain curve for an elastoplastic material with bilinear hardening that defines the current yield stress. The effective plastic strain EPEFF is calculated by integrating the effective plastic strain increment, $d\bar{\varepsilon}^p = \sqrt{2/3 d\varepsilon_{ij}^p d\varepsilon_{ij}^p}$ at each point within the concentric cylinder assemblage along the entire loading history. During plastic loading, the consistency condition requires that the stress vector remain on the yield surface. Therefore, by comparing STREFF and SIGEFF during elastoplastic deformation, the user can get an idea about the quality of the solution for the chosen maximum number of iterations. Ideally, these two quantities should be the same unless elastic unloading occurs at some point during the loading cycle. When the Bodner-Partom viscoplasticity model, or a user-defined inelastic model, is used STREFF is still calculated as before, but SIGEFF has no meaning and zero is written in its place.

If specified by the user, more precise information on the convergence of the iterative solution for the inelastic boundary-value problem of the concentric cylinder assemblage is written to the file **optcomp2.conv**. If convergence has been achieved at all points within the concentric cylinder assemblage along the entire loading path for a given optimization iteration, the following message is written:

```
OPTIMIZATION ITERATION # *****
ALL POINTS REACHED CONVERGENCE
```

Alternatively, if at any point along the loading history the iterative solution does not converge for a given optimization iteration, the following message is written:

```
OPTIMIZATION ITERATION # *****

Temperature = *.********
Radial traction = *.********
Average axial stress = *.********
Non-convergence in ring number ***
```

The message informs the user that convergence has not been achieved at the indicated magnitudes of loading parameters, for the specified load increment (defined by initial and final values of the applied load and the number of load increments), the maximum number of iterations and the specified error tolerance. This is written at every occurrance of non-convergence within each iteration of the optimization algorithm. If the quality of the solution is poor, as indicated by large discrepancies between STREFF and SIGEFF, the user can either decrease the load increment or increase the maximum number of iterations, or both. Increasing the maximum number of iterations should be the first step in an attempt to obtain a convergent solution since it generally does not substantially increase the execution time relative to the option of decreasing the load increment.

Examples of the **optcomp2.history**, **optcomp2.out** and **optcomp2.conv** files are provided in Appendices III through V.

3.2.1 Termination of program execution

The execution of the program is automatically terminated, without completion of the optimization run, if the iterative scheme for the plastic strain distributions or the integration of the chosen viscoplastic constitutive equations produces non-convergent results that are characterized by large values of the effective inelastic strain increment, $d\bar{\varepsilon}^{in} = \sqrt{2/3 d\varepsilon_{ij}^{in} d\varepsilon_{ij}^{in}}$, at any given load increment. In particular, execution will be terminated if the effective inelastic strain increment at any point within the concentric cylinder assemblage during the solution procedure exceeds 20% or 0.2. Such large increments typically indicate impending loss of convergence during the integration of the viscoplastic constitutive equations. In this case, a smaller load increment must be chosen. The following message is written to the **optcomp2.out** output file in this instance:

EXECUTION STOPPED, depeff > 0.2 (20%)

3.3 Option 3: Entering New Materials Into The Databank

The user can update the material property databanks contained in the three files **class.data, visco.data** and **user.data** by selecting option 3 (ENTER NEW MATERIALS INTO DATABANK) from the main menu. This option initiates a sequence of menu-driven commands by first providing the user with the following material input selection menu:

1. ENTER NEW CLASSICAL PLASTICITY MATERIALS
2. ENTER NEW VISCOPLASTIC MATERIALS
3. ENTER NEW USER-DEFINED MODEL MATERIALS
4. RETURN TO MAIN MENU

Upon selection of options 1 through 3, the user is presented with the existing materials residing in the chosen databank and is prompted to supply: the units (either SI or English) in which the material properties will be entered; name of the new material; number of temperatures at which properties of this material will be specified; material symmetry type (i.e., whether the material is isotropic, transversely isotropic or orthotropic); and finally the temperatures and the corresponding material properties (in either ascending or descending order). The user first specifies the elastic material parameters, namely the Young's modulus, Poisson's ratio, and instantaneous (tangential) thermal expansion coefficient, followed by inelastic parameters which depend on the chosen inelastic constitutive model. If the incremental plasticity theory is chosen, the user specifies yield stress in simple tension and hardening slope (based on a bilinear stress-strain representation of the elastoplastic behavior). In the case of the Bodner-Partom viscoplasticity theory, five parameters are specified, namely Z_0, Z_1, D_0, N and M, while for the user-defined constitutive model up to ten parameters may be entered, namely D_1, ..., D_{10}. Section 7.1.3 of Appendix I provides the general functional form for the user-defined constitutive model, together with an illustration of how these parameters are used in a specific model. The material parameters entered at the specified number of temperatures are subsequently re-calculated at ten equally-spaced temperatures in the interval defined by the highest and the lowest temperature using cubic splines, and subsequently written to one of the three data files.

If the user specifies the given material as isotropic, then only one set of the above material properties is specified at each temperature. For an elastic isotropic material, the values of the parameters that describe the material's inelastic response within the chosen constitutive model should be set so as to produce purely elastic response. Thus if the classical plasticity theory option is selected, the yield stress should be set to a very large value (e.g., 10^6 msi), and the hardening slope should be equal to the Young's modulus. Similarly, for the Bodner-Partom viscoplasticity option, the parameter D_0 is set to zero and the remaining parameters set to 1. At this time, only elastic transversely isotropic and orthotropic materials can be specified due to lack of generally accepted inelastic constitutive theories for anisotropic materials. If the material is specified as either transversely isotropic or orthotropic, three sets of elastic parameters consistent with the applied axisymmetric loading in the $x - r - \theta$ coordinate system must be entered at each temperature due to the directional nature of such materials. The user is not prompted to enter the inelastic parameters (such as the yield stress and the hardening slope in the case of classical plasticity theory) for transversely isotropic and orthotropic materials. These quantities are automatically set to pre-assigned values within the datafile which produce purely elastic response. In the case of the classical plasticity theory, for instance, the yield stress is set to a very large number (10^{99} psi or Pa), whereas for the hardening slope the value of the elastic Young's modulus is entered. These numbers are required to be present in the material databanks due to the logical structure of the search algorithm used in identifying the available materials. Appendix VI provides an example that illustrates the step-by-step construction of a material databank.

4.0 ILLUSTRATIONS

Appendices III through V present examples that illustrate the creation of the **optcomp2.data** files for three optimization problems involving time-independent and time-dependent optimization of the processing history, and microstructural optimization of the fiber/matrix interfacial region, and subsequent execution of these files together with the results of the optimization procedure. Appendix VI illustrates the addition to the **class.data** databank of a material (copper) which is modeled using the classical incremental plasticity theory with isotropic hardening. These examples are described in more detail in this section.

4.1 Example 1: Time-Independent Process History Optimization

This example illustrates the construction of the **optcomp2.data** file, its execution, and the results of the optimization process for the problem of a SiC/Ti composite cylinder, with no distinct interfacial layers, subjected to a fabrication cool down from 815°C to 24°C. The aim is to determine an optimum processing history involving application of external pressure during the fabrication cool down that minimizes the hoop stress at the fiber/matrix interface. The design variables are the external pressures at the beginning and end points of the intervals into which the processing history has been divided, Figure 4.

4.1.1 Construction of the optcomp2.data file

Section 7.3.1 of Appendix III illustrates the construction of the **optcomp2.data** file when option 1 is selected from the main menu of **OPTCOMP2**. The first block defines the concentric cylinder geometry, microstructure of the individual layers, choice of the constitutive model for the elastic/inelastic response of these layers and the materials residing in each layer. The concentric cylinder geometry consists of a fiber core, surrounded by a single interfacial layer, that, in turn, is embedded in a homogeneous matrix. The elastic properties of the fiber core are those of the SiC SCS-6 fiber. The properties of the homogeneous interfacial layer and the homogeneous matrix are those of the Ti-24Al-11Nb alloy so that the fiber is effectively embedded in a homogeneous matrix with no distinct interfacial layers at the fiber/matrix interface. The classical incremental plasticity theory is employed to model the inelastic response of the matrix region.

The second block defines the initial loading history for the optimization problem. The loading involves cool down from 815°C to 24°C with a non-zero radial pressure and no axial stress, as shown in Figure 4. The thermal loading history is divided into four intervals within which temperature and radial pressure are varied simultaneously. Each interval is divided into 500 increments. The maximum number of iterations allowed for convergence at each thermal load

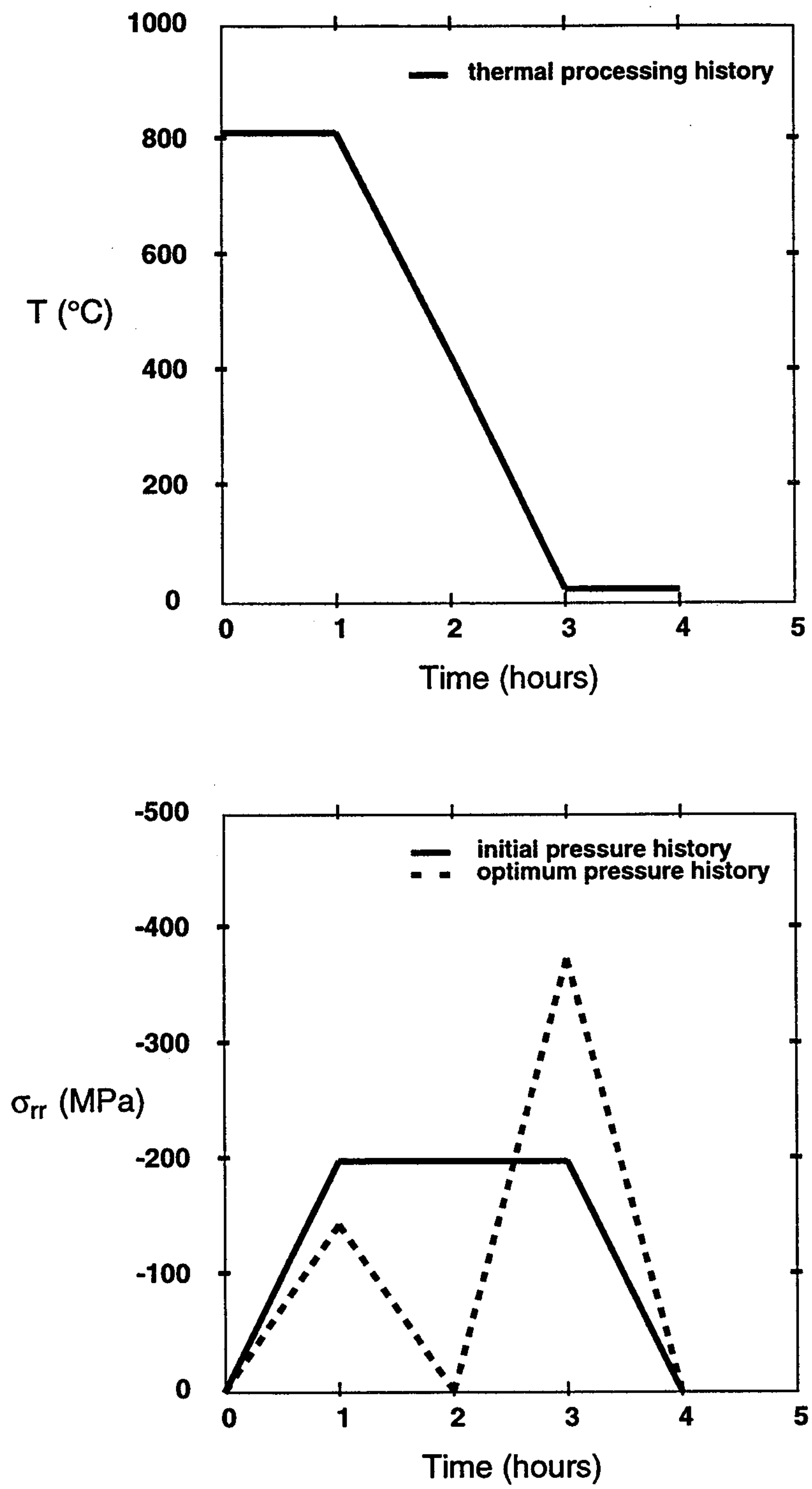

Figure 4. Initial and optimum processing history in Example 1.

increment is changed from the default value of 10 to 15, and the default value 0.01 for the error tolerance is used. Finally, information on the convergence of the iterative elastoplastic solution is written to the file **optcomp2.conv**, and the data recorded in the file **optcomp2.history** during the actual execution of the optimization procedure is simultaneously written to the screen.

The third and final block defines the optimization problem. In defining the optimization problem, optimization of processing history option was chosen involving pressure as the design variable at the five stations that divide the processing history into the four intervals. The lower and upper bounds on these design variables are 0 MPa at the beginning and end points of the processing history, -200 and 0 MPa at the second station and -350 and 0 MPa at the third and fourth stations. The optimization problem involves minimization of the hoop stress at the fiber/matrix interface. No constraints are placed on any of the field variables in the provided constraint list.

The formulated optimization problem is subsequently reviewed at the end of the data creation process and then the program is exited upon selecting option 4 from the main menu.

4.1.2 Execution of the optcomp2.data file

The **optcomp2.data** file constructed in the preceding step is executed when option 2 is selected from the main menu of **OPTCOMP2** after initiating the program by typing the command *optcomp2*, as illustrated in Section 7.3.2 of Appendix III. The information written to the **optcomp2.history** file, as it is also written to the screen during the actual execution, provides a permanent record of the optimization process. This file contains the definition of the design variables (in this case X1 through X5), followed by the current values of these design variables relative to their lower and upper bounds, together with the value of the objective function. For the given example, 18 iterations were necessary to find optimum values of the external pressure at the three inner points of the processing history. For this optimization problem, the optimum external pressure history that minimizes the fiber/matrix interfacial hoop stress is included in Figure 4.

4.1.3 Results of the time-independent process history optimization

The optimum values of the design variables are written to the **optcomp2.out** file that is provided in Section 7.3.3 of Appendix III. This file also contains additional information that may be useful to the designer. The material properties of the individual layers (when homogeneous layers are specified) or their constituents (when heterogeneous layers are specified) are given first at the ten temperatures. Provided next is the initial concentric cylinder configuration showing the makeup and the microstructure of the individual layers. This is followed by the initial loading history and the resulting stress and inelastic strain distributions based on the initial design variables. The next block of data contains information on the final concentric cylinder configuration and microstructure (provided for those cases when the microstructure of the interfacial region is a design variable), the final loading history (provided for processing history optimization problems) and the resulting stress and inelastic strain distributions based on the optimum design variables. Figure 5 presents a comparison of the initial and final distributions of the hoop stress on the matrix side of the fiber/matrix interface based on the initial and optimum processing histories. Included in the figure for comparison is the hoop stress distribution obtained when the external pressure is zero. Substantial reduction in the hoop stress distribution is obtained when the external pressure is applied in the manner illustrated.

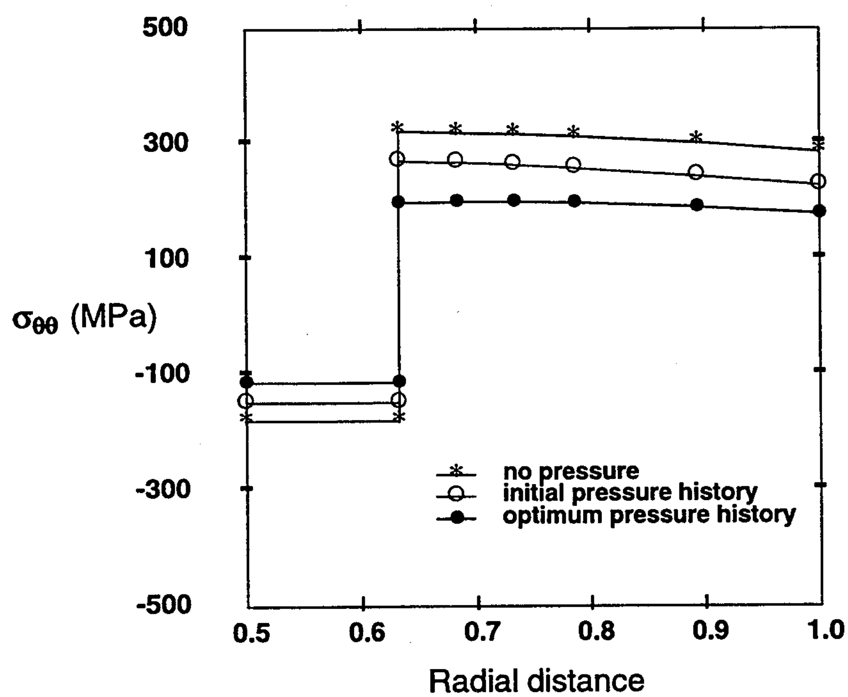

Figure 5. Initial and final $\sigma_{\theta\theta}(r)$ distributions for the time-independent processing history optimization.

Examining the stress and inelastic strain distributions based on the initial and optimum values of the design variables, one observes that the values of the effective stress, STREFF and SIGEFF, calculated using the two methods discussed in Section 3.2 are different in the matrix region, with the STREFF magnitudes being less than SIGEFF. These differences indicate that the effective stress vector does not remain on the yield surface at each point within the matrix region, suggesting that unloading has occurred at some point during the processing history. This occurs when the rate of strain hardening in the matrix region exceeds the material's capability to load plastically. The differences are greater for the optimum processing history than for the initial history. This suggests that more unloading occurs during the optimum processing history, thereby resulting in lower hoop stress distribution thoughout the matrix phase. It should be mentioned that in the absence of external pressure the values of STREFF and SIGEFF are identical throughout the entire matrix region at the end of the cool down from 815°C to 24°C. This indicates that the matrix phase undergoes continuous plastic deformation during the fabrication cool down, thereby producing a substantially higher hoop stress distribution relative to that obtained in the presence of the applied external pressure.

The information about the convergence of the iterative elastoplastic solution written to the file **optcomp2.conv**, included in Section 7.3.3 of Appendix III, indicates that convergence was achieved at all points within the concentric cylinder assemblage along the entire loading history at every optimization iteration.

4.2 Example 2: Time-Dependent Process History Optimization

This example illustrates the construction of the **optcomp2.data** file, its execution, and the results of the optimization process for the problem of a SiC/Ti composite cylinder subjected to a cool down from 900°C to 21°C. The aim is to determine the optimal rate at which the composite is cooled from the consolidation temperature in order to minimize the hoop stress at the fiber/matrix interface. The design variable is the time duration of the fabrication cool down that controls the cooling rate, subject to the side constraints that ensure that the fabrication cool down takes place within a certain time interval.

4.2.1 Construction and execution of the optcomp2.data file

Section 7.4.1 of Appendix IV illustrates the construction and execution of the **optcomp2.data** file when option 1 is selected from the main menu of **OPTCOMP2**. The concentric cylinder geometry consists of a fiber core, surrounded by a single interfacial layer, that, in turn, is embedded in a homogeneous matrix. The elastic properties of the fiber core are those of the SiC fiber. The properties of the homogeneous interfacial layer and the homogeneous matrix are those of the Ti-6Al-4V alloy. A power-law creep model described in Appendix I is employed to model the inelastic response of the matrix region. The above information is specified in **Block 1** of the data input.

The loading specified in **Block 2** of the data input involves cool down from 900°C to 21°C with no radial pressure nor axial stress applied. The initial duration of this cool down is 0.3822 hours. This corresponds to a cooling rate of 2300°C/hr. The thermal history is divided into 10,000 increments. Since a rate-dependent constitutive model is employed for the individual constituents, the maximum number of iterations allowed for convergence at each thermal load increment is changed from the default value of 10 to 3, as discussed in Section 3.1.2. The default value 0.01 is used for the error tolerance. Finally, information on the convergence of the iterative inelastic solution is written to the file **optcomp2.conv**, and the data recorded in the file **optcomp2.history** during the actual execution of the optimization procedure is simultaneously written to the screen.

In defining the optimization problem, **Block 3** of the data input, optimization of processing history option was chosen involving time duration of the fabrication cool down as the design variable. The lower and upper bounds on the design variable are 0.3822 and 38.22 hours, which correspond to cooling rates of 2300°C/hr and 23°C/hr, respectively. The optimization problem involves minimization of the hoop stress at the fiber/matrix interface. No constraints are placed on any of the field variables in the provided constraint list.

At the end of the data creation process, the formulated optimization problem is not reviewed and the constructed **optcomp2.data** file is executed when option 2 is selected from the main menu of **OPTCOMP2**. The information written to the **optcomp2.history** file, as it is also written to the screen during the actual execution illustrated in Section 7.4.1 of Appendix IV, provides a permanent record of the optimization process. This file contains the definition of the design variable (in this case X1), followed by the current value of this design variable relative to its lower and upper bounds, together with the value of the objective function. For the given example, 12 iterations were necessary to find an optimum fabrication cool down time duration of 38.22 hours for the chosen constraints on the design variable. This time duration produces a cooling rate of 23°C/hr. It should also be noted that this time duration corresponds to the upper bound on the processing time, and thus is not a global optimum. As easily verified, increasing the upper bound on the design variable will produce an optimum solution that coincides with this bound.

4.2.2 Results of the time-dependent process history optimization

The results of the optimization process include the material properties of each of the regions within the concentric cylinder assemblage, the initial and final geometry and microstructure of the concentric cylinder assemblage and the processing history with the optimum values of the design variables, and the corresponding stress and inelastic strain distributions. These are recorded in the **optcomp2.out** file provided in Section 7.4.2 of Appendix IV. Figure 6 presents a comparison of the initial and final distributions of the hoop stress on the matrix side of the fiber/matrix interface based on the initial and optimum processing histories. Substantial reduction in the hoop stress distribution is obtained when the matrix is allowed to relax due to the increase in the fabrication time or decrease in the cooling rate. Similar results were obtained by Brayshaw and Pindera [24] using strictly the two-dimensional version of the method of cells to model the unidirectional SiC/Ti-6Al-4V composite

The information about the convergence of the iterative inelastic solution written to the file **optcomp2.conv**, included in Section 7.4.2 of Appendix IV, indicates that convergence was achieved at all points within the concentric cylinder assemblage along the entire loading history at every optimization iteration.

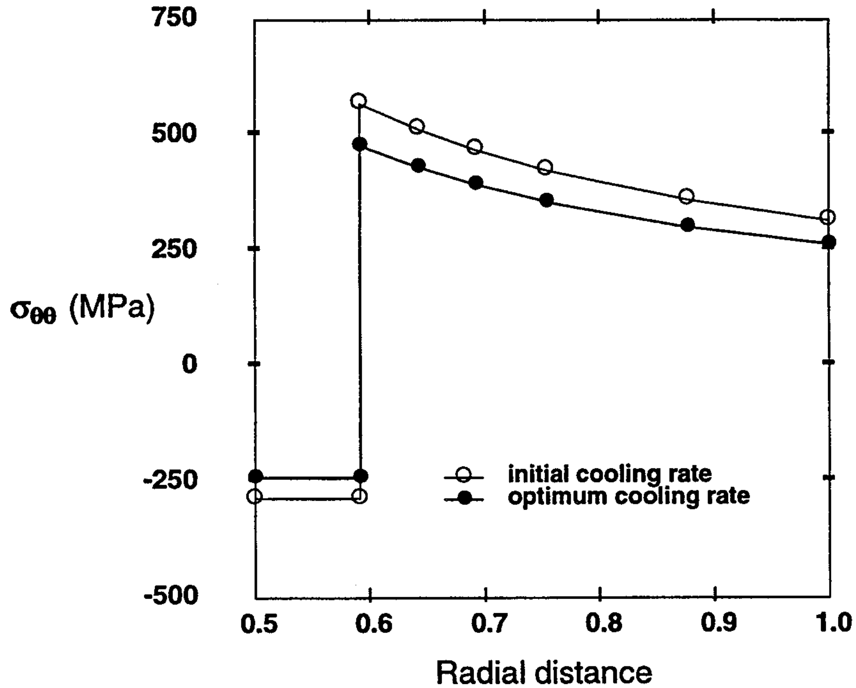

Figure 6. Initial and final $\sigma_{\theta\theta}(r)$ distributions for the time-dependent processing history optimization.

4.3 Example 3: Plastic Strain Optimization Using Graded Interfaces

This example illustrates the construction of the **optcomp2.data** file, its execution, and the results of the optimization process for the problem of a SiC/TiAl composite cylinder with three heterogeneous interfacial layers subjected to a cool down from 815°C to 24°C. The interfacial layers are composed of a matrix phase with embedded cubical inclusions. The aim is to determine the volume fraction of the inclusion phase in each interfacial layer that will produce uniform effective plastic strain distribution in the entire interfacial region under the constraint that the average inclusion volume fraction in the interfacial three-layer region remain fixed.

4.3.1 Construction and execution of the optcomp2.data file

Section 7.5.1 of Appendix V illustrates the construction and execution of the **optcomp2.data** file when option 1 is selected from the main menu of **OPTCOMP2**. The concentric cylinder geometry consists of a fiber core, surrounded by three interfacial layers that, in turn, are embedded in a homogeneous matrix. The elastic properties of the fiber core are those of the SiC SCS-6 fiber. The interfacial layers are composed of a copper matrix with embedded cubical graphite inclusions (i.e., inclusion aspect ratio is 1), with an initial inclusion volume fraction of 0.10 in each layer. The properties of the homogeneous matrix are those of the Ti-24Al-11Nb alloy. The classical incremental plasticity theory is employed to model the inelastic response of the individual layers. The above information is specified in **Block 1** of the data input.

The loading specified in **Block 2** of the data input involves cool down from 815°C to 24°C without any radial pressure or axial stress. The thermal loading segment is divided into 791 increments so that the temperature change per increment is 1.0°C. The maximum number of iterations allowed for convergence at each thermal load increment is changed from the default value of 10 to 50 due to the presence of heterogeneous interfacial layers, and the default value 0.01 for the error tolerance is used. Finally, information on the convergence of the iterative elastoplastic solution is written to the file **optcomp2.conv**, and the data recorded in the file **optcomp2.history** during the actual execution of the optimization procedure is simultaneously written to the screen.

In defining the optimization problem in **Block 3** of the data input, optimization of interfacial layers' microstructure was chosen. The design variables in this example are the inclusion volume fractions in each of the three interfacial layers, producing a total of three design variables. The lower and upper bounds on these design variables are 0.0 and 0.3 subject to the constraint that the average inclusion volume fraction be 0.1. The optimization problem involves minimization of the deviation of the effective plastic strains in the middle of each interfacial layer from the average value. The constraint and objective functions are user-defined functions that have been programmed in the subroutines EXTCONST and EXTOBJ given in Appendix II.

At the end of the data creation process, the formulated optimization problem is reviewed and the constructed **optcomp2.data** file is executed when option 2 is selected from the main menu of **OPTCOMP2**. The information written to the **optcomp2.history** file, as it is also written to the screen during the actual execution illustrated in Section 7.5.1 of Appendix V, provides a permanent record of the optimization process. This file contains the definition of the design variables (in this case X1 through X3), followed by the current values of these design variables relative to their lower and upper bounds, together with the value of the objective function and the specified constraint at each iteration (two in this case since an equality constraint is required). For the given example, 50 iterations were necessary to find optimum values of the inclusion volume fractions for the three interfacial layers. For this optimization problem, the optimum inclusion volume fractions that produce a nearly uniform effective plastic strain distribution in the interfacial region are 0.146, 0.101 and 0.054, starting from the innermost layer and progressing outward. The average value of the effective plastic strain is approximately 0.0165 or 1.65%.

4.3.2 Results of the plastic strain optimization

The results of the optimization process recorded in the **optcomp2.out** file provided in Section 7.5.2 of Appendix V include the material properties of the homogeneous fiber, constituent phases in the three heterogeneous interfacial layers, and homogeneous matrix, the initial and final geometry and microstructure of the concentric cylinder assemblage with the optimum values of the design variables (inclusion phase volume content), the processing history, and the corresponding stress and inelastic strain distributions.

Figure 7 presents a comparison between the initial and final effective plastic strain distributions $\bar{\varepsilon}^p$ based on the initial and optimum values of the design variables. Included in the figure for comparison is the effective plastic strain distribution in the same concentric cylinder with the interfacial region composed entirely of the copper matrix. The use of particulate reinforcement in the ductile copper matrix lowers the effective plastic strain in the interfacial region. Further, the optimum inclusion distribution in the heterogeneous interfacial layers produces a much more uniform effective plastic strain distribution relative to that when the inclusion phase content is uniform throughout the entire interfacial region, facilitating shake down during subsequent cyclic loading. The results presented in Figure 7 are similar to those reported by Pindera et al. [11] that were generated using a trial-and-error procedure.

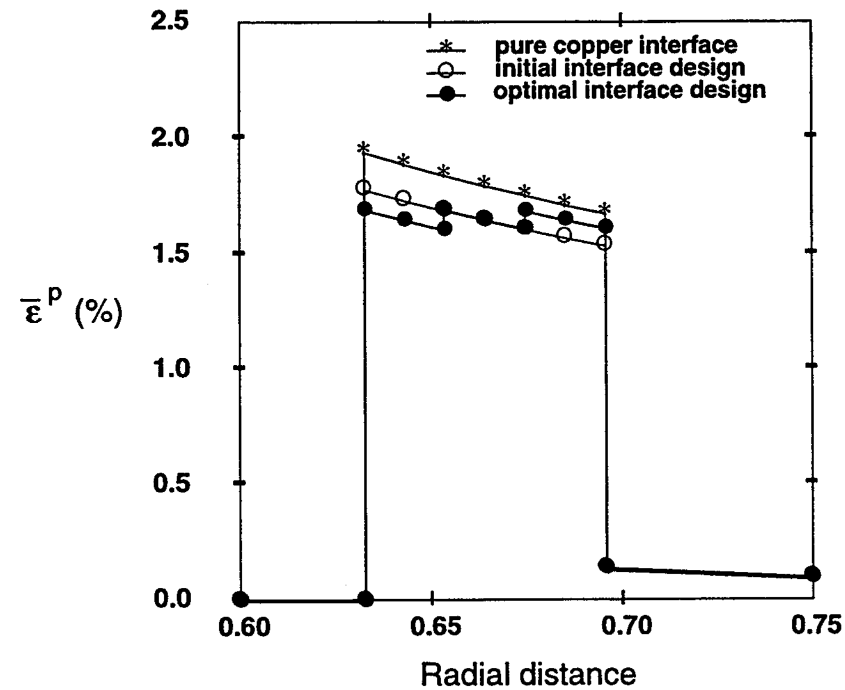

Figure 7. Initial and final $\bar{\varepsilon}^p(r)$ distributions for the interfacial plastic strain optimization problem.

Examining the stress and inelastic strain distributions based on the initial and optimum values of the design variables, one observes that the values of the effective stress, STREFF and SIGEFF, calculated using the two methods discussed in Section 3.2 are the same in the matrix region. This indicates that the effective stress vector remains on the yield surface at each point within the matrix region. The corresponding values for SIGEFF in the interfacial layers are not shown since a micromechanics model has been employed to calculate the instantaneous response of the heterogeneous interfacial layers.

The information about the convergence of the iterative elastoplastic solution written to the file **optcomp2.conv**, included in Section 7.5.2 of Appendix V, indicates that convergence was achieved at all points within the concentric cylinder assemblage along the entire loading history at most of the optimization iterations. Although convergence was not achieved at the following optimization iterations: 6, 18, 19, 20, 21, 24, 34, 36, 37, 43 and 45, this occurred at only one temperature along the entire fabrication cool down history in either the second or the third interfacial layer. This nonconvergence of the solution can be eliminated by further increasing the number of iterations employed in solving the boundary-value problem of the concentric cylinder, which is specified in **Block 2** of the data input immediately after specification of the load history. However, it should be mentioned that virtually the same results have been obtained using 10 iterations in the solution procedure at each load increment as with 50 iterations, despite a substantially greater number of non-convergence messages in the former case. Therefore, increasing the number of iterations further will eliminate the occurrence of non-convergence without improving the final solution's accuracy. The non-convergence occurrences typically take place at higher temperatures, and have negligible effect on the resulting stress and strain distributions after cool down since very stringent convergence conditions have been implemented in the iterative solution procedure, requiring convergence to occur simultaneously at every integration point in all layers where plastic deformation occurs. This indicates that the iterative scheme used in the solution procedure is sufficiently robust to produce convergent results at the lower temperatures despite occurrence of non-convergence at a few integration points at the higher temperatures.

4.4 Example 4: Construction of A Material Property Databank

As mentioned in Section 2.3, the standard version of **OPTCOMP2** contains three types of databanks with properties of materials that are modeled using either the classical incremental plasticity theory, the Bodner-Partom unified viscoplasticity theory or a user-defined inelastic constitutive model (power-law creep model in the present case). Each of these databanks includes several different types of matrix materials and also fiber materials (modeled as elastic with the appropriate inelastic parameters set accordingly) that are given in Table II. The material properties of these systems that were used in constructing the three sets of databanks are listed in Appendix I.

The example presented herein illustrates how the material properties for Cu matrix were added to the **class.data** databank file. Appendix VI, Section 7.6, illustrates the addition of Cu to the material property databank when option 3 is selected from the main menu of **OPTCOMP2.** The properties of the Cu matrix are entered at the given six temperatures (see Appendix I) and are subsequently re-evaluated at ten equally-spaced temperatures in the range 24°C - 815°C using cubic splines. The properties re-evaluated at the ten temperatures are then written to the **class.data** file according to the format illustrated in Appendix VI, Section 7.6.1. A similar procedure is employed to enter materials properties into the **visco.data** and **user.data** databanks. These files can be subsequently updated by entering additional materials using either option 3 from the main menu, or directly entering the properties into the files using a text editor according to the indicated format.

5.0 RTSHELL2: A SUBSET OF OPTCOMP2

The program **RTSHELL2** is a separate program with the same analytical capabilities as **OPTCOMP2**, but without the optimization option. This program facilitates efficient characterization and evaluation of different metal matrix unidirectional composites subjected to combined axisymmetric thermomechanical loading in the presence of different fiber and interfacial layer architectures. The program employs the same material property data banks as **OPTCOMP2**, and is driven by the same user-friendly interface, with **Block 3** that defines the optimization problem deleted. The user-friendly interface is directly embedded in **RTSHELL2**, unlike **OPTCOMP2** where it is located separately in the file **shell.f**. The construction of the input file **rtshell2.data** is carried out in the same manner as the construction of the **optcomp2.data** file, while the output produced by **RTSHELL2** is written to the **rtshell2.out** file. The executable version of **RTSHELL2** is obtained by compiling **rtshell2.f** and the user-defined constitutive model subroutine **user.f** separately, and linking them together.

Appendix VII, Section 7.7.1, provides an illustration of the construction and execution of a date file for the concentric cylinder geometry, materials and loading outlined in Example 3. The results written to the **rtshell2.out** file, shown in Section 7.7.2, are exactly the same as the initial results in Example 3 (Appendix V, Section 7.5.2).

6.0 REFERENCES

1. Larsen, J. M., Williams, K. A., Balsone, S. J. and Stucke, M. A., "Titanium Aluminides for Aerospace Applications," in **High Temperature Aluminides and Intermetallics**, C. T. Liu et al., Eds., TMS/ASM International, UAS, 1990.

2. Arnold, S. M., Arya, V. K. and Melis, M. E., "Elastic/Plastic Analyses of Advanced Composites Investigating the Use of the Compliant Layer Concept in reducing Residual Stresses Resulting from Processing," **NASA Technical Memorandum 103204**, Lewis Research Center, 1990.

3. Arnold, S. M. and Wilt, T. E., "Influence of Engineered Interfaces on Residual Stresses and Mechanical response in Metal Matrix Composites," **NASA Technical Memorandum 103204**, Lewis Research Center, 1992.

4. Pindera, M-J., Freed, A. D. and Arnold, S. M., "Effects of Fiber and Interfacial Layer Architectures on the Thermoplastic Response of Metal Matrix Composites," **NASA Technical Memorandum 105802**, Lewis Research Center, 1992.

5. Pindera, M-J., Williams, T. O., Salzar, R. S. and Arnold, S. M., "Optimization of Residual Stresses in Metal Matrix Composites Using the Multiple Concentric Cylinder Model," in **Proc. HITEMP Review 1992: NASA Conference Publication 10104**, Vol. 2, pp. 34-1:34-21, 1992.

6. Williams, T. O., Arnold, S. M. and Pindera, M-J., "Effectiveness of Graded Interfacial Layers in Reducing Residual Stresses in Titanium Matrix Composites," in **Residual Stresses in Composites (Measurement, Modeling & Effects on Thermo-Mechanical Behavior)**, E. V. Barrera and I. Dutta, Eds., pp. 187 - 204. The Minerals, Metals & Materials Society, 1993.

7. Pindera, M-J., Salzar, R. S. and Williams, T. O., "An Analytical/Numerical Correlation Study of the Multiple Concentric Cylinder Model for the Thermoplastic Response of Metal Matrix Composites," **NASA Contractor Report 191142**, NASA Lewis Research Center, Cleveland, OH. May 1993.

8. Pindera, M-J., Freed, A. D. and Arnold, S. M., "Effects of Fiber and Interfacial Layer Morphologies on the Thermoplastic Response of Metal Matrix Composites", **Int. J. Solids Structures**, Vol. 30, No. 9, 1993, pp. 1213-1238.

9. Pindera, M-J., Salzar, R. S. and Williams, T. O., "An Evaluation of a New Approach for the Thermoplastic Response of Metal Matrix Composites," **Composites Engineering**, Vol. 3, No. 12, 1993, pp. 1185-1201.

10. Williams, T. O. and Pindera, M-J., "Convergence Rates of the Method of Successive Elastic Solutions in Thermoplastic Problems of a Layered Concentric Cylinder," in **Engineering, Construction, and Operations in Space IV**, Rodney G. Galloway and Stanley Lokaj, Eds., Vol. 1, 1994, pp. 348-357, ASCE, New York.

11. Pindera, M-J., Arnold, S. M., and Williams, T. O., "Thermoplastic Response of Metal Matrix Composites with Homogenized and Functionally Graded Interfaces," **Composites Engineering**, Vol. 4, No. 1, 1994, pp. 129-145.

12. Pindera, M-J. and Freed, A. D., "The Effect of Matrix Microstructure on Thermally-Induced Residual Stresses in SiC/Titanium Aluminide Composites," **Journal of Engineering Materials and Technology**, Vol. 116, No. 2, 1994, pp. 215-221.

13. Salzar, R. S., Williams, T. O., and Pindera, M-J., "OPTCOMP: A User-Friendly Algorithm for Optimization of Residual Stresses in MMC's Using Compensating/Compliant Interfacial Layers," in **Proc. HITEMP Review 1994: NASA Conference Publication 10146**, Vol. 2, pp. 29-1:29-12, 1994.

14. Williams, T. O. and Pindera, M-J., "Multiple Concentric Cylinder Model (MCCM) User's Guide," **NASA Contractor Report 195299**, NASA Lewis Research Center, Cleveland, OH. April 1994.

15. Pindera, M-J., Salzar, R. S. and Williams, T. O., "Optimization of Residual Stresses in MMC's Using Compensating/Compliant Interfacial Layers: Part II - OPTCOMP User's Guide," **NASA Contractor Report 195337**, NASA Lewis Research Center, Cleveland, OH. May 1994.

16. Williams, T. O. and Pindera, M-J., "TMF Modeling of Advanced Metal Matrix Composites in the Presence of Microstructural Details," **Materials Science and Engineering**, Vol. A200, No. 1-2, 1995, pp. 156-172.

17. Bufler, H., "Theory of Elasticity of a Multilayered Media," **J. Elasticity**, Vol. 1, 1971, pp. 125-143.

18. Pindera, M-J., "Local/Global Stiffness Matrix Formulation for Composite Materials and Structures," **Composites Engineering**, Vol. 1, No. 2, 1991, pp. 69-83.

19. Mendelson, A., **Plasticity: Theory and Applications**, Robert E. Krieger Publishing Company, Malabar, Florida, 1983 (reprint edition).

20. Aboudi, J., **Mechanics of Composite Materials: A Unified Micromechanical Approach**, Elsevier, The Netherlands, 1991.

21. **DOT** User's Manual, Version 4.00, VMA Engineering, Vanderplaats, Miura & Associates, Inc., Goleta, California, 1993.

22. Bodner, S. R. and Partom, Y., "Constitutive Equations for Elastic-Viscoplastic Strain Hardening Material," **J. Applied Mechanics**, Vol. 42, pp. 385-389, 1975.

23. Bodner, S. R., "Review of a Unified Elastic-Viscoplastic Theory," in **Unified Constitutive Equations for Creep and Plasticity**, A. K. Miller, Ed., Elsevier, Amsterdam, 1987, pp. 273-301.

24. Brayshaw, J. B. and Pindera, M-J., "The Effect of Matrix Constitutive Model on Residual Thermal Stresses in MMC," **Journal of Engineering Materials and Technology**, Vol. 116, No. 4, 1994, pp. 505-511.

25. Nimmer, R. P., Bankert, R. J., Russell, E. S. and Smith, G. A., "Micromechanical Modeling of Fiber/Matrix Interface Effects in SiC/Ti Metal Matrix Composites," **NASA Conference Publication 3054, Part 2**, J. D. Buckley, Ed., 1990, pp. 493 - 535.

26. Williams, T. O., "Inelastic Microstructural Modeling of Metal Matrix Composites," Ph.D. dissertation, Civil Engineering & Applied Mechanics Department, University of Virginia, May 1995.

7.0 APPENDICES

7.1 Appendix I: Constitutive Models and Material Properties

Three different constitutive theories are presently available within **micro.f** for modeling the temperature-dependent inelastic response of the individual constituents employed in constructing the concentric cylinder assemblage. These are the rate-independent incremental plasticity theory, rate-dependent Bodner-Partom viscoplasticity theory, and one user-defined theory formulated in terms of inelastic strain rates. A brief description of these theories is provided next, followed by the material properties of the constituents modeled using these theories that have been entered into the **class.data, visco.data** and **user.data** files.

7.1.1 Classical incremental plasticity theory

In the classical incremental plasticity theory, the plastic strain increment is derived from a von Mises yield condition of the form,

$$F = \frac{1}{2}\sigma'_{ij}\sigma'_{ij} - \frac{1}{3}\bar{\sigma}^2(\bar{\varepsilon}^p, T) = 0 \tag{A1.1}$$

where the effective yield stress $\bar{\sigma}$ is a function of both the effective plastic strain $\bar{\varepsilon}^p$ and temperature T. Using the associated flow rule, the plastic strain increment is thus,

$$d\varepsilon^p_{ij} = \frac{\partial F}{\partial \sigma'_{ij}} d\lambda = \sigma'_{ij} d\lambda \tag{A1.2}$$

where $d\lambda > 0$ for plastic loading, and $d\lambda \le 0$ for neutral loading or unloading. The proportionality constant $d\lambda$ is obtained from a consistency condition that ensures that the stress vector remains on the yield surface during plastic loading, and is given in terms of the elastic stiffness elements, stresses, elastic strains and the strain-hardening characteristics (Pindera et al. [4]). This form of the incremental plasticity equations was employed in previous investigations and found to yield generally good convergence. For materials with very low rates of strain-hardening however, difficulties can be encountered using this form of the incremental plasticity. To ensure convergence of the iterative scheme for a wide class of materials in a wide temperature range, so-called plastic strain-total strain plasticity relations were employed in the present investigation by rewriting eqn (A1.2) in terms of total strains without recourse to the stresses (Mendelson [19]). In this formulation of the incremental plasticity equations, the plastic strain increments are now given in terms of so-called modified total strain deviators e_{ij},

$$d\varepsilon_{ij}^{p} = e_{ij} / \bar{e}_{eff}\, d\bar{\varepsilon}^{p} \quad (A1.3)$$

where $e_{ij} = \varepsilon_{ij} - 1/3\varepsilon_{kk}\delta_{ij} - \varepsilon_{ij}^{p}\,|_{previous}$, $\bar{e}_{eff} = \sqrt{2/3 e_{ij} e_{ij}}$, and the effective plastic strain increment $d\bar{\varepsilon}^{p}$ is given by

$$d\bar{\varepsilon}^{p} = \bar{e}_{eff} - \bar{\sigma}/3G \quad (A1.4)$$

Herein, the elastoplastic stress-strain response of the matrix is taken to be **bilinear**, with the effective stress $\bar{\sigma}(\bar{\varepsilon}^{p}, T)$ given by,

$$\bar{\sigma}(\bar{\varepsilon}^{p}, T) = \bar{\sigma}_{y}(T) + H_{p}(T)\bar{\varepsilon}^{p} \quad (A1.5)$$

The implementation of these plastic strain-total strain plasticity relations is carried out in the same manner as the classical form. That is, the yield condition is first checked at each point within the elastoplastic material to determine whether the material continues to load elastically or whether it has yielded. If the material has yielded, then continued loading is ensured by $d\bar{\varepsilon}^{p} > 0$ and unloading by $d\bar{\varepsilon}^{p} \leq 0$ (see Williams [26] for a more detailed discussion).

7.1.2 Bodner-Partom unified viscoplasticity theory

The Bodner-Partom theory currently available within **micro.f** is limited to viscoplastic materials that exhibit isotropic hardening. While the theory, in general, models rate-dependent behavior of metals at elevated temperatures, it is particularly suitable for modeling rate-dependent plastic deformation at different loading rates.

According to the Bodner-Parton theory, the viscoplastic strain rate is expressed as

$$\dot{\varepsilon}_{ij}^{p} = \Lambda\, s_{ij} \quad (A2.1)$$

where Λ is the flow rule function of the inelastic layer and s_{ij} are the deviatoric stress components, that is $s_{ij} = \sigma_{ij} - \sigma_{kk}\delta_{ij}/3$. The explicit form of the flow rule function is given by

$$\Lambda = D_0 \exp\{-\hat{n}\,[Z^2/(3J_2)]^{n}\} / \sqrt{J_2} \quad (A2.2)$$

where $\hat{n} = (n+1)/2n$, and $J_2 = \mathbf{s}\cdot\mathbf{s}/2$ is the second invariant of the deviatoric stresses. D_0 and n are inelastic parameters, and Z is a state variable given for an isotropic hardening material by

$$Z = Z_1 + (Z_0 - Z_1) \exp[-m\, W_p / Z_0] \quad (A2.3)$$

where W_p is the plastic work per unit volume.

The five parameters D_0, Z_0, Z_1, n and m appearing in eqns (A2.2) and (A2.3) have the following interpretation: D_0 is the limiting strain rate in shear for large values of the second stress invariant J_2; Z_0 is the initial value of the hardening variable Z which is related to the yield stress of the material in simple tension; Z_1 is the saturation value of the hardening variable for large values of stresses; m is a parameter that controls the rate of work-hardening of the material; and n is a parameter that controls the rate sensitivity of the material. More information regarding the meaning and physical interpretation of these parameters can be found in Bodner [23].

7.1.3 User defined, rate-dependent inelastic model

The user can provide his own rate-dependent, inelastic constitutive theory which allows calculation of the inelastic strain rate according to the following format:

$$\dot{\varepsilon}_{ij}^{in} = f_{ij}\,(\,t,\,T,\,\sigma_{kl}^{initial},\,\sigma_{kl}^{current},\,\alpha_{kl}^{initial},\,\alpha_{kl}^{current},\,\varepsilon_{kl}^{(initial)in},\,d_1,\dots,\,d_{10}\,) \qquad (A3.1)$$

where t is the time, T is the temperature, α_{kl} are back stress components, and d_1 through d_{10} are the constitutive model parameters generic to the particular model. This functional form is limited to viscoplasticity or creep theories that do not contain drag or yield stresses.

As an illustration, consider the power-law creep model presented by Nimmer et al. [25]. The creep strain rate is given by

$$\dot{\varepsilon}_{ij}^{vp} = \frac{3}{2}\,a_0\,a_2\,t^{a_2-1}\,\exp\left[-a_3\,/\,(T+274)\right]\,\bar{\sigma}_{eff}^{\,a_1-1}\,s_{ij} \qquad (A3.2)$$

where $\bar{\sigma}_{eff}$ is the effective stress, s_{ij} are the stress deviators, and a_0, a_1, a_2, and a_3 are the power-law coefficients for the matrix provided in Section 7.1.7.

The subroutine that provides the means for employing a user-constructed inelastic constitutive theory in **OPTCOMP2** is presented in the following section. As an illustration, calculation of the inelastic strain increments based on the above power-law creep model is included in the appropriate section of the subroutine. In this example, the power-law coefficients a_0 through a_3 correspond to the constitutive model parameters d_1 through d_4.

7.1.4 User.f file: construction of a user-defined constitutive model subroutine

The subroutine USERVP that calculates strain rates based on the user-defined flow rule of the form given in eqn (A3.1) is provided below. The specific equation coded in the allocated space in the subroutine is the power-creep model given by eqn (A3.2) in the preceding section. The inelastic strain rates (and back stress rates if defined within the subroutine) calculated in the subroutine are returned to the main program where their increments are subsequently determined using a predictor-corrector scheme discussed in Section 3.1.2.

```
      SUBROUTINE USERVP(TIME,TEMPC,STRXXI,STRRRI,STRTTI,
     &  STRXX,STRRR,STRTT,AXXI,ARRI,ATTI,AXTI,AXRI,ATRI,
     &  AXX,ARR,ATT,AXT,AXR,ATR,EPIXXP,EPIRRP,EPITTP,
     &  D1I,D2I,D3I,D4I,D5I,D6I,D7I,D8I,D9I,D10I,
     &  AXXRI,ARRRI,ATTRI,AXTRI,AXRRI,ATRRI,AXXR,ARRR,ATTR,
     &  AXTR,AXRR,ATRR,EPSXXR,EPSRRR,EPSTTR)
      IMPLICIT REAL*8(A-H,O-Z)
      REAL*8 STRXXI,STRTTI,STRRRI,STRXX,STRTT,STRRR
      REAL*8 EPSXXR,EPSTTR,EPSRRR,
      REAL*8 EPIXXP,EPIRRP,EPITTP
      REAL*8 AXX,ARR,ATT,AXT,AXR,ATR
      REAL*8 AXXI,ARRI,ATTI,AXTI,AXRI,ATRI
      REAL*8 AXXR,ARRR,ATTR,AXTR,AXRR,ATRR
      REAL*8 AXXRI,ARRRI,ATTRI,AXTRI,AXRRI,ATRRI
      REAL*8 D1I(40),D2I(40),D3I(40),D4I(40),D5I(40),D6I(40)
      REAL*8 D7I(40),D8I(40),D9I(40),D10I(40)
      REAL*8 TIME,TEMPC
C*********INSERT CONSTITUTIVE MODEL BELOW IN THIS FORM***************
            PMEAN=(STRXXI+STRRRI+STRTTI)/3.0
            SXX=STRXXI-PMEAN
            SRR=STRRRI-PMEAN
            STT=STRTTI-PMEAN
            STREFF=DSQRT((3./2.)*(SXX*SXX+SRR*SRR+STT*STT))
            C1=1.5*D1I*D3I*TIME**(D3I-1)
            C2=DEXP(-D4I/(TEMPC+274))
            EPSXXR=C1*C2*STREFF**(D2I-1)*SXX
            EPSRRR=C1*C2*STREFF**(D2I-1)*SRR
            EPSTTR=C1*C2*STREFF**(D2I-1)*STT
C*******************************************************************
      RETURN
      END
C*******************************************************************
```

```
where:

TIME  = current time
TEMPC = current temperature
STRXX,STRRR,STRTT     = current longitudinal, radial, and circumferential stresses
STRXXI,STRRRI,STRTTI = initial longitudinal, radial, and circumferential stresses
AXX,ARR,ATT,AXT,AXR,ATR       = current back stresses in the user-defined model
AXXI,ARRI,ATTI,AXTI,AXRI,ATRI = initial back stresses in the user-defined model
AXXR,ARRR,ATTR,AXTR,AXRR,ATRR       = current back stress rates
AXXRI,ARRRI,ATTRI,AXTRI,AXRRI,ATRRI = initial back stress rates
EPIXXP,EPIRRP,EPITTP = initial inelastic longitudinal, radial, and circumferential
                       strains
EPSXXR,EPSRRR,EPSTTR = inelastic longitudinal, radial, and circumferential strain
                       rates
D1I,D2I,...,D10I = 10 inelastic constitutive model parameters
```

7.1.5 Material properties in the class.data file

The material properties for the three elastic fibers, SCS-6 SiC, Al_2O_3 and Gr, and the six elastoplastic matrices Ti-24Al-11Nb, Ti-6Al-4V, NiAl, FeAl, FeAl1, and Cu, modeled using the classical incremental plasticity equations with isotropic hardening described in Appendix I (Section 7.1.1) are given below.

Material properties of Al_2O_3, NiAl and FeAl(1).

Material properties	27°C	127°C	227°C	327°C	427°C	527°C	627°C	727°C
Al_2O_3								
α (x 10^{-6} /°C)	9.0	9.0	9.0	9.0	9.0	9.0	9.0	9.0
E (GPa)	451.6	447.6	443.6	439.6	435.6	431.6	427.6	423.6
ν	0.3	0.3	0.3	0.3	0.3	0.3	0.3	0.3
NiAl								
α (x 10^{-6} /°C)	12.7	13.0	13.3	13.6	13.8	14.1	14.3	14.5
E (GPa)	192.6	188.5	184.4	180.3	176.2	172.1	168.0	163.9
ν	0.3	0.3	0.3	0.3	0.3	0.3	0.3	0.3
σ_y (MPa)	315.0	275.5	225.0	195.0	150.0	135.0	115.0	95.0
H (GPa)	180.2	85.2	6.1	3.6	1.6	0.8	0.0	0.0
FeAl and FeAl1								
α (x 10^{-6} /°C) (FeAl)	13.0	15.7	17.7	19.0	19.9	20.7	21.4	22.2
α (x 10^{-6} /°C) (FeAl1)	17.3	18.3	19.3	20.0	20.7	21.1	21.4	21.6
E (GPa)	260.8	252.8	244.9	237.2	229.4	222.1	217.1	208.3
ν	0.3	0.3	0.3	0.3	0.3	0.3	0.3	0.3
σ_y (MPa)	578.0	555.0	533.0	533.0	533.0	436.0	333.0	213.0
H (GPa)	51.9	50.3	48.8	26.5	4.2	4.0	3.8	3.8

Material properties of the titanium alloy Ti-6Al-4V

Temperature (°C)	E (GPa)	ν	α_{low} (10^{-6}/ °C)	α_{high} (10^{-6}/ °C)	σ_Y (MPa)	H (GPa)
21	113.7	0.3	9.44	11.0	900	4.6
149	107.5	0.3	9.62	11.3	730	4.7
315	97.9	0.3	9.78	11.7	517	5.4
482	81.3	0.3	9.83	12.0	482	4.8
649	49.6	0.3	9.72	12.2	303	1.7
900	20.7	0.3	9.81	12.4	35	1.2

Material properties SCS-6 SiC, Ti_3Al, Cu and Gr.

Material properties	24°C	204°C	427°C	649°C	760°C	815°C
SCS-6 SiC						
α (x 10^{-6} /°C)	3.53	3.62	3.91	4.27	4.42	4.49
E (GPa)	400.0	400.0	400.0	400.0	400.0	400.0
ν	0.25	0.25	0.25	0.25	0.25	0.25
Ti-24-Al-11Nb						
α (x 10^{-6} /°C)	9.00	9.36	10.26	10.62	10.91	11.07
E (GPa)	110.3	100.0	75.8	68.2	51.3	42.7
ν	0.26	0.26	0.26	0.26	0.26	0.26
σ_y (MPa)	371.5	406.7	370.2	269.5	200.4	165.5
H (GPa)	22.98	3.04	2.22	0.69	0.23	0.11
Cu						
α (x 10^{-6} /°C)	16.0	17.0	18.36	19.25	19.80	20.08
E (GPa)	78.8	58.9	36.8	23.8	16.8	14.4
ν	0.38	0.38	0.38	0.38	0.38	0.38
σ_y (MPa)	37.1	31.6	26.7	22.5	20.0	19.6
H (GPa)	6.37	4.27	2.38	1.16	0.98	0.90
Gr						
α (x 10^{-6} /°C)	10.0	10.0	10.0	10.0	10.0	10.0
E (GPa)	41.4	41.4	41.4	41.4	41.4	41.4
ν	0.24	0.24	0.24	0.24	0.24	0.24

In the above tables, α is the instantaneous thermal expansion coefficient; E is the Young's modulus, ν is the Poisson's ratio; σ_y is the yield stress in simple tension; and H is the hardening slope based on a bilinear representation of the elastoplastic stress-strain response. The properties of the aluminum oxide fiber, nickel and iron aluminide matrices are given at eight different temperatures, whereas the properties for the remaining material systems are given at six different temperatures.

7.1.6 Material properties in the visco.data file

The material properties for the elastic SiC fiber and the viscoplastic Ti-6Al-4V matrix modeled using the Bodner-Partom unified viscoplasticity equations with isotropic hardening described in Appendix I (Section 7.1.2) are given below. The elastic properties of the Ti-6Al-4V matrix are the same as those provided in Section 7.1.5 and are thus not repeated.

Material properties of the SiC fibers

Fiber	E(GPa)	ν	α (x 10^{-6} /°C)
SiC	414	0.3	4.86

Bodner-Partom parameters of the titanium alloy Ti-6Al-4V

Temperature (°C)	D_0^{-1} (s)	Z_0 (MPa)	Z_1 (MPa)	m	n
21	10^{-4}	1060	1500	12.7	10.0
149	10^{-4}	890	1500	11.68	8.42
315	10^{-4}	800	1500	19.2	3.6
482	10^{-4}	1140	1500	121	1.71
649	10^{-4}	1160	1500	85.6	1.038
900	10^{-4}	580	1500	340	0.396

7.1.7 Material properties in the user.data file

The material properties for the viscoplastic Ti-6Al-4V matrix modeled using the power-law creep equations described in Appendix I (Section 7.1.3) are given below. The elastic properties of the Ti-6Al-4V matrix are the same as those provided in Section 7.1.5 and are thus not repeated.

Power-law creep coefficients of the titanium alloy Ti-6Al-4V

a_0 (10^9) (MPa)$^{-3.403}$ (hr.)$^{-0.9251}$	a_1	a_2	a_3 (10^4 °C)
3.6	3.403	0.9251	3.6

7.2 Appendix II: Constraint.f and Objective.f Files

7.2.1 Constraint.f file: construction of the EXTCONST subroutine

The EXTCONST subroutine contained in the **constraint.f** file that allows the user to define his or her own constraint function or functions is provided below. The constraint functions are assigned to the variable G, starting with G(ICON) and ending with G(ICON+N), where N+1 is the total number of constraint functions specified by the user. The constraints must be defined consecutively in the subroutine, starting with the first constraint, according to the format G(ICON) = ..., G(ICON+1) = ..., G(ICON+2) = ..., etc. The optimization subroutines included in **dot.f** interpret the expressions assigned to the variables G(ICON) through G(ICON+N) to be always less than or equal to zero (i.e., G(ICON) $\leq$ 0). Thus when formulating constraints to be included in the user-defined subroutine, the following rules need to be followed. The constraint $x \leq 5000.0$ should be written

$$G(ICON) = X/5000.0 - 1.0$$

Likewise, the constraint $x \geq 5000.0$ should be written

$$G(ICON) = -(X/5000.0 - 1.0)$$

To formulate an equality constraint $x = 5000.0$ the following expressions should be coded

$$G(ICON) = X/5000.0 - 1.0$$

$$G(ICON+1) = -(X/5000.0 - 1.0)$$

The user has the option to construct constraint functions using the field variables specified in the common blocks BK1 through BK5 of the EXTCONST subroutine. These include stresses, total, elastic and inelastic strains, effective plastic strains, interfacial radial displacements and inclusion volume fraction (in the case of heterogeneous layers) in each of the layers within the concentric cylinder assemblage. With the exception of the interfacial radial displacements, these field quantities are dimensioned as double arrays with the maximum dimensions (maxnring,maxncolp). The first dimension pertains to the maximum number of layers in the concentric cylinder assemblage and the second dimension pertains to the maximum number of radial locations within each layer at which a given quantity is calculated in the program. These radial locations are equally spaced and divide a given cylinder into the specified number of sub-layers for computational purposes. The maximum dimensions are defined in the **paraccm.v2.h** file with the following defaults maxnring = 25 and maxncolpt = 250. The actual dimensions are specified by the user during the **optcomp2.data** construction procedure as illustrated in the examples provided.

The two constraint functions included in the EXTCONST subroutine provided herein impose constraints on the average value of the inclusion volume fraction FVF in the three interfacial layers of the concentric cylinder assemblage in Example 3. The first constraint given by G(ICON)=(((FVF(2)+FVF(3)+FVF(4))/3.0)/.10)-1.0 specifies that the average inclusion volume fraction in the three interfacial layers be less than 0.10, while the second constraint G(ICON+1)=-((((FVF(2)+FVF(3)+FVF(4))/3.0)/.10)-1.0) specifies that this value be greater than 0.10. When these two constraint functions are employed together, the result is an equality constraint that requires the average inclusion volume fraction in the specified layers be equal to 0.10. The variables employed in the subroutine EXTCONST below are the same as the variables in the subroutine EXTOBJ which are defined in the following section.

```
      SUBROUTINE EXTCONST(G,ICON,KKSUBCON,R)
      INCLUDE 'paraccm.v2.h'
      REAL G(40)
      REAL*8 FVF(MAXNRING)
      REAL*8 STRXX(maxnring,maxncolpt),STRRR(maxnring,maxncolpt)
      REAL*8 STRTT(maxnring,maxncolpt)
      REAL*8 EPSXX,EPSRR(maxnring,maxncolpt)
      REAL*8 EPSTT(maxnring,maxncolpt)
      REAL*8 EPSXXE(maxnring,maxncolpt),EPSRRE(maxnring,maxncolpt)
      REAL*8 EPSTTE(maxnring,maxncolpt)
      REAL*8 EPSXXP(maxnring,maxncolpt),EPSRRP(maxnring,maxncolpt)
      REAL*8 EPSTTP(maxnring,maxncolpt)
      REAL*8 EPEFF(maxnring,maxncolpt),W(MAXNEQ),R(MAXNRING)
      COMMON /BK1/ STRXX,STRRR,STRTT
      COMMON /BK2/ EPSXX,EPSRR,EPSTT
      COMMON /BK3/ EPSXXE,EPSRRE,EPSTTE
      COMMON /BK4/ EPSXXP,EPSRRP,EPSTTP
      COMMON /BK5/ EPEFF,W,FVF
C********INSERT CONSTRAINTS BELOW IN THIS FORM**************************
      G(ICON)=(((FVF(2)+FVF(3)+FVF(4))/3.0)/.10)-1.
      G(ICON+1)=-((((FVF(2)+FVF(3)+FVF(4))/3.0)/.10)-1.)
C**********************************************************************
      ICON=ICON+KKSUBCON-1
      RETURN
      END
```

7.2.2 Objective.f file: construction of the EXTOBJ subroutine

The EXTOBJ subroutine contained in the **objective.f** file that allows the user to define his or her own objective function is provided below. The objective function is assigned to the variable OBJ. Any combination or function of the available variables contained in the common blocks BK1 through BK5 may be used in creating a user-defined objective function. These variables are the same as those specified in the EXTCONST subroutine and described in the preceding sub-section. It should be noted that only one objective function can be defined in the subroutine. The objective function must be written in standard fortran according to the format OBJ = ...

The objective function provided in this example minimizes the difference between the effective plastic strain in the middle of each of the three interfacial layers employed in Example 3 and the average effective plastic strain in these layers. This, effectively, makes the effective plastic strain distribution uniform throughout the interfacial region,

$$\left| 1.0 - e^{\sum_{i=2}^{i=4}(\varepsilon_{eff}^{pl}(i,11) - \bar{\varepsilon}_{eff}^{pl})} \right|$$

where $\varepsilon_{eff}^{pl}(i,11)$ is the effective plastic strain in the ith layer evaluated in the middle of the interfacial layer (i.e., 11th integration point, where the total number of integration points in each of the interfacial layers is 21), defined as $\varepsilon_{eff}^{pl} = \sqrt{2/3\varepsilon_{ij}^{pl}\varepsilon_{ij}^{pl}}$, and $\bar{\varepsilon}_{eff}^{pl}$ is the average value of these effective plastic strains,

$$\bar{\varepsilon}_{eff}^{pl} = \frac{1}{3}\sum_{i=2}^{i=4}\varepsilon_{eff}^{pl}(i,11)$$

```
      SUBROUTINE EXTOBF(OBJ)
      INCLUDE 'paraccm.v2.h'
      REAL OBJ
      REAL*8 STRXX(maxnring,maxncolpt),STRRR(maxnring,maxncolpt)
      REAL*8 STRTT(maxnring,maxncolpt)
      REAL*8 EPSXX,EPSRR(maxnring,maxncolpt)
      REAL*8 EPSTT(maxnring,maxncolpt)
      REAL*8 EPSXXE(maxnring,maxncolpt)
      REAL*8 EPSRRE(maxnring,maxncolpt)
      REAL*8 EPSTTE(maxnring,maxncolpt)
      REAL*8 EPSXXP(maxnring,maxncolpt)
      REAL*8 EPSRRP(maxnring,maxncolpt)
      REAL*8 EPSTTP(maxnring,maxncolpt)
      REAL*8 EPEFF(maxnring,maxncolpt),W(maxneq)
      REAL*8 EPT,EFFDIF
      REAL*8 STREFF(MAXNRING,MAXNCOLPT)
      COMMON /BK1/ STRXX,STRRR,STRTT,STREFF
      COMMON /BK2/ EPSXX,EPSRR,EPSTT
      COMMON /BK3/ EPSXXE,EPSRRE,EPSTTE
      COMMON /BK4/ EPSXXP,EPSRRP,EPSTTP
      COMMON /BK5/ EPEFF,W,FVF
C*************INSERT OBJECTIVE FUNCTION BELOW IN THIS FORM***************
       AVE=(EPEFF(2,11)+EPEFF(3,11)+EPEFF(4,11))/3.
       ZOBJ=ABS(EPEFF(2,11)-AVE)+
     &      ABS(EPEFF(3,11)-AVE)+ABS(EPEFF(4,11)-AVE)
       OBJ=ABS(1-EXP(ZOBJ*1000))
C**********************************************************************
      RETURN
      END
```

```
C**********************************************************************
C**********************************************************************
C**********************************************************************
C   STRXX(I,J), STRRR(I,J), STRTT(I,J): The longitudinal, radial and
C             circumferential stress at the jth collocation point in
C             each of the ith layer of the cylinder assemblage.
C
C   EPSXX(I,J), EPSRR(I,J), EPSTT(I,J): The total longitudinal, radial
C             and circumferential strain at the jth collocation point
C             in each of the ith layer of the cylinder assemblage.
C
C   EPSXXE(I,J), EPSRRE(I,J), EPSTTE(I,J): The elastic longitudinal,
C             radial and circumferential strain at the jth collocation
C             point in each of the ith layer of the cylinder assemblage.
C
C   EPSXXP(I,J), EPSRRP(I,J), EPSTTP(I,J): The inelastic longitudinal,
C             radial and circumferential strain at the jth collocation
C             point in each of the ith layer of the cylinder assemblage.
C
C   EPEFF(I,J): The effective plastic strain at the jth collocation point
C             in each of the ith layer of the cylinder assemblage.
C
C   W(I): The common interfacial radial displacement between the ith and
C             ith+1 layer.
C
C  FVF(I): The inclusion volume fraction in the ith layer.
C**********************************************************************
C**********************************************************************
C**********************************************************************
```

7.3 Appendix III: Example 1 - Time-Independent Process History Optimization

7.3.1 Construction of the optcomp2.data file

The construction of the **optcomp2.data** file for the time-independent optimization problem of Example 1, menu-driven by the user-friendly interface **shell.f**, is illustrated below. In this and the following examples, the text that appears in Courier-type capital letters is written to the screen at each step in the construction of the **optcomp2.data** file. User's responses to the menu-driven commands are shown in bold Courier-type letters. ***The text in bold italics preceded by the word "Note:" represents manually inserted comments that explain in more detail certain options available to the user.***

```
NASA-LEWIS RESEARCH CENTER, STRUCTURES DIVISION, CLEVELAND OHIO
           ****************************************************
           ***                    OPTCOMP2                  ***
           ***                                              ***
           ***   CONCENTRIC CYLINDER OPTIMIZATION PROGRAM   ***
           *** FOR THE DETERMINATION OF IDEALIZED INTERFACE ***
           ***    MICROSTRUCTURE AND PROCESSING HISTORY     ***
           ***                                              ***
           ***                  WRITTEN BY                  ***
           ***                                              ***
           ***              ROBERT SCOTT SALZAR             ***
           ***              MAREK-JERZY PINDERA             ***
           ***                                              ***
           ***           THE UNIVERSITY OF VIRGINIA         ***
           ***                  JUNE 1995                   ***
           ***                                              ***
           ***    DEVELOPED FOR THE FATIGUE AND FRACTURE    ***
           ***     BRANCH OF NASA-LEWIS RESEARCH CENTER     ***
           ***          UNDER CONTRACT NAS3-26571           ***
           ***     DR. S. M. ARNOLD  (CONTRACT MONITOR)     ***
           ****************************************************

==================================LEGAL NOTICE================================

Neither NASA, nor any of its employees, contractors or grantees, nor any
person acting on behalf of either:
a.  makes any warranty or representation, expressed or implied, with respect
    to the accuracy, completeness, or usefulness of this software, or that the
    use of this software may not infringe privately owned rights; or

b.  assumes any liabilities with respect to the use of, or for damages
    resulting from use of this software.

==============================================================================

          ******MAIN MENU******

         1.  CREATE NEW DATA FILE
         2.  RUN EXISTING DATA FILE
         3.  ENTER NEW MATERIALS INTO DATABANK
         4.  EXIT SHELL
        ENTER CHOICE -> 1
```

Note: The main menu allows the user to: 1. create a new data file to be stored in optcomp2.data; 2. execute the data file in optcomp2.data; 3. enter new or additional materials into the material databank, or; 4. exit the OPTCOMP2 program.

```
***********************************BLOCK 1***********************************
              SPECIFY CONCENTRIC CYLINDER GEOMETRY, MATERIALS
                          INCLUDING PROPERTIES
```

Note: Select choice n for a solid, homogeneous fiber. Select choice y if the fiber has a layered morphology. If a layered fiber is selected, the user will be asked to supply the number of layers in the fiber, including the core. The radius of the core is then requested, followed by the thickness of each layer.

```
DOES THE FIBER HAVE LAYERED MORPHOLOGY? <Y/N> n
ENTER RADIUS OF FIBER CORE -> 0.63246
ENTER FIBER VOLUME FRACTION -> 0.40
```

Note: Next, enter the number of interfacial layers for the concentric cylinder assemblage, followed by the thickness of each of the interfacial layers.

```
ENTER THE NUMBER OF INTERFACE LAYERS -> 1

ENTER THICKNESS OF INTERFACE 1 -> 0.10


                 ***CONCENTRIC CYLINDER GEOMETRY***

 LAYER           MATERIAL       NORMALIZED OUTER RADIUS     VOLUME FRACTION

   1               FIBER                0.63246                  0.4000
   2            INTERFACE 1             0.73245                  0.1365
   3              MATRIX                1.00000                  0.4635

FIBER VOLUME FRACTION =        0.4000
INTERFACE VOLUME FRACTION =    0.1365
MATRIX VOLUME FRACTION =       0.4635

   IS INFORMATION CORRECT? <Y/N> y
```

Note: If the information supplied in the summary is correct, enter y. If the information is incorrect and needs to be changed, enter n.

```
***COMPOSITE MICRO-STRUCTURE***
```

Note: Fiber, interfacial or matrix regions can be either homogeneous or heterogeneous (two-phase composite regions).

```
IF FIBER CORE IS HOMOGENEOUS,    ENTER 1
IF FIBER CORE IS HETEROGENEOUS,  ENTER 2
-> 1

IF INTERFACE LAYER  1 IS HOMOGENEOUS,   ENTER 1
IF INTERFACE LAYER  1 IS HETEROGENEOUS, ENTER 2
-> 1

IF MATRIX LAYER IS HOMOGENEOUS,   ENTER 1
IF MATRIX LAYER IS HETEROGENEOUS, ENTER 2
-> 1

***INELASTIC CONSTITUTIVE MODEL SELECTION***
```

Note: Three different constitutive models for the response of the various concentric cylinder regions are available, namely: classical incremental plasticity theory, Bodner-Partom unified viscoplasticity theory, and a user-defined model (in this case a power-law creep model).

```
FOR CLASSICAL PLASTICITY, ENTER 1
FOR BODNER-PARTOM,        ENTER 2
FOR USER-DEFINED MODEL,   ENTER 3
-> 1

        ***MATERIAL PROPERTY SELECTION***

    AVAILABLE MATERIALS           AVAILABLE CONSTITUTIVE MODELS
1    SiC (SCS-6)                        1  ELASTIC
2    Al2O3                              2  PLASTIC
```

```
  3   Gr
  4   Ti-24Al-11Nb
  5   Ti-6Al-4V
  6   NiAl
  7   FeAl
  8   FeAll
  9   Cu
 10   ENTER NEW MATERIAL
```

Note: Enter the material for the fiber (or fiber layers if layered fiber has been specified), the interfacial layer(s) and the matrix, and the constitutive model as prompted. The fiber core must be an isotropic material. If a material other than isotropic is selected, an error message will result. For subsequent fiber layers, enter fiber material and constitutive model as prompted. If the material you wish is not available, select ENTER NEW MATERIAL and follow the instructions given. You will then be presented with a new material menu including the material just entered. You may now select that material.

```
ENTER MATERIAL FOR FIBER CORE -> 1


ENTER MATERIAL FOR INTERFACE LAYER 1 -> 4
ENTER CONSTITUTIVE MODEL FOR INTERFACE LAYER 1 -> 2

ENTER MATERIAL FOR MATRIX LAYER -> 4
ENTER CONSTITUTIVE MODEL FOR MATRIX LAYER -> 2


         ***LAMINATED CYLINDER MATERIAL SPECIFICATION***

     LAYER               INCLUSION          MATRIX             FVF

FIBER CORE                ------           SiC (SCS-6)        -----
INTERFACE LAYER  1        ------           Ti-24Al-11Nb       -----
MATRIX LAYER              ------           Ti-24Al-11Nb       -----

        IS INFORMATION CORRECT? <Y/N> y


**********************************BLOCK 2**********************************
                DEFINE PROCESSING/LOAD HISTORY, INCREMENT,
                             AND ITERATIONS


                              ***CAUTION***
THE APPLIED TEMPERATURE LOAD MUST REMAIN BETWEEN   23.89 deg AND  815.56 deg
                              *************
```

Note: The data for the materials chosen have been internally analyzed and the applied thermal load cannot exceed the stated limits due to the temperature range of the supplied data.

Note: Enter total number of loading segments followed by axial load control mode for the first segment (i.e., whether axial stress or axial strain will be specified). Next, enter the initial temperature, the initial external pressure, and the initial axial stress or strain. Then, enter the time duration of the load segment and the number of increments that this load segment will be divided into, followed by the final temperature, the final external pressure and the final axial stress or axial strain for each of the specified loading segments. In this example, each load segment has been divided into 500 increments for the specified temperature and axial stress loading.

```
NUMBER OF LOAD SEGMENTS -> 4

IF FIRST LOAD SEGMENT IS UNDER STRESS CONTROL, ENTER 1
IF FIRST LOAD SEGMENT IS UNDER STRAIN CONTROL, ENTER 2
-> 1

INITIAL TEMPERATURE, INITIAL EXTERNAL PRESSURE, INITIAL AXIAL STRESS
-> 815 0 0
DURATION OF LOAD STEP,  NUMBER OF LOAD INCREMENTS
-> 1 500
ENDING TEMPERATURE, ENDING PRESSURE, ENDING AXIAL STRESS
-> 815 -200 0
IF LOAD SEGMENT  2 IS UNDER STRESS CONTROL, ENTER 1
IF LOAD SEGMENT  2 IS UNDER STRAIN CONTROL, ENTER 2
-> 1
```

```
DURATION OF LOAD STEP,  NUMBER OF LOAD INCREMENTS
-> 1 500
ENDING TEMPERATURE, ENDING PRESSURE, ENDING AXIAL STRESS
-> 430 -200 0
IF LOAD SEGMENT  3 IS UNDER STRESS CONTROL, ENTER 1
IF LOAD SEGMENT  3 IS UNDER STRAIN CONTROL, ENTER 2
-> 1

DURATION OF LOAD STEP,  NUMBER OF LOAD INCREMENTS
-> 1 500
ENDING TEMPERATURE, ENDING PRESSURE, ENDING AXIAL STRESS
-> 24 -200 0
IF LOAD SEGMENT  4 IS UNDER STRESS CONTROL, ENTER 1
IF LOAD SEGMENT  4 IS UNDER STRAIN CONTROL, ENTER 2
-> 1

DURATION OF LOAD STEP,  NUMBER OF LOAD INCREMENTS
-> 1 500
ENDING TEMPERATURE, ENDING PRESSURE, ENDING AXIAL STRESS
-> 24 0 0

                                          ***LOAD HISTORY***

STEP  TIME   INCREMENTS  TEMPERATURE  PRESSURE   AXIAL FORCE  AXIAL STRAIN
  1                         815.0       0.00         0.00
      1.0000        500
  2                         815.0    -200.00         0.00
      1.0000        500
  3                         430.0    -200.00         0.00
      1.0000        500
  4                          24.0    -200.00         0.00
      1.0000        500
  5                          24.0       0.00         0.00
  IS INFORMATION CORRECT? <Y/N> y

        ***SET INTERNAL VARIABLES***
```

Note: Choose the maximum number of iterations and the error tolerance allowed for convergence. Iteration will be terminated after reaching this limit.

```
CHANGE MAXIMUM NUMBER OF ITERATIONS (DEFAULT=10)? <Y/N> y
MAXIMUM NUMBER OF ITERATIONS -> 15

CHANGE CONVERGENCE ERROR TOLERANCES (DEFAULT=0.01)? <Y/N> n
```

Note: Choose the number of integration points for the calculation of the plastic strain distributions in each layer and the number of points at which the field variables will be printed to the optcomp2.out data file.

```
CHANGE NUMBER OF INTEGRATION POINTS (DEFAULT= 21/LAYER) AND
PRINT OPTIONS (DEFAULT= 2/LAYER) FROM DEFAULT VALUES? <Y/N> y

                      INT. POINTS     PRINT NUMBER
LAYER  1                 2                2
LAYER  2                 21               3
LAYER  3                 151              6
```

Note: Indicate whether to suppress or write convergence messages to the optcomp2.conv file.

```
WRITE CONVERGENCE INFORMATION TO optcomp2.conv FILE? <Y/N> y
```

Note: Indicate whether to suppress or write optimization iterations to the screen.

```
WRITE OPTIMIZATION ITERATIONS TO SCREEN? <Y/N> y


INTERNAL VARIABLE REVIEW

          MAXIMUM NUMBER OF ITERATIONS =     15
```

```
            CONVERGENCE ERROR TOLERANCE = 0.01000
            CONVERGENCE INFORMATION WRITTEN TO optcomp2.conv
            OPTIMIZATION ITERATIONS WRITTEN TO SCREEN

  IS INFORMATION CORRECT? <Y/N> y


**********************************BLOCK 3**********************************
                         DEFINE OPTIMIZATION PROBLEM
```

Note: Indicate whether processing history or the interfacial layer microstructure, i.e., fiber volume fraction, is to be optimized.

```
   TO OPTIMIZE PROCESSING HISTORY,          ENTER 1
   TO OPTIMIZE INTERFACE VOLUME FRACTION,   ENTER 2
   -> 1


  ***PROCESSING HISTORY MENU***

  I.D.    DESIGN VARIABLES
   1.       TEMPERATURE
   2.       EXTERNAL PRESSURE
   3.       AXIAL LOAD
```

Note: Enter the number of design variables (1-3), followed by the identification number for each design variable. By selecting any combination of the design variables in any order, those variables will become activated for the specified optimization problem (in this case processing history optimization).

```
 ENTER NUMBER OF SELECTIONS -> 1
 ENTER I.D. CHOICE(S) -> 2
```

Note: Enter the lower and upper bounds for each design variable.

```
 ENTER LOWER AND UPPER BOUNDS FOR EXTERNAL PRESSURE INPUT  1 ->    0 0
 ENTER LOWER AND UPPER BOUNDS FOR EXTERNAL PRESSURE INPUT  2 -> -200 0
 ENTER LOWER AND UPPER BOUNDS FOR EXTERNAL PRESSURE INPUT  3 -> -375 0
 ENTER LOWER AND UPPER BOUNDS FOR EXTERNAL PRESSURE INPUT  4 -> -375 0
 ENTER LOWER AND UPPER BOUNDS FOR EXTERNAL PRESSURE INPUT  5 ->    0 0

                    ***DESIGN VARIABLE SUMMARY***

            EXTERNAL PRESSURE DESIGN RANGE
           STEP       LOWER BOUND       UPPER BOUND
            1              0.00              0.00
            2           -200.00              0.00
            3           -375.00              0.00
            4           -375.00              0.00
            5              0.00              0.00

  IS INFORMATION CORRECT? <Y/N> y
```

Note: Select an objective function from the menu below, or choose the User Defined Objective Function (15) that has been entered by the user into the EXTOBJ subroutine residing in the file objective.f.

```
          CHOOSE AN OBJECTIVE FUNCTION:

  ITEM#         FUNCTION

              FIBER FUNCTIONS
    1. RADIAL STRESS (INTERFACE)

         INTERFACIAL LAYER FUNCTIONS
    2. HOOP STRESS (FIBER/I.L.)
    3. HOOP STRESS (AVERAGE)
    4. RADIAL STRESS (FIBER/I.L.)
    5. RADIAL STRESS (I.L./MATRIX)
    6. HYDROSTATIC PRESSURE (I.L./MATRIX)
    7. LONGITUDINAL STRESS (AVERAGE)
```

```
          MATRIX FUNCTIONS
   8. HOOP STRESS (INTERFACE)
   9. RADIAL STRESS (INTERFACE)
  10. RADIAL STRAIN (INTERFACE)
  11. HYDROSTATIC PRESSURE (INTERFACE)
  12. LONGITUDINAL STRESS (INTERFACE)
  13. LONGITUDINAL STRESS (AVERAGE)

          MISCELLANEOUS FUNCTIONS
  14. LONGITUDINAL STRAIN (ASSEMBLAGE)
  15. USER DEFINED OBJECTIVE FUNCTION
ENTER CHOICE -> 2
```

Note: Specify whether the objective function selected is to be minimized or maximized.

```
OBJECTIVE FUNCTION IS TO BE:
      1.  MINIMIZED
      2.  MAXIMIZED
ENTER CHOICE -> 1


OBJECTIVE FUNCTION:

MINIMIZATION OF THE
I.L. HOOP STRESS (FIBER/I.L.)

IS INFORMATION CORRECT? <Y/N> y
```

Note: Enter total number of constraints. Each inequality constraint counts as one. Create equality constraints by selecting both greater than and less than constraints. Enter 0 for an unconstrained problem.

```
              CHOOSE DESIRED CONSTRAINTS:

ITEM#           FUNCTION                               CONSTRAINT

          INTERFACIAL LAYER FUNCTIONS
   1.   HOOP STRESS (FIBER/I.L.)             1.  >  (NOT TO BE LESS THAN)
   2.   HOOP STRESS (I.L./MATRIX)            2.  <  (NOT TO EXCEED)
   3.   RADIAL STRESS (I.L./MATRIX)
   4.   RADIAL STRESS (FIBER/I.L.)
   5.   LONGITUDINAL STRESS (AVERAGE)

          MATRIX FUNCTIONS
   6.   HOOP STRESS (INTERFACE)
   7.   RADIAL STRESS (INTERFACE)
   8.   HYDROSTATIC PRESSURE (INTERFACE)
   9.   LONGITUDINAL STRESS (INTERFACE)
  10.   LONGITUDINAL STRESS (AVERAGE)

          MISCELLANEOUS FUNCTIONS
  11.   LONGITUDINAL STRAIN (ASSEMBLAGE)
  12.   USER DEFINED CONSTRAINT FUNCTION

  ENTER NUMBER OF SELECTIONS (ENTER 0 FOR NO CONSTRAINTS) -> 0

       CONSTRAINTS:

       NO CONSTRAINTS

       IS INFORMATION CORRECT? <Y/N> y

  WOULD YOU LIKE TO SEE A PROBLEM REVIEW? <Y/N> y
```

Note: The problem review information is written to the file optcomp2.review.

******************************PROBLEM REVIEW******************************

CONCENTRIC CYLINDER GEOMETRY

LAYER	MATERIAL	NORMALIZED OUTER RADIUS	VOLUME FRACTION
1	FIBER	0.63246	0.4000
2	INTERFACE 1	0.73245	0.1365
3	MATRIX	1.00000	0.4635

FIBER VOLUME FRACTION = 0.4000
INTERFACE VOLUME FRACTION = 0.1365
MATRIX VOLUME FRACTION = 0.4635

HIT RETURN TO CONTINUE ->

CONCENTRIC CYLINDER MATERIAL SPECIFICATION

LAYER	INCLUSION	MATRIX	FVF
FIBER CORE	------	SiC (SCS-6)	-----
INTERFACE LAYER 1	------	Ti-24Al-11Nb	-----
MATRIX LAYER	------	Ti-24Al-11Nb	-----

HIT RETURN TO CONTINUE ->

LOAD HISTORY

STEP	TIME	INCREMENTS	TEMPERATURE	PRESSURE	AXIAL FORCE	AXIAL STRAIN
1			815.0	0.00	0.00	
	1.0000	500				
2			815.0	-200.00	0.00	
	1.0000	500				
3			430.0	-200.00	0.00	
	1.0000	500				
4			24.0	-200.00	0.00	
	1.0000	500				
5			24.0	0.00	0.00	

MAXIMUM NUMBER OF ITERATIONS = 15
CONVERGENCE ERROR TOLERANCE = 0.01000
CONVERGENCE INFORMATION WRITTEN TO optcomp2.conv
OPTIMIZATION ITERATIONS WRITTEN TO SCREEN

HIT RETURN TO CONTINUE ->

DESIGN VARIABLE SUMMARY

EXTERNAL PRESSURE DESIGN RANGE

STEP	LOWER BOUND	UPPER BOUND
1	0.00	0.00
2	-200.00	0.00
3	-375.00	0.00
4	-375.00	0.00
5	0.00	0.00

HIT RETURN TO CONTINUE ->

OBJECTIVE FUNCTION

MINIMIZATION OF THE
I.L. HOOP STRESS (FIBER/I.L.)

CONSTRAINTS:
NO CONSTRAINTS

```
HIT RETURN TO CONTINUE ->

          ******MAIN MENU******

        1.  CREATE NEW DATA FILE
        2.  RUN EXISTING DATA FILE
        3.  ENTER NEW MATERIALS INTO DATABANK
        4.  EXIT SHELL
       ENTER CHOICE -> 4
```

7.3.2 Execution of the optcomp2.data file

The execution of the data file **optcomp2.data,** whose contruction has been outlined in Section 7.3.1, is presented below as it is written to the screen during the actual optimization run. The information presented, excluding the header and the initial menu, is written independently to the file **optcomp2.history**.

```
    NASA-LEWIS RESEARCH CENTER, STRUCTURES DIVISION, CLEVELAND OHIO
          ****************************************************
          ***                    OPTCOMP2                  ***
          ***                                              ***
          ***   CONCENTRIC CYLINDER OPTIMIZATION PROGRAM   ***
          *** FOR THE DETERMINATION OF IDEALIZED INTERFACE ***
          ***     MICROSTRUCTURE AND PROCESSING HISTORY    ***
          ***                                              ***
          ***                   WRITTEN BY                 ***
          ***                                              ***
          ***              ROBERT SCOTT SALZAR             ***
          ***              MAREK-JERZY PINDERA             ***
          ***                                              ***
          ***          THE UNIVERSITY OF VIRGINIA          ***
          ***                  JUNE 1995                   ***
          ***                                              ***
          ***    DEVELOPED FOR THE FATIGUE AND FRACTURE    ***
          ***     BRANCH OF NASA-LEWIS RESEARCH CENTER     ***
          ***          UNDER CONTRACT NAS3-26571           ***
          ***     DR. S. M. ARNOLD  (CONTRACT MONITOR)     ***
          ****************************************************

==================================LEGAL NOTICE=================================

Neither NASA, nor any of its employees, contractors or grantees, nor any
person acting on behalf of either:
a.  makes any warranty or representation, expressed or implied, with respect
    to the accuracy, completeness, or usefulness of this software, or that the
    use of this software may not infringe privately owned rights; or

b.  assumes any liabilities with respect to the use of, or for damages
    resulting from use of this software.

==============================================================================


          ******MAIN MENU******

        1.  CREATE NEW DATA FILE
        2.  RUN EXISTING DATA FILE
        3.  ENTER NEW MATERIALS INTO DATABANK
        4.  EXIT SHELL
       ENTER CHOICE -> 2

***************LEGEND FOR DESIGN VARIABLES**********
 X 1 = EXTERNAL PRESSURE FOR STEP  1
 X 2 = EXTERNAL PRESSURE FOR STEP  2
 X 3 = EXTERNAL PRESSURE FOR STEP  3
 X 4 = EXTERNAL PRESSURE FOR STEP  4
 X 5 = EXTERNAL PRESSURE FOR STEP  5

 ITERATION #:      1
DESIGN VARIABLE     LOWER BOUND     CURRENT VALUE    UPPER BOUND    OBJECTIVE FN
     X 1              0.0000           0.0000          0.0000
     X 2           -200.0000        -200.0000          0.0000
     X 3           -375.0000        -200.0000          0.0000
```

X 4	-375.0000	-200.0000	0.0000	
X 5	0.0000	0.0000	0.0000	
				270.78708
ITERATION #:	2			
DESIGN VARIABLE	LOWER BOUND	CURRENT VALUE	UPPER BOUND	OBJECTIVE FN
X 1	0.0000	0.0000	0.0000	
X 2	-200.0000	-200.0000	0.0000	
X 3	-375.0000	-200.0000	0.0000	
X 4	-375.0000	-200.0000	0.0000	
X 5	0.0000	0.0000	0.0000	
				270.78708
ITERATION #:	3			
DESIGN VARIABLE	LOWER BOUND	CURRENT VALUE	UPPER BOUND	OBJECTIVE FN
X 1	0.0000	-0.0001	0.0000	
X 2	-200.0000	-200.0000	0.0000	
X 3	-375.0000	-200.0000	0.0000	
X 4	-375.0000	-200.0000	0.0000	
X 5	0.0000	0.0000	0.0000	
				270.78708
ITERATION #:	4			
DESIGN VARIABLE	LOWER BOUND	CURRENT VALUE	UPPER BOUND	OBJECTIVE FN
X 1	0.0000	0.0000	0.0000	
X 2	-200.0000	-199.8000	0.0000	
X 3	-375.0000	-200.0000	0.0000	
X 4	-375.0000	-200.0000	0.0000	
X 5	0.0000	0.0000	0.0000	
				270.78357
ITERATION #:	5			
DESIGN VARIABLE	LOWER BOUND	CURRENT VALUE	UPPER BOUND	OBJECTIVE FN
X 1	0.0000	0.0000	0.0000	
X 2	-200.0000	-200.0000	0.0000	
X 3	-375.0000	-199.8000	0.0000	
X 4	-375.0000	-200.0000	0.0000	
X 5	0.0000	0.0000	0.0000	
				270.76797
ITERATION #:	6			
DESIGN VARIABLE	LOWER BOUND	CURRENT VALUE	UPPER BOUND	OBJECTIVE FN
X 1	0.0000	0.0000	0.0000	
X 2	-200.0000	-200.0000	0.0000	
X 3	-375.0000	-200.0000	0.0000	
X 4	-375.0000	-199.8000	0.0000	
X 5	0.0000	0.0000	0.0000	
				270.85623
ITERATION #:	7			
DESIGN VARIABLE	LOWER BOUND	CURRENT VALUE	UPPER BOUND	OBJECTIVE FN
X 1	0.0000	0.0000	0.0000	
X 2	-200.0000	-200.0000	0.0000	
X 3	-375.0000	-200.0000	0.0000	
X 4	-375.0000	-200.0000	0.0000	
X 5	0.0000	-0.0001	0.0000	
				270.78702
ITERATION #:	8			
DESIGN VARIABLE	LOWER BOUND	CURRENT VALUE	UPPER BOUND	OBJECTIVE FN
X 1	0.0000	0.0000	0.0000	
X 2	-200.0000	-196.3161	0.0000	
X 3	-375.0000	-179.9468	0.0000	
X 4	-375.0000	-272.5887	0.0000	
X 5	0.0000	0.0000	0.0000	
				243.87105
ITERATION #:	9			
DESIGN VARIABLE	LOWER BOUND	CURRENT VALUE	UPPER BOUND	OBJECTIVE FN
X 1	0.0000	0.0000	0.0000	
X 2	-200.0000	-190.3555	0.0000	
X 3	-375.0000	-147.5000	0.0000	
X 4	-375.0000	-375.0000	0.0000	
X 5	0.0000	0.0000	0.0000	
				206.31479
ITERATION #:	10			
DESIGN VARIABLE	LOWER BOUND	CURRENT VALUE	UPPER BOUND	OBJECTIVE FN
X 1	0.0000	0.0000	0.0000	
X 2	-200.0000	-174.7502	0.0000	

X 3	-375.0000	-62.5533	0.0000	
X 4	-375.0000	-375.0000	0.0000	
X 5	0.0000	0.0000	0.0000	
				201.07736

ITERATION #: 11

DESIGN VARIABLE	LOWER BOUND	CURRENT VALUE	UPPER BOUND	OBJECTIVE FN
X 1	0.0000	0.0000	0.0000	
X 2	-200.0000	-133.8952	0.0000	
X 3	-375.0000	0.0000	0.0000	
X 4	-375.0000	-375.0000	0.0000	
X 5	0.0000	0.0000	0.0000	
				198.17862

ITERATION #: 12

DESIGN VARIABLE	LOWER BOUND	CURRENT VALUE	UPPER BOUND	OBJECTIVE FN
X 1	0.0000	0.0000	0.0000	
X 2	-200.0000	-26.9354	0.0000	
X 3	-375.0000	0.0000	0.0000	
X 4	-375.0000	-375.0000	0.0000	
X 5	0.0000	0.0000	0.0000	
				198.17862

ITERATION #: 13

DESIGN VARIABLE	LOWER BOUND	CURRENT VALUE	UPPER BOUND	OBJECTIVE FN
X 1	0.0000	0.0000	0.0000	
X 2	-200.0000	-145.5201	0.0000	
X 3	-375.0000	0.0000	0.0000	
X 4	-375.0000	-375.0000	0.0000	
X 5	0.0000	0.0000	0.0000	
				198.17862

ITERATION #: 14

DESIGN VARIABLE	LOWER BOUND	CURRENT VALUE	UPPER BOUND	OBJECTIVE FN
X 1	0.0000	-0.0001	0.0000	
X 2	-200.0000	-145.5201	0.0000	
X 3	-375.0000	0.0000	0.0000	
X 4	-375.0000	-375.0000	0.0000	
X 5	0.0000	0.0000	0.0000	
				198.17862

ITERATION #: 15

DESIGN VARIABLE	LOWER BOUND	CURRENT VALUE	UPPER BOUND	OBJECTIVE FN
X 1	0.0000	0.0000	0.0000	
X 2	-200.0000	-145.3746	0.0000	
X 3	-375.0000	0.0000	0.0000	
X 4	-375.0000	-375.0000	0.0000	
X 5	0.0000	0.0000	0.0000	
				198.17862

ITERATION #: 16

DESIGN VARIABLE	LOWER BOUND	CURRENT VALUE	UPPER BOUND	OBJECTIVE FN
X 1	0.0000	0.0000	0.0000	
X 2	-200.0000	-145.5201	0.0000	
X 3	-375.0000	-0.0001	0.0000	
X 4	-375.0000	-375.0000	0.0000	
X 5	0.0000	0.0000	0.0000	
				198.17862

ITERATION #: 17

DESIGN VARIABLE	LOWER BOUND	CURRENT VALUE	UPPER BOUND	OBJECTIVE FN
X 1	0.0000	0.0000	0.0000	
X 2	-200.0000	-145.5201	0.0000	
X 3	-375.0000	0.0000	0.0000	
X 4	-375.0000	-374.6250	0.0000	
X 5	0.0000	0.0000	0.0000	
				198.30333

ITERATION #: 18

DESIGN VARIABLE	LOWER BOUND	CURRENT VALUE	UPPER BOUND	OBJECTIVE FN
X 1	0.0000	0.0000	0.0000	
X 2	-200.0000	-145.5201	0.0000	
X 3	-375.0000	0.0000	0.0000	
X 4	-375.0000	-375.0000	0.0000	
X 5	0.0000	-0.0001	0.0000	
				198.17856

7.3.3 Results of the time-independent process history optimization

The file **optcomp2.out**, containing information on the material properties of the fiber, interfacial layer(s) and matrix (or their constituents if these have been specified as heterogeneous), and initial and final (optimum) concentric cylinder make-up, load history, stresses and inelastic strains, for the data file **optcomp2.data** constructed in Section 7.3.1, is given below.

```
NASA-LEWIS RESEARCH CENTER, STRUCTURES DIVISION, CLEVELAND OHIO
            ****************************************************
            ***                   OPTCOMP2                   ***
            ***                                              ***
            ***   CONCENTRIC CYLINDER OPTIMIZATION PROGRAM   ***
            *** FOR THE DETERMINATION OF IDEALIZED INTERFACE ***
            ***     MICROSTRUCTURE AND PROCESSING HISTORY    ***
            ***                                              ***
            ***                  WRITTEN BY                  ***
            ***                                              ***
            ***              ROBERT SCOTT SALZAR             ***
            ***              MAREK-JERZY PINDERA             ***
            ***                                              ***
            ***          THE UNIVERSITY OF VIRGINIA          ***
            ***                  JUNE 1995                   ***
            ***                                              ***
            ***     DEVELOPED FOR THE FATIGUE AND FRACTURE   ***
            ***      BRANCH OF NASA-LEWIS RESEARCH CENTER    ***
            ***          UNDER CONTRACT NAS3-26571           ***
            ***      DR. S. M. ARNOLD  (CONTRACT MONITOR)    ***
            ****************************************************
```

Note: The material properties of the individual layers of the specified concentric cylinder (i.e., fiber, interfacial layer with the same properties as the surrounding matrix, and the matrix), initial make-up of the concentric cylinder, initial load history and the resulting stresses and inelastic strains are given below.

```
Inelastic model (VPFLAG = 1) : Classical Plasticity

Units in MPa, degree C, and seconds


MATERIAL # 1

TEMPERATURE =  0.8150E+03

0.3999E+06  0.3999E+06  0.3999E+06       EXX     ETT     ERR
0.2500E+00  0.2500E+00  0.2500E+00       VXR     VXT     VRT
0.4499E-05  0.4499E-05  0.4499E-05       ALFXX   ALFTT   ALFRR
0.6895E+05  0.3999E+06                   Y       HS

.
.
.

TEMPERATURE =  0.2400E+02

0.3999E+06  0.3999E+06  0.3999E+06
0.2500E+00  0.2500E+00  0.2500E+00
0.3528E-05  0.3528E-05  0.3528E-05
0.6895E+05  0.3999E+06
```

MATERIAL # 2

TEMPERATURE = 0.8150E+03

```
0.4281E+05  0.4281E+05  0.4281E+05
0.2600E+00  0.2600E+00  0.2600E+00
0.1107E-04  0.1107E-04  0.1107E-04
0.1658E+03  0.1107E+03
```

.
.
.

TEMPERATURE = 0.2400E+02

```
0.1103E+06  0.1103E+06  0.1103E+06
0.2600E+00  0.2600E+00  0.2600E+00
0.9000E-05  0.9000E-05  0.9000E-05
0.3716E+03  0.2297E+05
```

MATERIAL # 3

TEMPERATURE = 0.8150E+03

```
0.4281E+05  0.4281E+05  0.4281E+05
0.2600E+00  0.2600E+00  0.2600E+00
0.1107E-04  0.1107E-04  0.1107E-04
0.1658E+03  0.1107E+03
```

.
.
.

TEMPERATURE = 0.2400E+02

```
0.1103E+06  0.1103E+06  0.1103E+06
0.2600E+00  0.2600E+00  0.2600E+00
0.9000E-05  0.9000E-05  0.9000E-05
0.3716E+03  0.2297E+05
```

INITIAL LAMINATED CYLINDER CONFIGURATION

LAYER	OUTER RADIUS	INCLUSION	MATRIX	FVF	ASP. RATIO
1	0.6325	-----	1	----	----
2	0.7325	-----	2	----	----
3	1.0000	-----	3	----	----

INITIAL LOAD HISTORY

STEP	TIME	INCREMENTS	TEMPERATURE	PRESSURE	AXIAL STRESS	AXIAL STRAIN
1			815.00	0.00	0.00	
	1.0000	500				
2			815.00	-200.00	0.00	
	1.0000	500				
3			430.00	-200.00	0.00	
	1.0000	500				
4			24.00	-200.00	0.00	
	1.0000	500				
5			24.00	0.00	0.00	

INITIAL STRESSES AND INELASTIC STRAINS

Time = 0.4000E+01
Temperature = 0.2400E+02
Radial traction = 0.8603E-12
Axial strain = -0.3961E-02
Axial stress = -0.5450E-04

RING NO.	RADIUS	STRXX	STRRR	STRTT	W
1	0.0000E+00	-0.4140E+03	-0.1469E+03	-0.1469E+03	0.0000E+00
1	0.6325E+00	-0.4140E+03	-0.1469E+03	-0.1469E+03	-0.1977E-02
2	0.6325E+00	0.1785E+03	-0.1469E+03	0.2708E+03	-0.1977E-02
2	0.6825E+00	0.2114E+03	-0.1164E+03	0.2679E+03	
2	0.7325E+00	0.2398E+03	-0.9027E+02	0.2636E+03	-0.3417E-02
3	0.7325E+00	0.2398E+03	-0.9027E+02	0.2636E+03	-0.3417E-02
3	0.7860E+00	0.2655E+03	-0.6637E+02	0.2581E+03	
3	0.8395E+00	0.2870E+03	-0.4589E+02	0.2516E+03	
3	0.8930E+00	0.3049E+03	-0.2829E+02	0.2441E+03	
3	0.9465E+00	0.3196E+03	-0.1312E+02	0.2361E+03	
3	0.1000E+01	0.3316E+03	0.2558E-12	0.2280E+03	-0.6611E-02

RING NO.	EPXXP	EPRRP	EPTTP	EPEFF	STREFF	SIGEFF
1	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.2672E+03	0.0000E+00
1	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.2672E+03	0.0000E+00
2	0.2674E-02	-0.5129E-02	0.2455E-02	0.5130E-02	0.3800E+03	0.5206E+03
2	0.2441E-02	-0.4209E-02	0.1769E-02	0.4227E-02	0.3593E+03	0.4943E+03
2	0.2235E-02	-0.3494E-02	0.1260E-02	0.3540E-02	0.3426E+03	0.4743E+03
3	0.2235E-02	-0.3494E-02	0.1260E-02	0.3540E-02	0.3426E+03	0.4743E+03
3	0.2046E-02	-0.2900E-02	0.8545E-03	0.2981E-02	0.3282E+03	0.4581E+03
3	0.1883E-02	-0.2441E-02	0.5574E-03	0.2558E-02	0.3167E+03	0.4458E+03
3	0.1745E-02	-0.2086E-02	0.3408E-03	0.2238E-02	0.3073E+03	0.4365E+03
3	0.1628E-02	-0.1811E-02	0.1822E-03	0.1994E-02	0.2998E+03	0.4295E+03
3	0.1532E-02	-0.1597E-02	0.6489E-04	0.1807E-02	0.2938E+03	0.4240E+03

Note: The final make-up of the concentric cylinder, final load history and the resulting stresses and inelastic strains are given below.

FINAL LAMINATED CYLINDER CONFIGURATION

LAYER	OUTER RADIUS	INCLUSION	MATRIX	FVF	ASP. RATIO
1	0.6325	-----	1	----	----
2	0.7325	-----	2	----	----
3	1.0000	-----	3	----	----

FINAL LOAD HISTORY

STEP	TIME	INCREMENTS	TEMPERATURE	PRESSURE	AXIAL STRESS	AXIAL STRAIN
1			815.00	0.00	0.00	
	1.0000	500				
2			815.00	-145.52	0.00	
	1.0000	500				
3			430.00	0.00	0.00	
	1.0000	500				
4			24.00	-375.00	0.00	
	1.0000	500				
5			24.00	0.00	0.00	

FINAL STRESSES AND INELASTIC STRAINS

Time = 0.4000E+01
Temperature = 0.2400E+02
Radial traction = 0.5662E-13
Axial strain = -0.3695E-02
Axial stress = -0.2233E-04

RING NO.	RADIUS	STRXX	STRRR	STRTT	W
1	0.0000E+00	-0.2902E+03	-0.1124E+03	-0.1124E+03	0.0000E+00
1	0.6325E+00	-0.2902E+03	-0.1124E+03	-0.1124E+03	-0.1985E-02
2	0.6325E+00	0.1246E+03	-0.1124E+03	0.1982E+03	-0.1985E-02
2	0.6825E+00	0.1470E+03	-0.8962E+02	0.1998E+03	
2	0.7325E+00	0.1668E+03	-0.6986E+02	0.1998E+03	-0.3491E-02
3	0.7325E+00	0.1668E+03	-0.6986E+02	0.1998E+03	-0.3491E-02
3	0.7860E+00	0.1852E+03	-0.5156E+02	0.1979E+03	
3	0.8395E+00	0.2010E+03	-0.3576E+02	0.1945E+03	
3	0.8930E+00	0.2143E+03	-0.2210E+02	0.1899E+03	
3	0.9465E+00	0.2253E+03	-0.1027E+02	0.1845E+03	
3	0.1000E+01	0.2343E+03	-0.6253E-12	0.1788E+03	-0.6864E-02

RING NO.	EPXXP	EPRRP	EPTTP	EPEFF	STREFF	SIGEFF
1	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.1777E+03	0.0000E+00
1	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.1777E+03	0.0000E+00
2	0.3339E-02	-0.6394E-02	0.3055E-02	0.6396E-02	0.2812E+03	0.5574E+03
2	0.3194E-02	-0.5430E-02	0.2237E-02	0.5458E-02	0.2670E+03	0.5301E+03
2	0.3061E-02	-0.4674E-02	0.1612E-02	0.4748E-02	0.2547E+03	0.5095E+03
3	0.3061E-02	-0.4674E-02	0.1612E-02	0.4748E-02	0.2547E+03	0.5095E+03
3	0.2933E-02	-0.4038E-02	0.1105E-02	0.4174E-02	0.2434E+03	0.4928E+03
3	0.2819E-02	-0.3541E-02	0.7216E-03	0.3742E-02	0.2336E+03	0.4802E+03
3	0.2720E-02	-0.3149E-02	0.4295E-03	0.3416E-02	0.2252E+03	0.4707E+03
3	0.2635E-02	-0.2840E-02	0.2045E-03	0.3168E-02	0.2180E+03	0.4635E+03
3	0.2564E-02	-0.2593E-02	0.2871E-04	0.2978E-02	0.2121E+03	0.4580E+03

The file **optcomp2.conv**, containing convergence messages at each optimization iteration, is given below.

```
OPTIMIZATION ITERATION #    1
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #    2
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #    3
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #    4
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #    5
ALL POINTS REACHED CONVERGENCE

.
.
.
.
.

OPTIMIZATION ITERATION #   14
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #   15
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #   16
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #   17
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #   18
ALL POINTS REACHED CONVERGENCE
```

7.4 Appendix IV: Example 2 - Time-Dependent Process History Optimization

7.4.1 Construction and execution of the optcomp2.data file

The construction of the **optcomp2.data** file for the time-dependent optimization problem of Example 2, menu-driven by the user-friendly interface **shell.f**, is illustrated below. The text that appears in Courier-type capital letters is written to the screen at each step in the construction of the **optcomp2.data** file. User's responses to the menu-driven commands are shown in bold Courier-type letters.

```
NASA-LEWIS RESEARCH CENTER, STRUCTURES DIVISION, CLEVELAND OHIO
          ***************************************************
          ***                  OPTCOMP2                   ***
          ***                                             ***
          ***   CONCENTRIC CYLINDER OPTIMIZATION PROGRAM  ***
          *** FOR THE DETERMINATION OF IDEALIZED INTERFACE ***
          ***     MICROSTRUCTURE AND PROCESSING HISTORY   ***
          ***                                             ***
          ***                 WRITTEN BY                  ***
          ***                                             ***
          ***             ROBERT SCOTT SALZAR             ***
          ***             MAREK-JERZY PINDERA             ***
          ***                                             ***
          ***         THE UNIVERSITY OF VIRGINIA          ***
          ***                 JUNE 1995                   ***
          ***                                             ***
          ***    DEVELOPED FOR THE FATIGUE AND FRACTURE   ***
          ***     BRANCH OF NASA-LEWIS RESEARCH CENTER    ***
          ***          UNDER CONTRACT NAS3-26571          ***
          ***     DR. S. M. ARNOLD  (CONTRACT MONITOR)    ***
          ***************************************************

==================================LEGAL NOTICE================================

Neither NASA, nor any of its employees, contractors or grantees, nor any
person acting on behalf of either:
a.  makes any warranty or representation, expressed or implied, with respect
    to the accuracy, completeness, or usefulness of this software, or that the
    use of this software may not infringe privately owned rights; or

b.  assumes any liabilities with respect to the use of, or for damages
    resulting from use of this software.

=============================================================================


          ******MAIN MENU******

        1.  CREATE NEW DATA FILE
        2.  RUN EXISTING DATA FILE
        3.  ENTER NEW MATERIALS INTO DATABANK
        4.  EXIT SHELL
      ENTER CHOICE -> 1

 **********************************BLOCK 1**********************************
               SPECIFY CONCENTRIC CYLINDER GEOMETRY, MATERIALS
                           INCLUDING PROPERTIES


      DOES THE FIBER HAVE LAYERED MORPHOLOGY? <Y/N> n
```

```
ENTER RADIUS OF FIBER CORE -> 0.5916
ENTER FIBER VOLUME FRACTION -> 0.35

ENTER THE NUMBER OF INTERFACE LAYERS -> 1

ENTER THICKNESS OF INTERFACE 1 -> 0.10


          ***CONCENTRIC CYLINDER GEOMETRY***

 LAYER          MATERIAL      NORMALIZED OUTER RADIUS     VOLUME FRACTION

   1             FIBER                0.59161                 0.3500
   2          INTERFACE 1             0.69161                 0.1283
   3             MATRIX               1.00000                 0.5217

 FIBER VOLUME FRACTION =        0.3500
 INTERFACE VOLUME FRACTION =    0.1283
 MATRIX VOLUME FRACTION =       0.5217

  IS INFORMATION CORRECT? <Y/N> y

       ***COMPOSITE MICRO-STRUCTURE***


     IF FIBER CORE IS HOMOGENEOUS,   ENTER 1
     IF FIBER CORE IS HETEROGENEOUS, ENTER 2
     -> 1

     IF INTERFACE LAYER  1 IS HOMOGENEOUS,  ENTER 1
     IF INTERFACE LAYER  1 IS HETEROGENEOUS, ENTER 2
     -> 1

     IF MATRIX LAYER IS HOMOGENEOUS,  ENTER 1
     IF MATRIX LAYER IS HETEROGENEOUS, ENTER 2
     -> 1

 ***INELASTIC CONSTITUTIVE MODEL SELECTION***

      FOR CLASSICAL PLASTICITY, ENTER 1
      FOR BODNER-PARTOM,        ENTER 2
      FOR USER-DEFINED MODEL,   ENTER 3
      -> 3
ENTER NUMBER OF INELASTIC PARAMETERS USED IN USER-DEFINED SUBROUTINE -> 10

            ***MATERIAL PROPERTY SELECTION***

      AVAILABLE MATERIALS            AVAILABLE CONSTITUTIVE MODELS
   1   SiC                                 1  ELASTIC
   2   Ti-6Al-4V (low CTE)                 2  INELASTIC
   3   Ti-6Al-4V (high CTE)
   4   ENTER NEW MATERIAL

 ENTER MATERIAL FOR FIBER CORE -> 1

 ENTER MATERIAL FOR INTERFACE LAYER 1 -> 3
 ENTER CONSTITUTIVE MODEL FOR INTERFACE LAYER 1 -> 2

 ENTER MATERIAL FOR MATRIX LAYER -> 3
 ENTER CONSTITUTIVE MODEL FOR MATRIX LAYER -> 2


        ***LAMINATED CYLINDER MATERIAL SPECIFICATION***

    LAYER              INCLUSION            MATRIX                FVF

FIBER CORE              ------         SiC                      -----
INTERFACE LAYER  1      ------         Ti-6Al-4V (high CTE)     -----
MATRIX LAYER            ------         Ti-6Al-4V (high CTE)     -----
```

```
IS INFORMATION CORRECT? <Y/N> y


**********************************BLOCK 2**********************************
                  DEFINE PROCESSING/LOAD HISTORY, INCREMENT,
                               AND ITERATIONS


                                ***CAUTION***
THE APPLIED TEMPERATURE LOAD MUST REMAIN BETWEEN    21.00 deg AND  900.00 deg
                                *************

NUMBER OF LOAD SEGMENTS -> 1

IF FIRST LOAD SEGMENT IS UNDER STRESS CONTROL, ENTER 1
IF FIRST LOAD SEGMENT IS UNDER STRAIN CONTROL, ENTER 2
-> 1

INITIAL TEMPERATURE, INITIAL EXTERNAL PRESSURE, INITIAL AXIAL STRESS
-> 900 0 0
DURATION OF LOAD STEP,  NUMBER OF LOAD INCREMENTS
-> 0.3822 10000
ENDING TEMPERATURE, ENDING PRESSURE, ENDING AXIAL STRESS
-> 21 0 0


                                ***LOAD HISTORY***

STEP  TIME   INCREMENTS  TEMPERATURE  PRESSURE   AXIAL FORCE  AXIAL STRAIN
  1                         900.0       0.00          0.00
     0.3822      10000
  2                          21.0       0.00          0.00
  IS INFORMATION CORRECT? <Y/N> y


     ***SET INTERNAL VARIABLES***

 CHANGE MAXIMUM NUMBER OF ITERATIONS (DEFAULT=10)? <Y/N> y
 MAXIMUM NUMBER OF ITERATIONS -> 3

 CHANGE CONVERGENCE ERROR TOLERANCES (DEFAULT=0.01)? <Y/N> n

 CHANGE NUMBER OF INTEGRATION POINTS (DEFAULT= 21/LAYER) AND
 PRINT OPTIONS (DEFAULT= 2/LAYER) FROM DEFAULT VALUES? <Y/N> y

                          INT. POINTS      PRINT NUMBER
 LAYER  1                     2                2
 LAYER  2                     21               3
 LAYER  3                     151              6

 WRITE CONVERGENCE INFORMATION TO optcomp2.conv FILE? <Y/N> y

 WRITE OPTIMIZATION ITERATIONS TO SCREEN? <Y/N> y


 INTERNAL VARIABLE REVIEW

           MAXIMUM NUMBER OF ITERATIONS =      3
           CONVERGENCE ERROR TOLERANCE = 0.01000
           CONVERGENCE INFORMATION WRITTEN TO optcomp2.conv
           OPTIMIZATION ITERATIONS WRITTEN TO SCREEN

 IS INFORMATION CORRECT? <Y/N> y


**********************************BLOCK 3**********************************
                          DEFINE OPTIMIZATION PROBLEM
```

```
TO OPTIMIZE PROCESSING HISTORY,         ENTER 1
TO OPTIMIZE INTERFACE VOLUME FRACTION,  ENTER 2
-> 1

   ***LOAD HISTORY MENU***

I.D.   DESIGN VARIABLES
 1.      TEMPERATURE
 2.      EXTERNAL PRESSURE
 3.      AXIAL LOAD
 4.      TIME DURATION

ENTER NUMBER OF SELECTIONS -> 1
ENTER I.D. CHOICE(S) -> 4

ENTER LOWER AND UPPER BOUNDS FOR TIME DURATION 1 -> 0.3822 38.22

                    ***DESIGN VARIABLE SUMMARY***

              TIME DURATION DESIGN RANGE
            STEP       LOWER BOUND        UPPER BOUND
             1              0.38             38.22

  IS INFORMATION CORRECT? <Y/N> y


          CHOOSE AN OBJECTIVE FUNCTION:

  ITEM#          FUNCTION

              FIBER FUNCTIONS
   1. RADIAL STRESS (INTERFACE)

          INTERFACIAL LAYER FUNCTIONS
   2. HOOP STRESS (FIBER/I.L.)
   3. HOOP STRESS (AVERAGE)
   4. RADIAL STRESS (FIBER/I.L.)
   5. RADIAL STRESS (I.L./MATRIX)
   6. HYDROSTATIC PRESSURE (I.L./MATRIX)
   7. LONGITUDINAL STRESS (AVERAGE)

              MATRIX FUNCTIONS
   8. HOOP STRESS (INTERFACE)
   9. RADIAL STRESS (INTERFACE)
  10. RADIAL STRAIN (INTERFACE)
  11. HYDROSTATIC PRESSURE (INTERFACE)
  12. LONGITUDINAL STRESS (INTERFACE)
  13. LONGITUDINAL STRESS (AVERAGE)

            MISCELLANEOUS FUNCTIONS
  14. LONGITUDINAL STRAIN (ASSEMBLAGE)
  15. USER DEFINED OBJECTIVE FUNCTION
 ENTER CHOICE -> 2

               OBJECTIVE FUNCTION IS TO BE:
                     1.  MINIMIZED
                     2.  MAXIMIZED
               ENTER CHOICE -> 1


               OBJECTIVE FUNCTION:

               MINIMIZATION OF THE
               I.L. HOOP STRESS (FIBER/I.L.)

               IS INFORMATION CORRECT? <Y/N> y
```

```
            CHOOSE DESIRED CONSTRAINTS:

  ITEM#          FUNCTION                           CONSTRAINT

        INTERFACIAL LAYER FUNCTIONS
    1.  HOOP STRESS (FIBER/I.L.)              1.  >  (NOT TO BE LESS THAN)
    2.  HOOP STRESS (I.L./MATRIX)             2.  <  (NOT TO EXCEED)
    3.  RADIAL STRESS (I.L./MATRIX)
    4.  RADIAL STRESS (FIBER/I.L.)
    5.  LONGITUDINAL STRESS (AVERAGE)

        MATRIX FUNCTIONS
    6.  HOOP STRESS (INTERFACE)
    7.  RADIAL STRESS (INTERFACE)
    8.  HYDROSTATIC PRESSURE (INTERFACE)
    9.  LONGITUDINAL STRESS (INTERFACE)
   10.  LONGITUDINAL STRESS (AVERAGE)

         MISCELLANEOUS FUNCTIONS
   11.  LONGITUDINAL STRAIN (ASSEMBLAGE)
   12.  USER DEFINED CONSTRAINT FUNCTION

   ENTER NUMBER OF SELECTIONS (ENTER 0 FOR NO CONSTRAINTS) -> 0

          CONSTRAINTS:

          NO CONSTRAINTS

          IS INFORMATION CORRECT? <Y/N> y

   WOULD YOU LIKE TO SEE A PROBLEM REVIEW? <Y/N> n


          ******MAIN MENU******

        1.  CREATE NEW DATA FILE
        2.  RUN EXISTING DATA FILE
        3.  ENTER NEW MATERIALS INTO DATABANK
        4.  EXIT SHELL
       ENTER CHOICE -> 2



 ***************LEGEND FOR DESIGN VARIABLES**********
  X 1 = TIME DURATION FOR STEP  1

  ITERATION #:     1
 DESIGN VARIABLE    LOWER BOUND    CURRENT VALUE   UPPER BOUND   OBJECTIVE FN
      X 1             0.3822          0.3822         38.2200
                                                                  568.55139

  ITERATION #:     2
 DESIGN VARIABLE    LOWER BOUND    CURRENT VALUE   UPPER BOUND   OBJECTIVE FN
      X 1             0.3822          0.3822         38.2200
                                                                  568.55139

  ITERATION #:     3
 DESIGN VARIABLE    LOWER BOUND    CURRENT VALUE   UPPER BOUND   OBJECTIVE FN
      X 1             0.3822          0.3826         38.2200
                                                                  568.52911

  ITERATION #:     4
 DESIGN VARIABLE    LOWER BOUND    CURRENT VALUE   UPPER BOUND   OBJECTIVE FN
      X 1             0.3822          0.4586         38.2200
                                                                  564.50751

  ITERATION #:     5
 DESIGN VARIABLE    LOWER BOUND    CURRENT VALUE   UPPER BOUND   OBJECTIVE FN
      X 1             0.3822          0.5823         38.2200
                                                                  559.28369

  ITERATION #:     6
 DESIGN VARIABLE    LOWER BOUND    CURRENT VALUE   UPPER BOUND   OBJECTIVE FN
      X 1             0.3822          0.9061         38.2200
                                                                  549.81885
```

ITERATION #: 7

DESIGN VARIABLE	LOWER BOUND	CURRENT VALUE	UPPER BOUND	OBJECTIVE FN
X 1	0.3822	1.7539	38.2200	536.16644

ITERATION #: 8

DESIGN VARIABLE	LOWER BOUND	CURRENT VALUE	UPPER BOUND	OBJECTIVE FN
X 1	0.3822	3.9733	38.2200	520.01691

ITERATION #: 9

DESIGN VARIABLE	LOWER BOUND	CURRENT VALUE	UPPER BOUND	OBJECTIVE FN
X 1	0.3822	9.7837	38.2200	503.12756

ITERATION #: 10

DESIGN VARIABLE	LOWER BOUND	CURRENT VALUE	UPPER BOUND	OBJECTIVE FN
X 1	0.3822	24.9956	38.2200	486.49472

ITERATION #: 11

DESIGN VARIABLE	LOWER BOUND	CURRENT VALUE	UPPER BOUND	OBJECTIVE FN
X 1	0.3822	38.2200	38.2200	479.26535

ITERATION #: 12

DESIGN VARIABLE	LOWER BOUND	CURRENT VALUE	UPPER BOUND	OBJECTIVE FN
X 1	0.3822	38.1818	38.2200	479.28220

7.4.2 Results of the time-dependent process history optimization

The file **optcomp2.out**, containing information on the material properties of the fiber, interfacial layer(s) and matrix (or their constituents if these have been specified as heterogeneous), and initial and final (optimum) concentric cylinder make-up, load history, stresses and inelastic strains, for the data file **optcomp2.data** constructed in Section 7.4.1, is given below.

```
NASA-LEWIS RESEARCH CENTER, STRUCTURES DIVISION, CLEVELAND OHIO
       ****************************************************
       ***                 OPTCOMP2                    ***
       ***                                             ***
       ***   CONCENTRIC CYLINDER OPTIMIZATION PROGRAM  ***
       *** FOR THE DETERMINATION OF IDEALIZED INTERFACE ***
       ***     MICROSTRUCTURE AND PROCESSING HISTORY   ***
       ***                                             ***
       ***                WRITTEN BY                   ***
       ***                                             ***
       ***            ROBERT SCOTT SALZAR              ***
       ***            MAREK-JERZY PINDERA              ***
       ***                                             ***
       ***        THE UNIVERSITY OF VIRGINIA           ***
       ***                JUNE 1995                    ***
       ***                                             ***
       ***     DEVELOPED FOR THE FATIGUE AND FRACTURE  ***
       ***     BRANCH OF NASA-LEWIS RESEARCH CENTER    ***
       ***         UNDER CONTRACT NAS3-26571           ***
       ***     DR. S. M. ARNOLD  (CONTRACT MONITOR)    ***
       ****************************************************


Inelastic model (VPFLAG = 3)  : Power-Law Creep

Units in MPa, degree C, and seconds


MATERIAL # 1

TEMPERATURE =   0.9000E+03

0.4140E+06  0.4140E+06  0.4140E+06
0.3000E+00  0.3000E+00  0.3000E+00
0.4860E-05  0.4860E-05  0.4860E-05
0.1000E+01  0.1000E+01  0.1000E+01  0.1000E+01  0.0000E+00
0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00

 .
 .
 .

TEMPERATURE =   0.2100E+02

0.4140E+06  0.4140E+06  0.4140E+06
0.3000E+00  0.3000E+00  0.3000E+00
0.4860E-05  0.4860E-05  0.4860E-05
0.1000E+01  0.1000E+01  0.1000E+01  0.1000E+01  0.0000E+00
0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00
```

MATERIAL # 2

TEMPERATURE = 0.9000E+03

0.2070E+05	0.2070E+05	0.2070E+05		
0.3000E+00	0.3000E+00	0.3000E+00		
0.1240E-04	0.1240E-04	0.1240E-04		
0.3600E+10	0.3403E+01	0.9251E+00	0.3600E+05	0.0000E+00
0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00

.
.
.

TEMPERATURE = 0.2100E+02

0.1137E+06	0.1137E+06	0.1137E+06		
0.3000E+00	0.3000E+00	0.3000E+00		
0.1100E-04	0.1100E-04	0.1100E-04		
0.3600E+10	0.3403E+01	0.9251E+00	0.3600E+05	0.0000E+00
0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00

MATERIAL # 3

TEMPERATURE = 0.9000E+03

0.2070E+05	0.2070E+05	0.2070E+05		
0.3000E+00	0.3000E+00	0.3000E+00		
0.1240E-04	0.1240E-04	0.1240E-04		
0.3600E+10	0.3403E+01	0.9251E+00	0.3600E+05	0.0000E+00
0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00

.
.
.

TEMPERATURE = 0.2100E+02

0.1137E+06	0.1137E+06	0.1137E+06		
0.3000E+00	0.3000E+00	0.3000E+00		
0.1100E-04	0.1100E-04	0.1100E-04		
0.3600E+10	0.3403E+01	0.9251E+00	0.3600E+05	0.0000E+00
0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00

INITIAL LAMINATED CYLINDER CONFIGURATION

LAYER	OUTER RADIUS	INCLUSION	MATRIX	FVF	ASP. RATIO
1	0.5916	-----	1	----	----
2	0.6916	-----	2	----	----
3	1.0000	-----	3	----	----

INITIAL LOAD HISTORY

STEP	TIME	INCREMENTS	TEMPERATURE	PRESSURE	AXIAL STRESS	AXIAL STRAIN
1			900.00	0.00	0.00	
	0.3822	10000				
2			21.00	0.00	0.00	

INITIAL STRESSES AND INELASTIC STRAINS

Time = 0.3822E+00
Temperature = 0.2100E+02
Radial traction = 0.0000E+00

Axial strain = -0.5901E-02
Axial stress = -0.4433E-04

RING NO.	RADIUS	STRXX	STRRR	STRTT	W
1	0.0000E+00	-0.8458E+03	-0.2857E+03	-0.2857E+03	0.0000E+00
1	0.5916E+00	-0.8458E+03	-0.2857E+03	-0.2857E+03	-0.2450E-02
2	0.5916E+00	0.4272E+03	-0.2857E+03	0.5686E+03	-0.2450E-02
2	0.6416E+00	0.4369E+03	-0.2213E+03	0.5141E+03	
2	0.6916E+00	0.4447E+03	-0.1698E+03	0.4699E+03	-0.4213E-02
3	0.6916E+00	0.4447E+03	-0.1698E+03	0.4699E+03	-0.4213E-02
3	0.7533E+00	0.4524E+03	-0.1193E+03	0.4257E+03	
3	0.8150E+00	0.4582E+03	-0.7945E+02	0.3900E+03	
3	0.8766E+00	0.4626E+03	-0.4748E+02	0.3609E+03	
3	0.9383E+00	0.4660E+03	-0.2145E+02	0.3367E+03	
3	0.1000E+01	0.4686E+03	-0.1364E-11	0.3166E+03	-0.8827E-02

RING NO.	EPXXP	EPRRP	EPTTP	EPEFF	STREFF	SIGEFF
1	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.5601E+03	0.0000E+00
1	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.5601E+03	0.0000E+00
2	0.1516E-02	-0.3175E-02	0.1659E-02	0.3177E-02	0.7931E+03	0.0000E+00
2	0.1457E-02	-0.2699E-02	0.1242E-02	0.2702E-02	0.7000E+03	0.0000E+00
2	0.1407E-02	-0.2336E-02	0.9291E-03	0.2353E-02	0.6275E+03	0.0000E+00
3	0.1407E-02	-0.2336E-02	0.9291E-03	0.2353E-02	0.6275E+03	0.0000E+00
3	0.1357E-02	-0.2000E-02	0.6427E-03	0.2042E-02	0.5588E+03	0.0000E+00
3	0.1317E-02	-0.1749E-02	0.4319E-03	0.1822E-02	0.5070E+03	0.0000E+00
3	0.1285E-02	-0.1558E-02	0.2729E-03	0.1664E-02	0.4676E+03	0.0000E+00
3	0.1260E-02	-0.1411E-02	0.1501E-03	0.1550E-02	0.4374E+03	0.0000E+00
3	0.1241E-02	-0.1294E-02	0.5329E-04	0.1465E-02	0.4141E+03	0.0000E+00

FINAL LAMINATED CYLINDER CONFIGURATION

LAYER	OUTER RADIUS	INCLUSION	MATRIX	FVF	ASP. RATIO
1	0.5916	-----	1	----	----
2	0.6916	-----	2	----	----
3	1.0000	-----	3	----	----

FINAL LOAD HISTORY

STEP	TIME	INCREMENTS	TEMPERATURE	PRESSURE	AXIAL STRESS	AXIAL STRAIN
1			900.00	0.00	0.00	
	38.2200	10000				
2			21.00	0.00	0.00	

FINAL STRESSES AND INELASTIC STRAINS

Time = 0.3822E+02
Temperature = 0.2100E+02
Radial traction = 0.0000E+00
Axial strain = -0.5641E-02
Axial stress = -0.2483E-04

RING NO.	RADIUS	STRXX	STRRR	STRTT	W
1	0.0000E+00	-0.7108E+03	-0.2398E+03	-0.2398E+03	0.0000E+00
1	0.5916E+00	-0.7108E+03	-0.2398E+03	-0.2398E+03	-0.2462E-02
2	0.5916E+00	0.3617E+03	-0.2398E+03	0.4793E+03	-0.2462E-02
2	0.6416E+00	0.3689E+03	-0.1856E+03	0.4326E+03	
2	0.6916E+00	0.3747E+03	-0.1423E+03	0.3948E+03	-0.4284E-02
3	0.6916E+00	0.3747E+03	-0.1423E+03	0.3948E+03	-0.4284E-02
3	0.7533E+00	0.3804E+03	-0.9995E+02	0.3572E+03	
3	0.8150E+00	0.3848E+03	-0.6654E+02	0.3269E+03	

3	0.8766E+00	0.3881E+03	-0.3975E+02	0.3023E+03		
3	0.9383E+00	0.3906E+03	-0.1796E+02	0.2819E+03		
3	0.1000E+01	0.3925E+03	0.7958E-12	0.2649E+03	-0.9058E-02	

RING NO.	EPXXP	EPRRP	EPTTP	EPEFF	STREFF	SIGEFF
1	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.4710E+03	0.0000E+00
1	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.4710E+03	0.0000E+00
2	0.2238E-02	-0.4610E-02	0.2372E-02	0.4611E-02	0.6681E+03	0.0000E+00
2	0.2194E-02	-0.4001E-02	0.1808E-02	0.4008E-02	0.5890E+03	0.0000E+00
2	0.2157E-02	-0.3531E-02	0.1374E-02	0.3560E-02	0.5274E+03	0.0000E+00
3	0.2157E-02	-0.3531E-02	0.1374E-02	0.3560E-02	0.5274E+03	0.0000E+00
3	0.2119E-02	-0.3085E-02	0.9660E-03	0.3157E-02	0.4692E+03	0.0000E+00
3	0.2090E-02	-0.2746E-02	0.6568E-03	0.2869E-02	0.4254E+03	0.0000E+00
3	0.2066E-02	-0.2483E-02	0.4173E-03	0.2660E-02	0.3920E+03	0.0000E+00
3	0.2048E-02	-0.2276E-02	0.2280E-03	0.2507E-02	0.3665E+03	0.0000E+00
3	0.2033E-02	-0.2109E-02	0.7557E-04	0.2393E-02	0.3468E+03	0.0000E+00

The file **optcomp2.conv**, containing convergence messages at each optimization iteration, is given below.

```
OPTIMIZATION ITERATION #    1
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #    2
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #    3
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #    4
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #    5
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #    6
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #    7
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #    8
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #    9
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #   10
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #   11
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #   12
ALL POINTS REACHED CONVERGENCE
```

7.5 Appendix V: Example 3 - Plastic Strain Optimization Using Graded Interfaces

7.5.1 Construction and execution of the optcomp2.data file

The construction of the **optcomp2.data** file for the plastic strain optimization problem of Example 3, menu-driven by the user-friendly interface **shell.f**, is illustrated below. The text that appears in Courier-type capital letters is written to the screen at each step in the construction of the **optcomp2.data** file. User's responses to the menu-driven commands are shown in bold Courier-type letters.

```
NASA-LEWIS RESEARCH CENTER, STRUCTURES DIVISION, CLEVELAND OHIO
      ***************************************************
      ***                   OPTCOMP2                  ***
      ***                                             ***
      ***    CONCENTRIC CYLINDER OPTIMIZATION PROGRAM ***
      *** FOR THE DETERMINATION OF IDEALIZED INTERFACE ***
      ***     MICROSTRUCTURE AND PROCESSING HISTORY   ***
      ***                                             ***
      ***                  WRITTEN BY                 ***
      ***                                             ***
      ***              ROBERT SCOTT SALZAR            ***
      ***              MAREK-JERZY PINDERA            ***
      ***                                             ***
      ***          THE UNIVERSITY OF VIRGINIA         ***
      ***                  JUNE 1995                  ***
      ***                                             ***
      ***     DEVELOPED FOR THE FATIGUE AND FRACTURE  ***
      ***     BRANCH OF NASA-LEWIS RESEARCH CENTER    ***
      ***          UNDER CONTRACT NAS3-26571          ***
      ***     DR. S. M. ARNOLD  (CONTRACT MONITOR)    ***
      ***************************************************

=================================LEGAL NOTICE===============================

Neither NASA, nor any of its employees, contractors or grantees, nor any
person acting on behalf of either:
a.  makes any warranty or representation, expressed or implied, with respect
    to the accuracy, completeness, or usefulness of this software, or that the
    use of this software may not infringe privately owned rights; or

b.  assumes any liabilities with respect to the use of, or for damages
    resulting from use of this software.

============================================================================


          ******MAIN MENU******

         1.  CREATE NEW DATA FILE
         2.  RUN EXISTING DATA FILE
         3.  ENTER NEW MATERIALS INTO DATABANK
         4.  EXIT SHELL
        ENTER CHOICE -> 1

 **********************************BLOCK 1**********************************
             SPECIFY CONCENTRIC CYLINDER GEOMETRY, MATERIALS
                          INCLUDING PROPERTIES


        DOES THE FIBER HAVE LAYERED MORPHOLOGY? <Y/N> n
```

```
ENTER RADIUS OF FIBER CORE -> 0.63246
ENTER FIBER VOLUME FRACTION -> 0.40

ENTER THE NUMBER OF INTERFACE LAYERS -> 3

ENTER THICKNESS OF INTERFACE 1 -> 0.02108
ENTER THICKNESS OF INTERFACE 2 -> 0.02108
ENTER THICKNESS OF INTERFACE 3 -> 0.02108


          ***CONCENTRIC CYLINDER GEOMETRY***

LAYER          MATERIAL       NORMALIZED OUTER RADIUS      VOLUME FRACTION

  1              FIBER                 0.63246                  0.4000
  2           INTERFACE 1              0.65354                  0.0271
  3           INTERFACE 2              0.67462                  0.0280
  4           INTERFACE 3              0.69570                  0.0289
  5              MATRIX                1.00000                  0.5160

FIBER VOLUME FRACTION =            0.4000
INTERFACE VOLUME FRACTION =        0.0840
MATRIX VOLUME FRACTION =           0.5160

  IS INFORMATION CORRECT? <Y/N> y

       ***COMPOSITE MICRO-STRUCTURE***


    IF FIBER CORE IS HOMOGENEOUS,   ENTER 1
    IF FIBER CORE IS HETEROGENEOUS, ENTER 2
    -> 1

    IF INTERFACE LAYER  1 IS HOMOGENEOUS,   ENTER 1
    IF INTERFACE LAYER  1 IS HETEROGENEOUS, ENTER 2
    -> 2
    IF INTERFACE LAYER  2 IS HOMOGENEOUS,   ENTER 1
    IF INTERFACE LAYER  2 IS HETEROGENEOUS, ENTER 2
    -> 2
    IF INTERFACE LAYER  3 IS HOMOGENEOUS,   ENTER 1
    IF INTERFACE LAYER  3 IS HETEROGENEOUS, ENTER 2
    -> 2

    IF MATRIX LAYER IS HOMOGENEOUS,   ENTER 1
    IF MATRIX LAYER IS HETEROGENEOUS, ENTER 2
    -> 1

***INELASTIC CONSTITUTIVE MODEL SELECTION***

     FOR CLASSICAL PLASTICITY, ENTER 1
     FOR BODNER-PARTOM,        ENTER 2
     FOR USER-DEFINED MODEL,   ENTER 3
     -> 1

          ***MATERIAL PROPERTY SELECTION***

     AVAILABLE MATERIALS             AVAILABLE CONSTITUTIVE MODELS
  1   SiC (SCS-6)                          1  ELASTIC
  2   Al2O3                                2  PLASTIC
  3   Gr
  4   Ti-24Al-11Nb
  5   Ti-6Al-4V
  6   NiAl
  7   FeAl
  8   FeAl1
  9   Cu
 10   ENTER NEW MATERIAL

ENTER MATERIAL FOR FIBER CORE -> 1
```

```
ENTER INCLUSION PHASE MATERIAL FOR INTERFACE LAYER 1 -> 3
ENTER CONSTITUTIVE MODEL FOR INCLUSION PHASE  -> 1
ENTER MATRIX PHASE MATERIAL FOR INTERFACE LAYER 1 -> 9
ENTER CONSTITUTIVE MODEL FOR MATRIX PHASE  MATERIAL -> 2
ENTER INCLUSION VOLUME FRACTION FOR INTERFACE LAYER 1 -> 0.10
ENTER ASPECT RATIO FOR INCLUSION -> 1.0

ENTER INCLUSION PHASE MATERIAL FOR INTERFACE LAYER 2 -> 3
ENTER CONSTITUTIVE MODEL FOR INCLUSION PHASE  -> 1
ENTER MATRIX PHASE MATERIAL FOR INTERFACE LAYER 2 -> 9
ENTER CONSTITUTIVE MODEL FOR MATRIX PHASE  MATERIAL -> 2
ENTER INCLUSION VOLUME FRACTION FOR INTERFACE LAYER 2 -> 0.10
ENTER ASPECT RATIO FOR INCLUSION -> 1.0

ENTER INCLUSION PHASE MATERIAL FOR INTERFACE LAYER 3 -> 3
ENTER CONSTITUTIVE MODEL FOR INCLUSION PHASE  -> 1
ENTER MATRIX PHASE MATERIAL FOR INTERFACE LAYER 3 -> 9
ENTER CONSTITUTIVE MODEL FOR MATRIX PHASE  MATERIAL -> 2
ENTER INCLUSION VOLUME FRACTION FOR INTERFACE LAYER 3 -> 0.10
ENTER ASPECT RATIO FOR INCLUSION -> 1.0

ENTER MATERIAL FOR MATRIX LAYER -> 4
ENTER CONSTITUTIVE MODEL FOR MATRIX LAYER -> 2


           ***LAMINATED CYLINDER MATERIAL SPECIFICATION***

    LAYER                 INCLUSION              MATRIX               FVF

 FIBER CORE                 ------             SiC (SCS-6)           -----
 INTERFACE LAYER  1       Gr                   Cu                    0.100
 INTERFACE LAYER  2       Gr                   Cu                    0.100
 INTERFACE LAYER  3       Gr                   Cu                    0.100
 MATRIX LAYER               ------             Ti-24Al-11Nb          -----

         IS INFORMATION CORRECT? <Y/N> y


**********************************BLOCK 2**********************************
                  DEFINE PROCESSING/LOAD HISTORY, INCREMENT,
                               AND ITERATIONS


                               ***CAUTION***
THE APPLIED TEMPERATURE LOAD MUST REMAIN BETWEEN   24.00 deg AND  815.00 deg
                               *************

 NUMBER OF LOAD SEGMENTS -> 1

 IF FIRST LOAD SEGMENT IS UNDER STRESS CONTROL, ENTER 1
 IF FIRST LOAD SEGMENT IS UNDER STRAIN CONTROL, ENTER 2
 -> 1

 INITIAL TEMPERATURE, INITIAL EXTERNAL PRESSURE, INITIAL AXIAL STRESS
 -> 815 0 0
 DURATION OF LOAD STEP,  NUMBER OF LOAD INCREMENTS
 -> 1 791
 ENDING TEMPERATURE, ENDING PRESSURE, ENDING AXIAL STRESS
 -> 24 0 0


                               ***LOAD HISTORY***

 STEP  TIME   INCREMENTS  TEMPERATURE  PRESSURE   AXIAL FORCE  AXIAL STRAIN
  1                          815.0        0.00         0.00
       1.0000       791
  2                           24.0        0.00         0.00
  IS INFORMATION CORRECT? <Y/N> y
```

```
     ***SET INTERNAL VARIABLES***

 CHANGE MAXIMUM NUMBER OF ITERATIONS (DEFAULT=10)? <Y/N> y
 MAXIMUM NUMBER OF ITERATIONS -> 50

 CHANGE CONVERGENCE ERROR TOLERANCES (DEFAULT=0.01)? <Y/N> n

 CHANGE NUMBER OF INTEGRATION POINTS (DEFAULT= 21/LAYER) AND
 PRINT OPTIONS (DEFAULT= 2/LAYER) FROM DEFAULT VALUES? <Y/N> y

                         INT. POINTS     PRINT NUMBER
 LAYER  1                    2               2
 LAYER  2                    21              3
 LAYER  3                    21              3
 LAYER  4                    21              3
 LAYER  5                    151             6

 WRITE CONVERGENCE INFORMATION TO optcomp2.conv FILE? <Y/N> y

 WRITE OPTIMIZATION ITERATIONS TO SCREEN? <Y/N> y


 INTERNAL VARIABLE REVIEW

           MAXIMUM NUMBER OF ITERATIONS =    50
           CONVERGENCE ERROR TOLERANCE = 0.01000
           CONVERGENCE INFORMATION WRITTEN TO optcomp2.conv
           OPTIMIZATION ITERATIONS WRITTEN TO SCREEN

 IS INFORMATION CORRECT? <Y/N> y


**********************************BLOCK 3**********************************
                        DEFINE OPTIMIZATION PROBLEM


 TO OPTIMIZE PROCESSING HISTORY,         ENTER 1
 TO OPTIMIZE INTERFACE VOLUME FRACTION, ENTER 2
 -> 2


ENTER LOWER & UPPER INCLUSION VOLUME BOUNDS FOR INTERFACE LAYER 1 -> 0.0 0.30
ENTER LOWER & UPPER INCLUSION VOLUME BOUNDS FOR INTERFACE LAYER 2 -> 0.0 0.30
ENTER LOWER & UPPER INCLUSION VOLUME BOUNDS FOR INTERFACE LAYER 3 -> 0.0 0.30

                  ***DESIGN VARIABLE SUMMARY***

         HETEROGENEOUS INTERFACE VOLUME FRACTION
         LAYER       LOWER BOUND      UPPER BOUND
          1             0.00             0.30
          2             0.00             0.30
          3             0.00             0.30

 IS INFORMATION CORRECT? <Y/N> y


         CHOOSE AN OBJECTIVE FUNCTION:

 ITEM#          FUNCTION

           FIBER FUNCTIONS
  1. RADIAL STRESS (INTERFACE)

       INTERFACIAL LAYER FUNCTIONS
  2. HOOP STRESS (FIBER/I.L.)
  3. HOOP STRESS (AVERAGE)
  4. RADIAL STRESS (FIBER/I.L.)
  5. RADIAL STRESS (I.L./MATRIX)
```

```
  6. HYDROSTATIC PRESSURE (I.L./MATRIX)
  7. LONGITUDINAL STRESS (AVERAGE)

          MATRIX FUNCTIONS
  8. HOOP STRESS (INTERFACE)
  9. RADIAL STRESS (INTERFACE)
 10. RADIAL STRAIN (INTERFACE)
 11. HYDROSTATIC PRESSURE (INTERFACE)
 12. LONGITUDINAL STRESS (INTERFACE)
 13. LONGITUDINAL STRESS (AVERAGE)

        MISCELLANEOUS FUNCTIONS
 14. LONGITUDINAL STRAIN (ASSEMBLAGE)
 15. USER DEFINED OBJECTIVE FUNCTION
ENTER CHOICE -> 15

 TO USE OBJECTIVE FN IN SUBROUTINE EXTOBJ: ENTER 1

 TO CHANGE OBJ FN, EXIT SHELL PROGRAM EDIT,
 COMPILE, AND LINK SUBROUTINE EXTOBJ TO INCLUDE
 YOUR CHOICE OF OBJECTIVE FUNCTIONS: ENTER 2

-> 1

             OBJECTIVE FUNCTION IS TO BE:
                  1.  MINIMIZED
                  2.  MAXIMIZED
             ENTER CHOICE -> 1

             OBJECTIVE FUNCTION:

             MINIMIZATION OF THE
             USER DEFINED OBJECTIVE FUNCTION

             IS INFORMATION CORRECT? <Y/N> y


            CHOOSE DESIRED CONSTRAINTS:

 ITEM#          FUNCTION                         CONSTRAINT

      INTERFACIAL LAYER FUNCTIONS
  1.  HOOP STRESS (FIBER/I.L.)             1.  >  (NOT TO BE LESS THAN)
  2.  HOOP STRESS (I.L./MATRIX)            2.  <  (NOT TO EXCEED)
  3.  RADIAL STRESS (I.L./MATRIX)
  4.  RADIAL STRESS (FIBER/I.L.)
  5.  LONGITUDINAL STRESS (AVERAGE)

      MATRIX FUNCTIONS
  6.  HOOP STRESS (INTERFACE)
  7.  RADIAL STRESS (INTERFACE)
  8.  HYDROSTATIC PRESSURE (INTERFACE)
  9.  LONGITUDINAL STRESS (INTERFACE)
 10.  LONGITUDINAL STRESS (AVERAGE)

       MISCELLANEOUS FUNCTIONS
 11.  LONGITUDINAL STRAIN (ASSEMBLAGE)
 12.  USER DEFINED CONSTRAINT FUNCTION

 ENTER NUMBER OF SELECTIONS  (ENTER 0 FOR NO CONSTRAINTS) -> 1
 ENTER FUNCTION  1 -> 12

 *****************************************************
 TO USE CONSTRAINT(S) IN SUBROUTINE EXTCONST: ENTER 1

 TO CHANGE CONSTRAINTS, EXIT SHELL PROGRAM, EDIT, COMPILE
 AND LINK SUBROUTINE EXTCONST TO INCLUDE YOUR
 CHOICE OF CONSTRAINT FUNCTION(S): ENTER 2
 *****************************************************
```

```
ENTER CHOICE -> 1

ENTER NUMBER OF CONSTRAINTS IN SUBROUTINE -> 2

       CONSTRAINTS:

       USER DEFINED CONSTRAINT FUNCTION(S)


       IS INFORMATION CORRECT? <Y/N> y

WOULD YOU LIKE TO SEE A PROBLEM REVIEW? <Y/N> y


******************************PROBLEM REVIEW*****************************

               ***CONCENTRIC CYLINDER GEOMETRY***

LAYER         MATERIAL       NORMALIZED OUTER RADIUS     VOLUME FRACTION

  1             FIBER                0.63246                  0.4000
  2          INTERFACE 1             0.65354                  0.0271
  3          INTERFACE 2             0.67462                  0.0280
  4          INTERFACE 3             0.69570                  0.0289
  5             MATRIX               1.00000                  0.5160

FIBER VOLUME FRACTION =        0.4000
INTERFACE VOLUME FRACTION =    0.0840
MATRIX VOLUME FRACTION =       0.5160

HIT RETURN TO CONTINUE ->

         ***LAMINATED CYLINDER MATERIAL SPECIFICATION***

    LAYER               INCLUSION             MATRIX               FVF

FIBER CORE                ------            SiC (SCS-6)           -----
INTERFACE LAYER  1      Gr                  Cu                    0.100
INTERFACE LAYER  2      Gr                  Cu                    0.100
INTERFACE LAYER  3      Gr                  Cu                    0.100
MATRIX LAYER              ------            Ti-24Al-11Nb          -----
HIT RETURN TO CONTINUE ->

                           ***LOAD HISTORY***

STEP  TIME   INCREMENTS  TEMPERATURE  PRESSURE   AXIAL FORCE  AXIAL STRAIN
 1                          815.0       0.00         0.00
      1.0000       791
 2                           24.0       0.00         0.00

            MAXIMUM NUMBER OF ITERATIONS =     50
            CONVERGENCE ERROR TOLERANCE = 0.01000
            CONVERGENCE INFORMATION WRITTEN TO optcomp2.conv
            OPTIMIZATION ITERATIONS WRITTEN TO SCREEN

HIT RETURN TO CONTINUE ->

                ***DESIGN VARIABLE SUMMARY***

            HETEROGENEOUS INTERFACE VOLUME FRACTION
          LAYER       LOWER BOUND      UPPER BOUND
           1             0.00             0.30
           2             0.00             0.30
           3             0.00             0.30
```

```
HIT RETURN TO CONTINUE ->

          OBJECTIVE FUNCTION

          MINIMIZATION OF THE
USER DEFINED OBJECTIVE FUNCTION


          CONSTRAINTS:
          USER DEFINED CONSTRAINT FUNCTION(S)



HIT RETURN TO CONTINUE ->

           ******MAIN MENU******

          1.  CREATE NEW DATA FILE
          2.  RUN EXISTING DATA FILE
          3.  ENTER NEW MATERIALS INTO DATABANK
          4.  EXIT SHELL
        ENTER CHOICE -> 2




***************LEGEND FOR DESIGN VARIABLES**********
 X 1 = INCLUSION VOLUME FRACTION FOR INTERFACE  1
 X 2 = INCLUSION VOLUME FRACTION FOR INTERFACE  2
 X 3 = INCLUSION VOLUME FRACTION FOR INTERFACE  3

 ITERATION #:     1
DESIGN VARIABLE    LOWER BOUND    CURRENT VALUE    UPPER BOUND    OBJECTIVE FN
     X 1              0.0000          0.1000          0.3000
     X 2              0.0000          0.1000          0.3000
     X 3              0.0000          0.1000          0.3000
                                                                    4.20569
 CONSTRAINT          VALUE        LIMIT
      1            USER DEFINED CONSTRAINT
      2            USER DEFINED CONSTRAINT
 ITERATION #:     2
DESIGN VARIABLE    LOWER BOUND    CURRENT VALUE    UPPER BOUND    OBJECTIVE FN
     X 1              0.0000          0.1000          0.3000
     X 2              0.0000          0.1000          0.3000
     X 3              0.0000          0.1000          0.3000
                                                                    4.20569
 CONSTRAINT          VALUE        LIMIT
      1            USER DEFINED CONSTRAINT
      2            USER DEFINED CONSTRAINT
 ITERATION #:     3
DESIGN VARIABLE    LOWER BOUND    CURRENT VALUE    UPPER BOUND    OBJECTIVE FN
     X 1              0.0000          0.1001          0.3000
     X 2              0.0000          0.1000          0.3000
     X 3              0.0000          0.1000          0.3000
                                                                    4.19312
 CONSTRAINT          VALUE        LIMIT
      1            USER DEFINED CONSTRAINT
      2            USER DEFINED CONSTRAINT
 ITERATION #:     4
DESIGN VARIABLE    LOWER BOUND    CURRENT VALUE    UPPER BOUND    OBJECTIVE FN
     X 1              0.0000          0.1000          0.3000
     X 2              0.0000          0.1001          0.3000
     X 3              0.0000          0.1000          0.3000
                                                                    4.21159
 CONSTRAINT          VALUE        LIMIT
      1            USER DEFINED CONSTRAINT
      2            USER DEFINED CONSTRAINT
```

```
 ITERATION #:     5
DESIGN VARIABLE    LOWER BOUND    CURRENT VALUE    UPPER BOUND    OBJECTIVE FN
      X 1            0.0000          0.1000           0.3000
      X 2            0.0000          0.1000           0.3000
      X 3            0.0000          0.1001           0.3000
                                                                    4.21159
  CONSTRAINT        VALUE       LIMIT
       1           USER DEFINED CONSTRAINT
       2           USER DEFINED CONSTRAINT
 ITERATION #:     6
DESIGN VARIABLE    LOWER BOUND    CURRENT VALUE    UPPER BOUND    OBJECTIVE FN
      X 1            0.0000          0.1023           0.3000
      X 2            0.0000          0.0989           0.3000
      X 3            0.0000          0.0989           0.3000
                                                                    3.80089
  CONSTRAINT        VALUE       LIMIT
       1           USER DEFINED CONSTRAINT
       2           USER DEFINED CONSTRAINT
 ITERATION #:     7
DESIGN VARIABLE    LOWER BOUND    CURRENT VALUE    UPPER BOUND    OBJECTIVE FN
      X 1            0.0000          0.1060           0.3000
      X 2            0.0000          0.0970           0.3000
      X 3            0.0000          0.0970           0.3000
                                                                    3.42422
  CONSTRAINT        VALUE       LIMIT
       1           USER DEFINED CONSTRAINT
       2           USER DEFINED CONSTRAINT
 ITERATION #:     8
DESIGN VARIABLE    LOWER BOUND    CURRENT VALUE    UPPER BOUND    OBJECTIVE FN
      X 1            0.0000          0.1156           0.3000
      X 2            0.0000          0.0922           0.3000
      X 3            0.0000          0.0922           0.3000
                                                                    2.73422
  CONSTRAINT        VALUE       LIMIT
       1           USER DEFINED CONSTRAINT
       2           USER DEFINED CONSTRAINT
 ITERATION #:     9
DESIGN VARIABLE    LOWER BOUND    CURRENT VALUE    UPPER BOUND    OBJECTIVE FN
      X 1            0.0000          0.1409           0.3000
      X 2            0.0000          0.0796           0.3000
      X 3            0.0000          0.0796           0.3000
                                                                    1.37205
  CONSTRAINT        VALUE       LIMIT
       1           USER DEFINED CONSTRAINT
       2           USER DEFINED CONSTRAINT
 ITERATION #:    10
DESIGN VARIABLE    LOWER BOUND    CURRENT VALUE    UPPER BOUND    OBJECTIVE FN
      X 1            0.0000          0.2070           0.3000
      X 2            0.0000          0.0465           0.3000
      X 3            0.0000          0.0465           0.3000
                                                                   10.09749
  CONSTRAINT        VALUE       LIMIT
       1           USER DEFINED CONSTRAINT
       2           USER DEFINED CONSTRAINT
       .
       .
       .
       .
       .
       .
       .
       .
 ITERATION #:    50
DESIGN VARIABLE    LOWER BOUND    CURRENT VALUE    UPPER BOUND    OBJECTIVE FN
      X 1            0.0000          0.1455           0.3000
      X 2            0.0000          0.1010           0.3000
      X 3            0.0000          0.0535           0.3000
                                                                    0.00147
  CONSTRAINT        VALUE       LIMIT
       1           USER DEFINED CONSTRAINT
       2           USER DEFINED CONSTRAINT
```

7.5.2 Results of the plastic strain optimization

The file **optcomp2.out**, containing information on the material properties of the fiber, interfacial layer(s) and matrix (or their constituents if these have been specified as heterogeneous), and initial and final (optimum) concentric cylinder make-up, load history, stresses and inelastic strains, for the data file **optcomp2.data** constructed in Section 7.5.1, is given below.

```
     NASA-LEWIS RESEARCH CENTER, STRUCTURES DIVISION, CLEVELAND OHIO
           ***************************************************
           ***                    OPTCOMP2                  ***
           ***                                              ***
           ***    CONCENTRIC CYLINDER OPTIMIZATION PROGRAM  ***
           *** FOR THE DETERMINATION OF IDEALIZED INTERFACE ***
           ***      MICROSTRUCTURE AND PROCESSING HISTORY   ***
           ***                                              ***
           ***                   WRITTEN BY                 ***
           ***                                              ***
           ***              ROBERT SCOTT SALZAR             ***
           ***              MAREK-JERZY PINDERA             ***
           ***                                              ***
           ***          THE UNIVERSITY OF VIRGINIA          ***
           ***                   JUNE 1995                  ***
           ***                                              ***
           ***     DEVELOPED FOR THE FATIGUE AND FRACTURE   ***
           ***      BRANCH OF NASA-LEWIS RESEARCH CENTER    ***
           ***           UNDER CONTRACT NAS3-26571          ***
           ***      DR. S. M. ARNOLD  (CONTRACT MONITOR)    ***
           ***************************************************


   Inelastic model (VPFLAG = 1) : Classical Plasticity

   Units in MPa, degree C, and seconds


   MATERIAL # 1


   TEMPERATURE =   0.8150E+03

   0.3999E+06  0.3999E+06  0.3999E+06
   0.2500E+00  0.2500E+00  0.2500E+00
   0.4499E-05  0.4499E-05  0.4499E-05
   0.6895E+05  0.3999E+06

    .
    .
    .

   TEMPERATURE =   0.2400E+02

   0.3999E+06  0.3999E+06  0.3999E+06
   0.2500E+00  0.2500E+00  0.2500E+00
   0.3528E-05  0.3528E-05  0.3528E-05
   0.6895E+05  0.3999E+06

    .
    .
    .
    .
    .
```

MATERIAL # 8

TEMPERATURE = 0.8150E+03

0.4281E+05	0.4281E+05	0.4281E+05
0.2600E+00	0.2600E+00	0.2600E+00
0.1107E-04	0.1107E-04	0.1107E-04
0.1658E+03	0.1107E+03	

.
.
.

TEMPERATURE = 0.2400E+02

0.1103E+06	0.1103E+06	0.1103E+06
0.2600E+00	0.2600E+00	0.2600E+00
0.9000E-05	0.9000E-05	0.9000E-05
0.3716E+03	0.2297E+05	

INITIAL LAMINATED CYLINDER CONFIGURATION

LAYER	OUTER RADIUS	INCLUSION	MATRIX	FVF	ASP. RATIO
1	0.6325	-----	1	----	----
2	0.6535	2	3	0.100	1.0000
3	0.6746	4	5	0.100	1.0000
4	0.6957	6	7	0.100	1.0000
5	1.0000	-----	8	----	----

INITIAL LOAD HISTORY

STEP	TIME	INCREMENTS	TEMPERATURE	PRESSURE	AXIAL STRESS	AXIAL STRAIN
1			815.00	0.00	0.00	
	1.0000	791				
2			24.00	0.00	0.00	

INITIAL STRESSES AND INELASTIC STRAINS

Time = 0.1000E+01
Temperature = 0.2400E+02
Radial traction = 0.0000E+00
Axial strain = -0.4300E-02
Axial stress = -0.4092E-04

RING NO.	RADIUS	STRXX	STRRR	STRTT	W
1	0.0000E+00	-0.5359E+03	-0.1199E+03	-0.1199E+03	0.0000E+00
1	0.6325E+00	-0.5359E+03	-0.1199E+03	-0.1199E+03	-0.1897E-02
2	0.6325E+00	0.7537E+02	-0.1199E+03	0.8260E+02	-0.1897E-02
2	0.6430E+00	0.7621E+02	-0.1166E+03	0.8005E+02	
2	0.6535E+00	0.7703E+02	-0.1135E+03	0.7762E+02	-0.2602E-02
3	0.6535E+00	0.7703E+02	-0.1135E+03	0.7762E+02	-0.2602E-02
3	0.6641E+00	0.7783E+02	-0.1105E+03	0.7527E+02	
3	0.6746E+00	0.7860E+02	-0.1076E+03	0.7303E+02	-0.3290E-02
4	0.6746E+00	0.7860E+02	-0.1076E+03	0.7303E+02	-0.3290E-02
4	0.6851E+00	0.7934E+02	-0.1048E+03	0.7088E+02	
4	0.6957E+00	0.8006E+02	-0.1022E+03	0.6881E+02	-0.3960E-02
5	0.6957E+00	0.3454E+03	-0.1022E+03	0.2620E+03	-0.3960E-02
5	0.7566E+00	0.3734E+03	-0.7328E+02	0.2514E+03	
5	0.8174E+00	0.3957E+03	-0.4954E+02	0.2396E+03	
5	0.8783E+00	0.4129E+03	-0.2992E+02	0.2275E+03	
5	0.9391E+00	0.4261E+03	-0.1362E+02	0.2158E+03	
5	0.1000E+01	0.4361E+03	0.6821E-12	0.2048E+03	-0.7140E-02

RING NO.	EPXXP	EPRRP	EPTTP	EPEFF	STREFF	SIGEFF
1	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.4160E+03	0.0000E+00
1	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.4160E+03	0.0000E+00
2	0.8386E-02	-0.1777E-01	0.9541E-02	0.1783E-01	0.1990E+03	
2	0.8378E-02	-0.1731E-01	0.9093E-02	0.1737E-01	0.1948E+03	
2	0.8370E-02	-0.1688E-01	0.8667E-02	0.1693E-01	0.1908E+03	
3	0.8370E-02	-0.1688E-01	0.8667E-02	0.1693E-01	0.1908E+03	
3	0.8363E-02	-0.1646E-01	0.8259E-02	0.1652E-01	0.1870E+03	
3	0.8355E-02	-0.1607E-01	0.7871E-02	0.1612E-01	0.1834E+03	
4	0.8355E-02	-0.1607E-01	0.7871E-02	0.1612E-01	0.1834E+03	
4	0.8348E-02	-0.1569E-01	0.7500E-02	0.1575E-01	0.1801E+03	
4	0.8342E-02	-0.1533E-01	0.7147E-02	0.1540E-01	0.1769E+03	
5	0.9076E-03	-0.1376E-02	0.4681E-03	0.1399E-02	0.4122E+03	0.4122E+03
5	0.6963E-03	-0.9393E-03	0.2430E-03	0.9751E-03	0.3999E+03	0.3999E+03
5	0.5226E-03	-0.6358E-03	0.1131E-03	0.6783E-03	0.3913E+03	0.3913E+03
5	0.3838E-03	-0.4264E-03	0.4262E-04	0.4697E-03	0.3852E+03	0.3852E+03
5	0.2752E-03	-0.2825E-03	0.7338E-05	0.3221E-03	0.3809E+03	0.3809E+03
5	0.1913E-03	-0.1833E-03	-0.7937E-05	0.2164E-03	0.3779E+03	0.3779E+03

FINAL LAMINATED CYLINDER CONFIGURATION

LAYER	OUTER RADIUS	INCLUSION	MATRIX	FVF	ASP. RATIO
1	0.6325	-----	1	----	----
2	0.6535	2	3	0.146	1.0000
3	0.6746	4	5	0.101	1.0000
4	0.6957	6	7	0.054	1.0000
5	1.0000	-----	8	----	----

FINAL LOAD HISTORY

STEP	TIME	INCREMENTS	TEMPERATURE	PRESSURE	AXIAL STRESS	AXIAL STRAIN
1			815.00	0.00	0.00	
	1.0000	791				
2			24.00	0.00	0.00	

FINAL STRESSES AND INELASTIC STRAINS

Time = 0.1000E+01
Temperature = 0.2400E+02
Radial traction = 0.0000E+00
Axial strain = -0.4300E-02
Axial stress = -0.4091E-04

RING NO.	RADIUS	STRXX	STRRR	STRTT	W
1	0.0000E+00	-0.5362E+03	-0.1201E+03	-0.1201E+03	0.0000E+00
1	0.6325E+00	-0.5362E+03	-0.1201E+03	-0.1201E+03	-0.1897E-02
2	0.6325E+00	0.9404E+02	-0.1201E+03	0.1024E+03	-0.1897E-02
2	0.6430E+00	0.9492E+02	-0.1165E+03	0.9949E+02	
2	0.6535E+00	0.9577E+02	-0.1131E+03	0.9671E+02	-0.2585E-02
3	0.6535E+00	0.7786E+02	-0.1131E+03	0.7861E+02	-0.2585E-02
3	0.6641E+00	0.7865E+02	-0.1101E+03	0.7626E+02	
3	0.6746E+00	0.7942E+02	-0.1072E+03	0.7401E+02	-0.3273E-02
4	0.6746E+00	0.6471E+02	-0.1072E+03	0.5991E+02	-0.3273E-02
4	0.6851E+00	0.6544E+02	-0.1046E+03	0.5802E+02	
4	0.6957E+00	0.6616E+02	-0.1022E+03	0.5621E+02	-0.3960E-02
5	0.6957E+00	0.3453E+03	-0.1022E+03	0.2620E+03	-0.3960E-02
5	0.7566E+00	0.3734E+03	-0.7328E+02	0.2514E+03	
5	0.8174E+00	0.3957E+03	-0.4954E+02	0.2396E+03	
5	0.8783E+00	0.4129E+03	-0.2992E+02	0.2275E+03	
5	0.9391E+00	0.4261E+03	-0.1362E+02	0.2158E+03	
5	0.1000E+01	0.4361E+03	0.7674E-12	0.2048E+03	-0.7140E-02

RING NO.	EPXXP	EPRRP	EPTTP	EPEFF	STREFF	SIGEFF
1	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.4160E+03	0.0000E+00
1	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.4160E+03	0.0000E+00
2	0.7993E-02	-0.1687E-01	0.9118E-02	0.1696E-01	0.2185E+03	
2	0.7984E-02	-0.1643E-01	0.8692E-02	0.1652E-01	0.2138E+03	
2	0.7976E-02	-0.1602E-01	0.8286E-02	0.1610E-01	0.2093E+03	
3	0.8361E-02	-0.1688E-01	0.8681E-02	0.1694E-01	0.1913E+03	
3	0.8353E-02	-0.1646E-01	0.8273E-02	0.1652E-01	0.1875E+03	
3	0.8346E-02	-0.1607E-01	0.7884E-02	0.1613E-01	0.1839E+03	
4	0.8714E-02	-0.1688E-01	0.8244E-02	0.1691E-01	0.1695E+03	
4	0.8707E-02	-0.1649E-01	0.7855E-02	0.1652E-01	0.1665E+03	
4	0.8701E-02	-0.1611E-01	0.7484E-02	0.1615E-01	0.1636E+03	
5	0.9073E-03	-0.1375E-02	0.4681E-03	0.1399E-02	0.4122E+03	0.4122E+03
5	0.6961E-03	-0.9390E-03	0.2430E-03	0.9748E-03	0.3999E+03	0.3999E+03
5	0.5224E-03	-0.6354E-03	0.1131E-03	0.6780E-03	0.3913E+03	0.3913E+03
5	0.3835E-03	-0.4261E-03	0.4261E-04	0.4694E-03	0.3852E+03	0.3852E+03
5	0.2749E-03	-0.2822E-03	0.7342E-05	0.3217E-03	0.3809E+03	0.3809E+03
5	0.1910E-03	-0.1830E-03	-0.7917E-05	0.2161E-03	0.3779E+03	0.3779E+03

The file **optcomp2.conv**, containing convergence messages at each optimization iteration, is given below.

```
OPTIMIZATION ITERATION #   1
ALL POINTS REACHED CONVERGENCE

.
.
.

OPTIMIZATION ITERATION #   5
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #   6
NON-CONVERGENCE AT FOLLOWING LOADING STATES

     Temperature =     756.000
     Radial traction =        0.000
     Average axial stress =       0.000
     Nonconvergence in ring number 2

OPTIMIZATION ITERATION #   7
ALL POINTS REACHED CONVERGENCE

.
.
.

OPTIMIZATION ITERATION #  17
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #  18

     Temperature =     743.000
     Radial traction =        0.000
     Average axial stress =       0.000
     Nonconvergence in ring number 2

OPTIMIZATION ITERATION #  19

     Temperature =     743.000
     Radial traction =        0.000
     Average axial stress =       0.000
     Nonconvergence in ring number 2
```

```
OPTIMIZATION ITERATION #  20

     Temperature =     743.000
     Radial traction =       0.000
     Average axial stress =       0.000
     Nonconvergence in ring number 2

OPTIMIZATION ITERATION #  21

     Temperature =     743.000
     Radial traction =       0.000
     Average axial stress =       0.000
     Nonconvergence in ring number 2

OPTIMIZATION ITERATION #  22
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #  23
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #  24

     Temperature =     761.000
     Radial traction =       0.000
     Average axial stress =       0.000
     Nonconvergence in ring number 4

OPTIMIZATION ITERATION #  25
ALL POINTS REACHED CONVERGENCE

.
.
.
.

OPTIMIZATION ITERATION #  33
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #  34

     Temperature =     745.000
     Radial traction =       0.000
     Average axial stress =       0.000
     Nonconvergence in ring number 2

OPTIMIZATION ITERATION #  35
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #  36

     Temperature =     745.000
     Radial traction =       0.000
     Average axial stress =       0.000
     Nonconvergence in ring number 2

OPTIMIZATION ITERATION #  37

     Temperature =     745.000
     Radial traction =       0.000
     Average axial stress =       0.000
     Nonconvergence in ring number 2

OPTIMIZATION ITERATION #  38
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #  39
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #  40
ALL POINTS REACHED CONVERGENCE
```

```
OPTIMIZATION ITERATION #  41
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #  42
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #  43

     Temperature =     754.000
     Radial traction =       0.000
     Average axial stress =       0.000
     Nonconvergence in ring number 3

OPTIMIZATION ITERATION #  44
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #  45

     Temperature =     745.000
     Radial traction =       0.000
     Average axial stress =       0.000
     Nonconvergence in ring number 2

OPTIMIZATION ITERATION #  46
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #  47
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #  48
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #  49
ALL POINTS REACHED CONVERGENCE

OPTIMIZATION ITERATION #  50
ALL POINTS REACHED CONVERGENCE
```

7.6 Appendix VI: Example 4 - Construction of a Material Property Databank

An example illustrating the entry of material properties for copper, modeled using the classical incremental plasticity theory, into the databank **class.data**, menu-driven by the user-friendly interface **shell.f**, is provided below. The text that appears in Courier-type capital letters is written to the screen at each step in the construction of the **fiber.data** file. User's responses to the menu-driven commands are shown in bold Courier-type letters.

```
NASA-LEWIS RESEARCH CENTER, STRUCTURES DIVISION, CLEVELAND OHIO
        ***************************************************
        ***                  OPTCOMP2                   ***
        ***                                             ***
        ***   CONCENTRIC CYLINDER OPTIMIZATION PROGRAM  ***
        *** FOR THE DETERMINATION OF IDEALIZED INTERFACE ***
        ***     MICROSTRUCTURE AND PROCESSING HISTORY   ***
        ***                                             ***
        ***                 WRITTEN BY                  ***
        ***                                             ***
        ***            ROBERT SCOTT SALZAR              ***
        ***            MAREK-JERZY PINDERA              ***
        ***                                             ***
        ***         THE UNIVERSITY OF VIRGINIA          ***
        ***                 JUNE 1995                   ***
        ***                                             ***
        ***    DEVELOPED FOR THE FATIGUE AND FRACTURE   ***
        ***     BRANCH OF NASA-LEWIS RESEARCH CENTER    ***
        ***          UNDER CONTRACT NAS3-26571          ***
        ***     DR. S. M. ARNOLD  (CONTRACT MONITOR)    ***
        ***************************************************


=================================LEGAL NOTICE================================

Neither NASA, nor any of its employees, contractors or grantees, nor any
person acting on behalf of either:
a.  makes any warranty or representation, expressed or implied, with respect
    to the accuracy, completeness, or usefulness of this software, or that the
    use of this software may not infringe privately owned rights; or

b.  assumes any liabilities with respect to the use of, or for damages
    resulting from use of this software.

=============================================================================


         ******MAIN MENU******

       1.  CREATE NEW DATA FILE
       2.  RUN EXISTING DATA FILE
       3.  ENTER NEW MATERIALS INTO DATABANK
       4.  EXIT SHELL
      ENTER CHOICE -> 3


            ***MATERIAL INPUT MENU***

    1.  ENTER NEW CLASSICAL PLASTICITY MATERIALS
    2.  ENTER NEW VISCOPLASTIC MATERIALS
    3.  ENTER NEW USER-DEFINED MODEL MATERIALS
    4.  RETURN TO MAIN MENU
```

```
    ENTER CHOICE -> 1

          ***CLASSICAL PLASTICITY MATERIAL DATABANK***

          AVAILABLE MATERIALS
        SiC (SCS-6)
        Al2O3
        Gr
        Ti-24Al-11Nb
        Ti-6Al-4V
        NiAl
        FeAl
        FeAl1
    1.  ENTER NEW MATERIAL
    2.  RETURN TO MATERIAL MENU

    ENTER CHOICE -> 1

*******************SPECIFY NEW MATERIAL PROPERTIES*******************

 IF ENTERING PROPERTIES IN SI UNITS (C, MPa), ENTER 1
 IF ENTERING PROPERTIES IN ENGLISH UNITS  (F, PSI), ENTER 2
 -> 1

 ENTER NEW MATERIAL NAME -> Cu
 ENTER NUMBER OF TEMPERATURES AT WHICH PROPERTIES
 WILL BE ENTERED (3 OR GREATER) -> 6

 IF MATERIAL IS ISOTROPIC, ENTER 1
 IF MATERIAL IS TRANSVERSELY ISOTROPIC, ENTER 2
 IF MATERIAL IS ORTHOTROPIC, ENTER 3
 ENTER CHOICE -> 1

 MATERIAL NAME IS Cu
 NUMBER OF INPUT TEMPERATURES IS    6
 MATERIAL IS ISOTROPIC

 IS INFORMATION CORRECT? <Y/N> y

 ***ENTER PROPERTIES WITH EITHER ASCENDING OR***
          ***DESCENDING TEMPERATURES***

 ENTER TEMPERATURE -> 815
 ENTER ELASTIC MODULUS (EXX) -> 14400
 ENTER POISSON'S (VXR) -> 0.38
 ENTER C.T.E. (ALFXX) -> 0.00002008
 ENTER YIELD POINT (Y) -> 19.6
 ENTER HARDENING SLOPE (HS) -> 900

 TEMP =                       815.0000
 EXX, ETT, ERR  =           0.1440D+05  0.1440D+05  0.1440D+05
 VXR, VXT, VRT  =               0.3800      0.3800      0.3800
 ALFXX, ALFTT, ALFRR =      0.2008D-04  0.2008D-04  0.2008D-04
 Y, HS =                    0.1960D+02  0.9000D+03

 IS INFORMATION CORRECT? <Y/N> y

 ENTER TEMPERATURE -> 760
 ENTER ELASTIC MODULUS (EXX) -> 16800
 ENTER POISSON'S (VXR) -> 0.38
 ENTER C.T.E. (ALFXX) -> 0.00001980
 ENTER YIELD POINT (Y) -> 20.0
 ENTER HARDENING SLOPE (HS) -> 980

 TEMP =                       760.0000
 EXX, ETT, ERR  =           0.1680D+05  0.1680D+05  0.1680D+05
 VXR, VXT, VRT  =               0.3800      0.3800      0.3800
 ALFXX, ALFTT, ALFRR =      0.1980D-04  0.1980D-04  0.1980D-04
 Y, HS =                    0.2000D+02  0.9800D+03

 IS INFORMATION CORRECT? <Y/N> y
```

```
ENTER TEMPERATURE -> 649
ENTER ELASTIC MODULUS (EXX) -> 23800
ENTER POISSON'S (VXR) -> 0.38
ENTER C.T.E. (ALFXX) -> 0.00001925
ENTER YIELD POINT (Y) -> 22.5
ENTER HARDENING SLOPE (HS) -> 1160

TEMP =                        649.0000
EXX, ETT, ERR  =            0.2380D+05  0.2380D+05  0.2380D+05
VXR, VXT, VRT  =                0.3800      0.3800      0.3800
ALFXX, ALFTT, ALFRR =       0.1925D-04  0.1925D-04  0.1925D-04
Y, HS =                     0.2250D+02  0.1160D+04

IS INFORMATION CORRECT? <Y/N> y

ENTER TEMPERATURE -> 427
ENTER ELASTIC MODULUS (EXX) -> 36800
ENTER POISSON'S (VXR) -> 0.38
ENTER C.T.E. (ALFXX) -> 0.00001836
ENTER YIELD POINT (Y) -> 26.7
ENTER HARDENING SLOPE (HS) -> 2380

TEMP =                        427.0000
EXX, ETT, ERR  =            0.3680D+05  0.3680D+05  0.3680D+05
VXR, VXT, VRT  =                0.3800      0.3800      0.3800
ALFXX, ALFTT, ALFRR =       0.1836D-04  0.1836D-04  0.1836D-04
Y, HS =                     0.2670D+02  0.2380D+04

IS INFORMATION CORRECT? <Y/N> y

ENTER TEMPERATURE -> 204
ENTER ELASTIC MODULUS (EXX) -> 58900
ENTER POISSON'S (VXR) -> 0.38
ENTER C.T.E. (ALFXX) -> 0.000017
ENTER YIELD POINT (Y) -> 31.6
ENTER HARDENING SLOPE (HS) -> 4270

TEMP =                        204.0000
EXX, ETT, ERR  =            0.5890D+05  0.5890D+05  0.5890D+05
VXR, VXT, VRT  =                0.3800      0.3800      0.3800
ALFXX, ALFTT, ALFRR =       0.1700D-04  0.1700D-04  0.1700D-04
Y, HS =                     0.3160D+02  0.4270D+04

IS INFORMATION CORRECT? <Y/N> y

ENTER TEMPERATURE -> 24
ENTER ELASTIC MODULUS (EXX) -> 78800
ENTER POISSON'S (VXR) -> 0.38
ENTER C.T.E. (ALFXX) -> 0.000016
ENTER YIELD POINT (Y) -> 37.1
ENTER HARDENING SLOPE (HS) -> 6370

TEMP =                         24.0000
EXX, ETT, ERR  =            0.7880D+05  0.7880D+05  0.7880D+05
VXR, VXT, VRT  =                0.3800      0.3800      0.3800
ALFXX, ALFTT, ALFRR =       0.1600D-04  0.1600D-04  0.1600D-04
Y, HS =                     0.3710D+02  0.6370D+04

IS INFORMATION CORRECT? <Y/N> y

          ***CLASSICAL PLASTICITY MATERIAL DATABANK***

          AVAILABLE MATERIALS
        SiC (SCS-6)
        Al2O3
        Gr
        Ti-24Al-11Nb
        Ti-6Al-4V
        NiAl
        FeAl
```

```
    FeAl1
    Cu
1.  ENTER NEW MATERIAL
2.  RETURN TO MATERIAL MENU

ENTER CHOICE -> 2

        ***MATERIAL INPUT MENU***

1.  ENTER NEW CLASSICAL PLASTICITY MATERIALS
2.  ENTER NEW VISCOPLASTIC MATERIALS
3.  ENTER NEW USER-DEFINED MODEL MATERIALS
4.  RETURN TO MAIN MENU

ENTER CHOICE -> 4


     ******MAIN MENU******

   1.  CREATE NEW DATA FILE
   2.  RUN EXISTING DATA FILE
   3.  ENTER NEW MATERIALS INTO DATABANK
   4.  EXIT SHELL
  ENTER CHOICE -> 4
```

7.6.1 Class.data file format

The material properties in the files **class.data**, **visco.data** and **user.data** are stored sequentially for reasons of internal bookkeeping. For each set of material properties entered at a number of different temperatures, the program automatically re-evaluates these properties at ten equally spaced temperatures using cubic splines. Thus if properties at six temperatures were entered by the user for a given material, these properties would subsequently be re-valuated at ten temperatures and stored sequentially, as illustrated below for the Cu properties stored in the file **class.data**. The user can edit these properties using any text editor, making sure that the logical organization of the file is not re-arranged. To delete a material, the entire block (all ten temperature points) must be deleted, leaving no blank lines between material sets.

```
Cu                     Material name
1                      Internal flag indicating material's orthotropy
      815.00000        Temperature # 1
0.144000D+05           EXX
0.144000D+05           ETT
0.144000D+05           ERR
0.380000D+00           VXR
0.380000D+00           VXT
0.380000D+00           VRT
0.200800D-04           ALFXX
0.200800D-04           ALFTT
0.200800D-04           ALFRR
0.196000D+02           Y
0.900000D+03           HS
      727.11111        Temperature # 2
0.186611D+05           e
0.186611D+05           t
0.186611D+05           c
0.380000D+00           .
0.380000D+00           .
0.380000D+00           .
0.196305D-04
0.196305D-04
0.196305D-04
0.205500D+02
0.102088D+04
       .
       .
       .
       .
       .
       24.00000        Temperature # 10
0.788000D+05           e
0.788000D+05           t
0.788000D+05           c
0.380000D+00           .
0.380000D+00           .
0.380000D+00           .
0.160000D-04
0.160000D-04
0.160000D-04
0.371000D+02
0.637000D+04
```

7.7 Appendix VII: RTSHELL2 Example

7.7.1 Construction and execution of the rtshell2.data file

The construction and subsequent execution of the **rtshell2.data** file, menu-driven by a user-friendly interface embedded in **RTSHELL2,** is illustrated below. The input data is identical to that provided in Example 3, excluding the specification of optimization parameters. The text that appears in Courier-type capital letters is written to the screen at each step in the construction of the **rtshell2.data** file. User's responses to the menu-driven commands are shown in bold Courier-type letters.

```
NASA-LEWIS RESEARCH CENTER, STRUCTURES DIVISION, CLEVELAND OHIO
     ****************************************************
     ***                    OPTCOMP2                  ***
     ***                                              ***
     ***   CONCENTRIC CYLINDER OPTIMIZATION PROGRAM   ***
     *** FOR THE DETERMINATION OF IDEALIZED INTERFACE ***
     ***     MICROSTRUCTURE AND PROCESSING HISTORY    ***
     ***                                              ***
     ***                   WRITTEN BY                 ***
     ***                                              ***
     ***              ROBERT SCOTT SALZAR             ***
     ***              MAREK-JERZY PINDERA             ***
     ***                                              ***
     ***          THE UNIVERSITY OF VIRGINIA          ***
     ***                   JUNE 1995                  ***
     ***                                              ***
     ***    DEVELOPED FOR THE FATIGUE AND FRACTURE    ***
     ***     BRANCH OF NASA-LEWIS RESEARCH CENTER     ***
     ***          UNDER CONTRACT NAS3-26571           ***
     ***     DR. S. M. ARNOLD  (CONTRACT MONITOR)     ***
     ****************************************************

==================================LEGAL NOTICE================================

Neither NASA, nor any of its employees, contractors or grantees, nor any
person acting on behalf of either:
a.  makes any warranty or representation, expressed or implied, with respect
    to the accuracy, completeness, or usefulness of this software, or that the
    use of this software may not infringe privately owned rights; or

b.  assumes any liabilities with respect to the use of, or for damages
    resulting from use of this software.

=============================================================================

 HIT RETURN TO CONTINUE ->

          ******MAIN MENU******

        1.  CREATE NEW DATA FILE
        2.  RUN EXISTING DATA FILE
        3.  ENTER NEW MATERIALS INTO DATABANK
        4.  EXIT SHELL
       ENTER CHOICE -> 1

 *********************************BLOCK 1**********************************
              SPECIFY CONCENTRIC CYLINDER GEOMETRY, MATERIALS
                            INCLUDING PROPERTIES
```

```
DOES THE FIBER HAVE LAYERED MORPHOLOGY? <Y/N> n
ENTER RADIUS OF FIBER CORE -> 0.63246
ENTER FIBER VOLUME FRACTION -> 0.40

ENTER THE NUMBER OF INTERFACE LAYERS -> 3

ENTER THICKNESS OF INTERFACE 1 -> 0.02108
ENTER THICKNESS OF INTERFACE 2 -> 0.02108
ENTER THICKNESS OF INTERFACE 3 -> 0.02108


          ***CONCENTRIC CYLINDER GEOMETRY***

 LAYER       MATERIAL      NORMALIZED OUTER RADIUS     VOLUME FRACTION

   1          FIBER                0.63246                 0.4000
   2        INTERFACE 1            0.65354                 0.0271
   3        INTERFACE 2            0.67462                 0.0280
   4        INTERFACE 3            0.69570                 0.0289
   5          MATRIX               1.00000                 0.5160

FIBER VOLUME FRACTION =        0.4000
INTERFACE VOLUME FRACTION =    0.0840
MATRIX VOLUME FRACTION =       0.5160

  IS INFORMATION CORRECT? <Y/N> y

      ***COMPOSITE MICRO-STRUCTURE***


    IF FIBER CORE IS HOMOGENEOUS,    ENTER 1
    IF FIBER CORE IS HETEROGENEOUS, ENTER 2
    -> 1

    IF INTERFACE LAYER  1 IS HOMOGENEOUS,   ENTER 1
    IF INTERFACE LAYER  1 IS HETEROGENEOUS, ENTER 2
    -> 2
    IF INTERFACE LAYER  2 IS HOMOGENEOUS,   ENTER 1
    IF INTERFACE LAYER  2 IS HETEROGENEOUS, ENTER 2
    -> 2
    IF INTERFACE LAYER  3 IS HOMOGENEOUS,   ENTER 1
    IF INTERFACE LAYER  3 IS HETEROGENEOUS, ENTER 2
    -> 2

    IF MATRIX LAYER IS HOMOGENEOUS,   ENTER 1
    IF MATRIX LAYER IS HETEROGENEOUS, ENTER 2
    -> 1

***INELASTIC CONSTITUTIVE MODEL SELECTION***

     FOR CLASSICAL PLASTICITY, ENTER 1
     FOR BODNER-PARTOM,        ENTER 2
     FOR USER-DEFINED MODEL,   ENTER 3
     -> 1

          ***MATERIAL PROPERTY SELECTION***

     AVAILABLE MATERIALS           AVAILABLE CONSTITUTIVE MODELS
  1   SiC (SCS-6)                        1  ELASTIC
  2   Al2O3                              2  PLASTIC
  3   Gr
  4   Ti-24Al-11Nb
  5   Ti-6Al-4V
  6   NiAl
  7   FeAl
  8   FeAl1
  9   Cu
 10   ENTER NEW MATERIAL
```

```
ENTER MATERIAL FOR FIBER CORE -> 1

ENTER INCLUSION PHASE MATERIAL FOR INTERFACE LAYER 1 -> 3
ENTER CONSTITUTIVE MODEL FOR INCLUSION PHASE  -> 1
ENTER MATRIX PHASE MATERIAL FOR INTERFACE LAYER 1 -> 9
ENTER CONSTITUTIVE MODEL FOR MATRIX PHASE MATERIAL -> 2
ENTER INCLUSION VOLUME FRACTION FOR INTERFACE LAYER 1 -> 0.10
ENTER ASPECT RATIO FOR INCLUSION -> 1.0

ENTER INCLUSION PHASE MATERIAL FOR INTERFACE LAYER 2 -> 3
ENTER CONSTITUTIVE MODEL FOR INCLUSION PHASE  -> 1
ENTER MATRIX PHASE MATERIAL FOR INTERFACE LAYER 2 -> 9
ENTER CONSTITUTIVE MODEL FOR MATRIX PHASE MATERIAL -> 2
ENTER INCLUSION VOLUME FRACTION FOR INTERFACE LAYER 2 -> 0.10
ENTER ASPECT RATIO FOR INCLUSION -> 1.0

ENTER INCLUSION PHASE MATERIAL FOR INTERFACE LAYER 3 -> 3
ENTER CONSTITUTIVE MODEL FOR INCLUSION PHASE  -> 1
ENTER MATRIX PHASE MATERIAL FOR INTERFACE LAYER 3 -> 9
ENTER CONSTITUTIVE MODEL FOR MATRIX PHASE MATERIAL -> 2
ENTER INCLUSION VOLUME FRACTION FOR INTERFACE LAYER 3 -> 0.10
ENTER ASPECT RATIO FOR INCLUSION -> 1.0

ENTER MATERIAL FOR MATRIX LAYER -> 4
ENTER CONSTITUTIVE MODEL FOR MATRIX LAYER  -> 2


         ***CONCENTRIC CYLINDER MATERIAL SPECIFICATION***

    LAYER               INCLUSION            MATRIX             FVF

 FIBER CORE               ------           SiC (SCS-6)          -----
 INTERFACE LAYER  1     Gr                 Cu                   0.100
 INTERFACE LAYER  2     Gr                 Cu                   0.100
 INTERFACE LAYER  3     Gr                 Cu                   0.100
 MATRIX LAYER             ------           Ti-24Al-11Nb         -----

        IS INFORMATION CORRECT? <Y/N> y


 **********************************BLOCK 2*********************************
                 DEFINE PROCESSING/LOAD HISTORY, INCREMENT,
                              AND ITERATIONS


                               ***CAUTION***
THE APPLIED TEMPERATURE LOAD MUST REMAIN BETWEEN   24.00 deg AND  815.00 deg
                               *************

 NUMBER OF LOAD SEGMENTS -> 1

 IF FIRST LOAD SEGMENT IS UNDER STRESS CONTROL, ENTER 1
 IF FIRST LOAD SEGMENT IS UNDER STRAIN CONTROL, ENTER 2
 -> 1

 INITIAL TEMPERATURE, INITIAL EXTERNAL PRESSURE, INITIAL AXIAL STRESS
 -> 815 0 0
 DURATION OF LOAD STEP,  NUMBER OF LOAD INCREMENTS
 -> 1 791
 ENDING TEMPERATURE, ENDING PRESSURE, ENDING AXIAL STRESS
 -> 24 0 0


                              ***LOAD HISTORY***

 STEP  TIME  INCREMENTS  TEMPERATURE  PRESSURE   AXIAL STRESS  AXIAL STRAIN
   1                        815.0       0.00          0.00
     1.000       791
   2                         24.0       0.00          0.00
```

IS INFORMATION CORRECT? <Y/N> **y**

SET INTERNAL VARIABLES

CHANGE MAXIMUM NUMBER OF ITERATIONS (DEFAULT=10)? <Y/N> **y**
MAXIMUM NUMBER OF ITERATIONS -> **50**

CHANGE CONVERGENCE ERROR TOLERANCES (DEFAULT=0.01)? <Y/N> **n**

CHANGE NUMBER OF INTEGRATION POINTS (DEFAULT= 21/LAYER) AND
PRINT OPTIONS (DEFAULT =2/LAYER) FROM DEFAULT VALUES? <Y/N> **y**

	INT. POINTS	PRINT NUMBER
LAYER 1	**2**	**2**
LAYER 2	**21**	**3**
LAYER 3	**21**	**3**
LAYER 4	**21**	**3**
LAYER 5	**151**	**6**

WRITE CONVERGENCE INFORMATION TO rtshell2.conv FILE? <Y/N> **y**

INTERNAL VARIABLE REVIEW

MAXIMUM NUMBER OF ITERATIONS = 50
CONVERGENCE ERROR TOLERANCE = 0.01000
CONVERGENCE INFORMATION WRITTEN TO rtshell2.conv
OPTIMIZATION ITERATIONS SUPPRESSED FROM SCREEN

IS INFORMATION CORRECT? <Y/N> **y**
WOULD YOU LIKE TO SEE A PROBLEM REVIEW? <Y/N> **y**

*******************************PROBLEM REVIEW******************************

CONCENTRIC CYLINDER GEOMETRY

LAYER	MATERIAL	NORMALIZED OUTER RADIUS	VOLUME FRACTION
1	FIBER	0.63246	0.4000
2	INTERFACE 1	0.65354	0.0271
3	INTERFACE 2	0.67462	0.0280
4	INTERFACE 3	0.69570	0.0289
5	MATRIX	1.00000	0.5160

FIBER VOLUME FRACTION = 0.4000
INTERFACE VOLUME FRACTION = 0.0840
MATRIX VOLUME FRACTION = 0.5160

HIT RETURN TO CONTINUE ->

CONCENTRIC CYLINDER MATERIAL SPECIFICATION

LAYER	INCLUSION	MATRIX	FVF
FIBER CORE	------	SiC (SCS-6)	-----
INTERFACE LAYER 1	Gr	Cu	0.100
INTERFACE LAYER 2	Gr	Cu	0.100
INTERFACE LAYER 3	Gr	Cu	0.100
MATRIX LAYER	------	Ti-24Al-11Nb	-----

HIT RETURN TO CONTINUE ->

LOAD HISTORY

STEP	TIME	INCREMENTS	TEMPERATURE	PRESSURE	AXIAL STRESS	AXIAL STRAIN
1			815.0	0.00	0.00	
	1.000	791				

```
2                              24.0       0.00            0.00

             MAXIMUM NUMBER OF ITERATIONS =     50
             CONVERGENCE ERROR TOLERANCE = 0.01000
             CONVERGENCE INFORMATION WRITTEN TO rtshell2.conv
             OPTIMIZATION ITERATIONS SUPPRESSED FROM SCREEN

HIT RETURN TO CONTINUE ->

         ******MAIN MENU******

       1.  CREATE NEW DATA FILE
       2.  RUN EXISTING DATA FILE
       3.  ENTER NEW MATERIALS INTO DATABANK
       4.  EXIT SHELL
      ENTER CHOICE -> 2
```

7.7.2 Results

The file **rtshell2.out**, containing information on the geometry, material properties, stresses and inelastic strains in each layer of the specified concentric cylinder assemblage, for the data file **rtshell2.data** constructed in Section 7.7.1, is given below.

```
NASA-LEWIS RESEARCH CENTER, STRUCTURES DIVISION, CLEVELAND OHIO
     ***************************************************
     ***                 RTSHELL2                    ***
     ***                                             ***
     ***        CONCENTRIC CYLINDER PROGRAM          ***
     ***      FOR THE DETERMINATION OF EFFECT OF     ***
     ***     MICROSTRUCTURE AND PROCESSING HISTORY   ***
     ***                                             ***
     ***                WRITTEN BY                   ***
     ***                                             ***
     ***            ROBERT SCOTT SALZAR              ***
     ***            MAREK-JERZY PINDERA              ***
     ***                                             ***
     ***        THE UNIVERSITY OF VIRGINIA           ***
     ***                JUNE  1995                   ***
     ***                                             ***
     ***    DEVELOPED FOR THE FATIGUE AND FRACTURE   ***
     ***     BRANCH OF NASA-LEWIS RESEARCH CENTER    ***
     ***         UNDER CONTRACT NAS3-26571           ***
     ***     DR. S. M. ARNOLD  (CONTRACT MONITOR)    ***
     ***************************************************

Inelastic model (VPFLAG = 1) : Classical Plasticity

Units in MPa, degree C, and seconds


MATERIAL # 1

TEMPERATURE =  0.8150E+03

0.3999E+06  0.3999E+06  0.3999E+06
0.2500E+00  0.2500E+00  0.2500E+00
0.4499E-05  0.4499E-05  0.4499E-05
0.6895E+05  0.3999E+06

.
.
.

TEMPERATURE =  0.2400E+02

0.3999E+06  0.3999E+06  0.3999E+06
0.2500E+00  0.2500E+00  0.2500E+00
0.3528E-05  0.3528E-05  0.3528E-05
0.6895E+05  0.3999E+06

.
.
.
.
.
```

MATERIAL # 8

TEMPERATURE = 0.8150E+03

0.4281E+05	0.4281E+05	0.4281E+05
0.2600E+00	0.2600E+00	0.2600E+00
0.1107E-04	0.1107E-04	0.1107E-04
0.1658E+03	0.1107E+03	

.
.
.

TEMPERATURE = 0.2400E+02

0.1103E+06	0.1103E+06	0.1103E+06
0.2600E+00	0.2600E+00	0.2600E+00
0.9000E-05	0.9000E-05	0.9000E-05
0.3716E+03	0.2297E+05	

LAMINATED CYLINDER CONFIGURATION

LAYER	OUTER RADIUS	INCLUSION	MATRIX	FVF	ASP. RATIO
1	0.6325	-----	1	----	----
2	0.6535	2	3	0.100	1.0000
3	0.6746	4	5	0.100	1.0000
4	0.6957	6	7	0.100	1.0000
5	1.0000	-----	8	----	----

LOAD HISTORY

STEP	TIME	INCREMENTS	TEMPERATURE	PRESSURE	AXIAL STRESS	AXIAL STRAIN
1			815.00	0.00	0.00	
	1.000	791				
2			24.00	0.00	0.00	

STRESSES AND INELASTIC STRAINS

Time = 0.1000E+01
Temperature = 0.2400E+02
Radial traction = 0.0000E+00
Axial strain = -0.4300E-02
Axial stress = -0.4092E-04

RING NO.	RADIUS	STRXX	STRRR	STRTT	W
1	0.0000E+00	-0.5359E+03	-0.1199E+03	-0.1199E+03	0.0000E+00
1	0.6325E+00	-0.5359E+03	-0.1199E+03	-0.1199E+03	-0.1897E-02
2	0.6325E+00	0.7537E+02	-0.1199E+03	0.8260E+02	-0.1897E-02
2	0.6430E+00	0.7621E+02	-0.1166E+03	0.8005E+02	
2	0.6535E+00	0.7703E+02	-0.1135E+03	0.7762E+02	-0.2602E-02
3	0.6535E+00	0.7703E+02	-0.1135E+03	0.7762E+02	-0.2602E-02
3	0.6641E+00	0.7783E+02	-0.1105E+03	0.7527E+02	
3	0.6746E+00	0.7860E+02	-0.1076E+03	0.7303E+02	-0.3290E-02
4	0.6746E+00	0.7860E+02	-0.1076E+03	0.7303E+02	-0.3290E-02
4	0.6851E+00	0.7934E+02	-0.1048E+03	0.7088E+02	
4	0.6957E+00	0.8006E+02	-0.1022E+03	0.6881E+02	-0.3960E-02
5	0.6957E+00	0.3454E+03	-0.1022E+03	0.2620E+03	-0.3960E-02
5	0.7566E+00	0.3734E+03	-0.7328E+02	0.2514E+03	
5	0.8174E+00	0.3957E+03	-0.4954E+02	0.2396E+03	
5	0.8783E+00	0.4129E+03	-0.2992E+02	0.2275E+03	
5	0.9391E+00	0.4261E+03	-0.1362E+02	0.2158E+03	
5	0.1000E+01	0.4361E+03	0.6821E-12	0.2048E+03	-0.7140E-02

RING NO.	EPXXP	EPRRP	EPTTP	EPEFF	STREFF	SIGEFF
1	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.4160E+03	0.0000E+00
1	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.4160E+03	0.0000E+00
2	0.8386E-02	-0.1777E-01	0.9541E-02	0.1783E-01	0.1990E+03	
2	0.8378E-02	-0.1731E-01	0.9093E-02	0.1737E-01	0.1948E+03	
2	0.8370E-02	-0.1688E-01	0.8667E-02	0.1693E-01	0.1908E+03	
3	0.8370E-02	-0.1688E-01	0.8667E-02	0.1693E-01	0.1908E+03	
3	0.8363E-02	-0.1646E-01	0.8259E-02	0.1652E-01	0.1870E+03	
3	0.8355E-02	-0.1607E-01	0.7871E-02	0.1612E-01	0.1834E+03	
4	0.8355E-02	-0.1607E-01	0.7871E-02	0.1612E-01	0.1834E+03	
4	0.8348E-02	-0.1569E-01	0.7500E-02	0.1575E-01	0.1801E+03	
4	0.8342E-02	-0.1533E-01	0.7147E-02	0.1540E-01	0.1769E+03	
5	0.9076E-03	-0.1376E-02	0.4681E-03	0.1399E-02	0.4122E+03	0.4122E+03
5	0.6963E-03	-0.9393E-03	0.2430E-03	0.9751E-03	0.3999E+03	0.3999E+03
5	0.5226E-03	-0.6358E-03	0.1131E-03	0.6783E-03	0.3913E+03	0.3913E+03
5	0.3838E-03	-0.4264E-03	0.4262E-04	0.4697E-03	0.3852E+03	0.3852E+03
5	0.2752E-03	-0.2825E-03	0.7338E-05	0.3221E-03	0.3809E+03	0.3809E+03
5	0.1913E-03	-0.1833E-03	-0.7937E-05	0.2164E-03	0.3779E+03	0.3779E+03

The file **rtshell2.conv**, containing convergence messages, is given below.

```
ALL POINTS REACHED CONVERGENCE
```

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. AGENCY USE ONLY *(Leave blank)*

2. REPORT DATE
September 1996

3. REPORT TYPE AND DATES COVERED
Final Contractor Report

4. TITLE AND SUBTITLE
Optimization of Residual Stresses in MMC's Through Process Parameter Control and the Use of Heterogeneous Compensating/Compliant Interfacial Layers
OPTCOMP2 User's Guide

5. FUNDING NUMBERS
WU–505–65–12
C–NAS3–26571

6. AUTHOR(S)
Marek-Jerzy Pindera and Robert S. Salzar

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
University of Virginia
Civil Engineering & Applied Mechanics Department
Charlottesville, Virginia 22903

8. PERFORMING ORGANIZATION REPORT NUMBER
E–10314

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio 44135–3191

10. SPONSORING/MONITORING AGENCY REPORT NUMBER
NASA CR–198500

11. SUPPLEMENTARY NOTES
Marek-Jerzy Pindera, University of Virginia; Robert S. Salzar, University of Virginia, presently an NRC Fellow at NASA Lewis Research Center. Project Manager, Steven M. Arnold, organization code 5220, (216) 433–3334.

12a. DISTRIBUTION/AVAILABILITY STATEMENT
Unclassified - Unlimited
Subject Category 24

This publication is available from the NASA Center for AeroSpace Information, (301) 621–0390.

12b. DISTRIBUTION CODE

13. ABSTRACT ***(Maximum 200 words)***

A user's guide for the computer program **OPTCOMP2** is presented in this report. This program provides a capability to optimize the fabrication or service-induced residual stresses in unidirectional metal matrix composites subjected to combined thermomechanical axisymmetric loading by altering the processing history, as well as through the microstructural design of interfacial fiber coatings. The user specifies the initial architecture of the composite and the load history, with the constituent materials being elastic, plastic, viscoplastic, or as defined by the "user-defined" constitutive model, in addition to the objective function and constraints, through a user-friendly data input interface. The optimization procedure is based on an efficient solution methodology for the inelastic response of a fiber/interface layer(s)/matrix concentric cylinder model where the interface layers can be either homogeneous or heterogeneous. The response of heterogeneous layers is modeled using Aboudi's three-dimensional *method of cells* micromechanics model. The commercial optimization package **DOT** is used for the nonlinear optmization problem. The solution methodology for the arbitrarily layered cylinder is based on the *local-global stiffness matrix formulation* and Mendelson's iterative technique of *successive elastic solutions* developed for elastoplastic boundary-value problems. The optimization algorithm employed in **DOT** is based on the *method of feasible directions.*

14. SUBJECT TERMS
Optimization; Micromechanics; Residual stresses; Interface layers; Concentric cylinder model; Metal matrix composite

15. NUMBER OF PAGES
108

16. PRICE CODE
A06

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
Unclassified	Unclassified	Unclassified	